TOOL & DIE MAKER
(Dies & Moulds)

(Common for Press Tools, Jigs & Fixtures)

NSQF LEVEL - 4

1st Year

TRADE THEORY

SECTOR : CAPITAL GOODS AND MANUFACTURING

(As per revised syllabus July 2022 - 1200 of hrs)

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 & Manufacturing

Duration : 2 Years

Trades : Tool & Die Maker (Dies & Moulds) - 1st Year - Trade Theory - NSQF Level - 4 (Revised 2022)

Developed & Published by

National Instructional Media Institute
Post Box No.3142
Guindy, Chennai - 600 032
INDIA
Email: chennai-nimi@nic.in
Website: www.nimi.gov.in

First Edition: November 2022 Copies : 500

Rs.390/-

FOREWORD

The Government of India has set an ambitious target of imparting skills to 30 crores people, one out of every four Indians, by 2020 to help them secure jobs as part of the National Skills Development Policy. Industrial Training Institutes (ITIs) play a vital role in this process especially in terms of providing skilled manpower. Keeping this in mind, and for providing the current industry relevant skill training to Trainees, ITI syllabus has been recently updated with the help of 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 **Tool & Die Maker (Dies & Moulds) - Trade Theory - 1st Year (Revised 2022)** in **Capital Goods & Manufacturing 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 Executive Director & Staff of NIMI and members of Media Development Committee deserve appreciation for their contribution in bringing out this publication.

Jai Hind

Addl.Secretary / Director General (Training)
Ministry of Skill Development & Entrepreneurship,
Government of India.

New Delhi - 110 001

PREFACE

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

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

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

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

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

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

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

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 organisation to bring out this IMP for the trade of **Tool & Die Maker (Dies & Moulds) - 1st Year Trade Theory - NSQF Level - 4** (Revised 2022) under the **Capital Goods & Manufacturing** Sector for ITIs.

MEDIA DEVELOPMENT COMMITTEE MEMBERS

Shri. A. Vijayaraghavan	_	Assistant Director of Training (Retd.), ATI, Chennai - 32.
Shri. M. Sampath	_	Training Officer (Retd.), CTI, Chennai - 32.
Shri. R. Nirmal Raja	_	Assistant Professor, SRM Institute of science and Technology Ramapuram Campus, Chennai.
Shri. A.Srinivasan	_	Consultant Solid Works, NIMI Chennai

NIMI - COORDINATORS

Shri. Nirmalya Nath	_	Deputy Director, 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 intented to be used in workshop . It consists of a series of practical exercises to be completed by the trainees during the one year course of the **Tool & Die Maker (Dies & Moulds)** Trade supplemented and supported by instructions/ informations to assist in performing the exercises. These exercises are designed to ensure that all the skills in compliance with NSQF LEVEL - 5

The manual is divided into SIX modules.

Module 1	-	**Safety**
Module 2	-	**Fitting**
Module 3	-	**Turning**
Module 4	-	**Milling**
Module 5	-	**Grinding**
Module 6	-	**AutoCad & Pro - E**

The skill training in the shop floor is planned through a series of practical exercises centred around ome 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 one year course of the Tool & Die Maker Trade. The contents are sequenced according to the practical exercise contained in the manual on Trade practical. Attempt has been made to relate the theoretical aspects with the skill covered in each exercise to the extent possible. This co-relation 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 indicating about the corresponding practical exercise are given in every sheet of this manual.

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

The material is not 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

Sl.No.	Learning /Outcome	Refer Ex:No
1	Plan and organize the work to make job as per specification applying different types of basic fitting operation and check for dimensional accuracy following safety precautions. [Basic fitting operation – Filing, Marking, Hack sawing, Drilling, Taping, chipping and Grinding etc. Accuracy: ± 0.1mm]	1.1.01 - 1.2.27
2	Make different fit of components for assembling as per required tolerance observing principle of interchangeability and check for functionality. [Different Fit –Open, Angular, & Square Fit; Required tolerance: ±0.05 mm, angular tolerance: 1 degree.]	1.2.28 - 1.3.30
3	Set different shaped jobs on different chuck and demonstrate conventional lathe machine operation observing standard operation practice. [Different chucks:3 jaws & 4 jaws, different shaped jobs: round, square, hexagonal]	1.2.31 - 1.3.33
4	Prepare different cutting tool to produce jobs to appropriate accuracy by performing different turning operations. Different cutting tool – V tool, side cutting, parting, thread cutting (both LH & RH),Appropriate accuracy: ±0.06mm, Different turning operation – Plain, facing, drilling, boring (counter & stepped), grooving, Parallel Turning, Step Turning, parting, chamfering, U -cut, Reaming, internal recess, knurling.	1.3.34 - 1.3.43
5	Set the different machining parameters to produce threaded components applying method/ technique and test for proper assembly of the components with an accuracy of ± 0.05 mm. [Different threads viz., metric/ BSW/ Square]	1.3.44 - 1.3.45
6	Set the different machining parameters and cutters to prepare job by performing different milling operation and indexing. [Different machining parameters – feed, speed and depth of cut. Different milling operations – plain, stepped, angular, dovetail, T-slot, contour, gear milling]	1.3.46 - 1.4.62
7	Produce components of high accuracy by surface grinding operation. [Accuracy of +/- 0.02 mm]	1.4.63 - 1.5.72
8	Produce components of high accuracy by cylindrical grinding operations. [Accuracy of +/- 0.02mm.]	1.5.73 - 1.5.77
9	Sharpen different cutter or multipoint cutting tool. [Different cutters – end mill cutter, side & face milling cutter, single angle cutter, Reamer]	1.5.78
10	Develop isometric drawing and solid modelling of mould using CAD & Pro-E.	1.5.79 - 1.6.83

SYLLABUS

Duration	Reference Learning outcome	Professional Skills (Trade Practical) With Indicative Hours	Professional Knowledge (Trade Theory)
Professional Skill 138 Hrs.; Professional Knowledge 40 Hrs.	Plan and organize the work to make job as per specification applying different types of basic fitting operation and check for dimensional accuracy following safety precautions. [Basic fitting operation – Filing, Marking, Hack sawing, Drilling, Taping, chipping and Grinding etc. Accuracy: ± 0.1mm]. NOS:CSC/N0308	1. Introduction of trade skill and work application. (02hrs.) 2. Safety attitude development of the trainee by educating them to use Personal Protective Equipment (PPE). (02hrs.) 3. First Aid Method and basic training. (02hrs.) 4. Safe disposal of waste materials like cotton waste, metal chips/ burrs etc. (01hrs.) 5. Hazard identification and avoidance. (02hrs.) 6. Identification of safety signs for Danger, Warning, caution & personal safety message. (01 hr.) 7. Preventive measures for electrical accidents & steps to be taken in such accidents. (02hrs.) 8. Use of Fire extinguishers. (05hrs.) 9. Practice and understand precautions to be followed while working in fitting jobs. (02 hrs.) 10. Importance of trade training, List of tools & Machinery used in the trade. (01 hr.) 11. Safe use of tools and equipments used in the trade. (01 hr.) 12. Knowing games and memory training. (10 hrs.) 13. Motivational talk by experts. (02hrs.) 14. 5S training. (02 hrs.)	All necessary guidance to be provided to the newcomers to become familiar with the working of Industrial Training Institute system including store's procedures. Safe working practices. Soft Skills, its importance and Job area after completion of training.Importance of safety and general precautions observed in the industryshop floor. Introduction of First aid. Operation of electrical mains and electrical safety. Introduction of PPEs. Response to emergencies e.g. power failure, fire, and system failure.mportance of housekeeping & good shop floor practices. Introduction to 5S concept& its application.Occupational Safety &Health:Health, Safety and Environment guidelines, legislations & regulations as applicable. (08 hrs)
		15. Identification of tools &equipments as per desired specifications for filing and marking, visual inspection of raw material for rusting, scaling, corrosion etc. (03 hrs.) 16. Familiarisation of bench vice. (01hr.) 17. Filing- files different sector and measure with steel rule. (25 hrs.) 18. Mark with scriber and steel rule. (01 hr) 19. Measuring practice with steel rule, outside & inside callipers. (06 hrs.)	Bench work – Metal working hand tools and devices –Work bench – vices – files – hacksaw –hammer – spanners – screw drivers. Linear measurements- its units, steel rule dividers, Punch – types and uses. Description use and care of marking table. (05 Hrs.)

		20. Dot punching and letter and number punching. (05 hrs.)	Vernier calliper – its parts, principles, reading, uses and care. Outside micrometer – its parts, principles, reading, uses and care, vernier height gauge. Marking tools – scriber.Marking out – Coordinates system, Rectangular – Polar –Rules for marking. Bevel protractor, combination set- their components, uses and cares. Pedestal grinder, star wheel dresser, safety precautions, care and maintenance. (12 Hrs.)
		21. Grinding, centre punch, dot punch, flat chisel and scriber. (10hrs.) 22. Drill grinding practice. (10hrs.)	Marking media, special application. Surface plate and auxiliary marking equipment, 'V' block, angle plates,parallel block, description, types, uses, accuracy, care and maintenance.Drill, Tap, Die-types & application. Determination of tap drill size. Reamer- material, types (Hand and machine reamer), parts and their uses, determining hole size for reaming, Reaming procedure. Drilling machines-types and their application, construction of Pillar & Radial drilling machine. Countersunk, counter bore and spotfacing tools and nomenclature.Cutting Speed, feed, depth of cut and Drilling time calculations. (05Hrs.)
		23. Drill Plate filing to an accuracy of ±0.05mm. (10 hrs.) 24. Marking for centre punching, drilling, reaming, tapping, counter boring, counter sinking. (02 hrs.) 25. Centre punching, drilling, reaming, tapping, counter boring, counter sinking on drill plate. (12hrs.) 26. Die pass on standard material (M8). (08 hrs.) 27. Cutting tool filing and grinding on standard material. (10 hrs.)	Dial test indicator-its parts, types, construction and uses.Interchangeability: Necessityin Engineering. field, Limitˆ Definition, types,terminology of limits and fits-basic size, actualsize, deviation, high and low limit, zero-line, tolerance zone, allowances. Different standard systems of fits and limits. Geometrical tolerance. British standard system, BIS system. (10Hrs.)
Professional Skill 110 Hrs.; Professional Knowledge 20 Hrs.	Marking different of components for assembling as per required tolerance observing principle of interchange ability and check for functionality. [Different Fit – Open, Angular, & Square Fit; Required tolerance: ±0.05 mm, angular tolerance: 1 degree.] NOS:CSC/N0309	28. Make Male & Female 'Open' fitting with accuracy ±0.05 mm. (25 hrs.)	Introduction about metals, difference between Metal and Non-Metal, properties of metal, Classification of metals and its applications, pig – iron, cast iron, wrought iron, steel-plain carbon steel (Low carbon steel, medium and high carbon steels, high speed steel, stainless steel, carbides, etc.) (04 Hrs.)
		29. Make male & female for square fit with accuracy ± 0.05 mm. (30hrs.)	Heat treatment of metals, process- such as annealing, nit riding, hardening, tempering, case hardening, carburizing,cyaniding, flame hardening, Induction hardening, purposes and its effects on the properties of steel. (08 Hrs.)

		30. Angular fitting with male & female. (30 hrs.) Assembly fit with male & female by dowelling and screwing. (25 hrs.)	Getting to know the lathe with its main components, lever positions and various lubrication points as well. Definition of machine & machine tool and its classification. (08 Hrs.)
Professional Skill 32 Hrs.; Professional Knowledge 07 Hrs.	Set different shaped jobs on different chuck and demonstrate conventional lathe machine operation observing standard operation practice. [Different chucks:3 jaws & 4 jaws, different shaped jobs: round, square, hexagonal] NOS:CSC/N0316	31. Identify & function of different parts of lathe. Practice on operation of lathe (dry/idle run). (10 hrs.) 32. Setting lathe on different speed and feed. (02 hrs.) 33. Profile turning using hand tools- radius external and internal. (20hrs.)	Introduction to lathe. Centre lathe construction, detail function of parts, specification. Safety points to be observed while working on a lathe. (07 Hrs.)
Professional Skill 95 Hrs.; Professional Knowledge 21 Hrs.	Prepare different cutting tool to produce jobs to appropriate accuracy by performing different turning operations. [Different cutting tool – V tool, side cutting, parting, thread cutting (both LH & RH), Appropriate accuracy: ±0.06mm, Different turning operation –Plain, facing, drilling, boring (counter & stepped), grooving, Parallel Turning Step Turning, parting, chamfering, U -cut, Reaming, internal recess, knurling.] NOS:CSC/N0316	34. Grinding of R.H. and L.H. tools, parting tool, Round nose tool. (05hrs.) 35. Checking of angles with angle gauge / bevel protractor. (02 hrs.) 36. Grinding of "V" tools for threading of Metric/ British threads. (04hrs.) 37. Plain turning (holding in 4 –jaw chuck), step turning and forming shoulder, chamfering in between centres as per dimensions. (28hrs.) 38. Pillar turning between centres (07 hrs.)	Different types of Lathe operations - facing, turning, parting-off, grooving, chamfering, boring etc.Lathe cutting tool-different types, shapes and different angles (clearance, rake etc.), specification of lathe tools. Types of chips, chip breaker.Tool life, factors affecting tool life. (10 Hrs.)
		39. Bush turning, drilling and boring/ reaming. (14 hrs.) 40. Turning and die passing in a standard material. (03 hrs.) 41. Pin punch turning and knurling (05 hrs.) 42. Using 4 – jaw chuck; face both side of a plate thickness as per drawing. (02 hrs.)	Driving mechanism, speed and feed mechanism of Lathe Concept of Orthogonal and Oblique Cutting. Chucks & different types of job holding devices on lathe and advantages of each type. Mounting and dismounting of chucks.Knurling-types, grade & its necessity.Vernier Bevel Protractor – parts, reading and uses. (04 Hrs.)
		43. Taper turning male and female work pieces and assembly. (25hrs.)	Various material for single point cutting tools, tip tools- their brazing and grinding process. (07 Hrs.)
Professional Skill 25 Hrs.; Professional Knowledge 05Hrs.	Set the different machining parameters to produce threaded components applying method/ technique and test for proper assembly of the components with an accuracy of ± 0.05 mm. [Different threads viz., metric/ BSW/ Square] NOS:CSC/N0316	44. External thread cutting on step turned work piece. (Metric, BSW & Square Thread) (15hrs.) 45. Turn job for Internal thread and cut internal thread (10 hrs.).	Calculations of taper turning by off-setting tail stock. Sine Bar – description & uses Slip gauge –description and uses. (05 Hrs.)

(Professional Skill 128 Hrs.; Professional Knowledge 18 Hrs.)	Set the different machining parameters and cutters to prepare job by performing different milling operation and indexing. [Different machining parameters – feed, speed and depth of cut. Different milling operations –plain, stepped, angular, dovetail, T-slot, contour, gear milling] NOS:CSC/N0316	46. Identification of milling machine. (02 hrs.) 47. Demonstrate working principle of Milling Machine. (02 hrs.) 48. Set vice & job on the table of Milling Machine. (03 hrs.) 49. Set arbor on the spindle of milling machine. (04 hrs.) 50. Set the cutter on arbor. (02hrs.) 51. Safety points to be observed while working on a milling machine. (02 hrs.)	Milling Machine: importance, types, construction and specification. Driving and feed mechanism of Milling Machine Different milling cutter angles, Milling cutter materials (05 Hrs.)
		52. Demonstrate Up Milling and down Milling Process. (04 hrs.) 53. Perform sequence of milling for six faces of a solid block 2 numbers. (13 hrs.) 54. Check the accuracy with the help of tri-square and vernier height gauge. (02 hrs.) 55. Perform Step milling using side and face cutter checking with depth micrometer. (05 hrs.)	Job holding devices-vice, clamps, V-block, parallel block etc. Milling cutter holding devices, milling process – Up milling and Down milling. (02 Hrs.)
		56. Milling blank piece (plain milling).\ (10 hrs.) 57. Slot milling with side and face cutter (08 hrs.) 58. 90° angular milling with equal angle cutter. (08 hrs.) 59. Dove tail milling. (09 hrs.) 60. Tee slot milling. (08 hrs.)	Calculation of cutting speed, feed, machining time for milling machine. Milling machine operations. Milling machine attachments – vertical milling attachment, (03 Hrs.)
		61.Concave and Convex milling. (16hrs.)	Introduction to coolant & lubricant-difference between them, types and uses of each. (03 Hrs.)
		62. Simple indexing practice (30hrs.)	Dividing head – Introduction, construction, types. Simple and universal dividing head. Indexing methods – direct indexing, simple indexing, angular indexing, its calculations. (05 Hrs.)
Professional Skill 108 Hrs.; Professional Knowledge 09 Hrs.	Produce components of high accuracy by surface grinding operation. [Accuracy of +/- 0.02 mm] NOS:CSC/N0316	63. Identification of different types of grinding machine. (02 hrs.) 64. Wheel balancing & truing. (06hrs.) 65.Dressing of grinding wheel. (02hrs.) 66.Grinding of block (six sides) in surface grinding machine with an accuracy of ±0.01 mm. (15 hrs.) 67.Grinding of step block in surface grinding machine with an accuracy of ± 0.01 mm. (15 hrs.)	Grinding machine introduction, types, Surface & Cylindrical grinding Machine- their parts, functions, specification, and uses. Safety points to be observed while working on a Grinding machine. (05 Hrs.)

		68.Grinding of slot block in surface grinding machine with an accuracy of ± 0.01 mm. (15 hrs.) 69. Set and perform angular grinding using sine plate to stranded angle. (18 hrs.) 70. Make slide fit (male/female) (12hrs.) 71. Perform form grinding. (08 hrs.) 72. Taper angle grinding fitting. (15hrs.)	Grinding wheel shapes and sizes. Standard marking system. Selection of grinding wheel. (04 Hrs.)
Professional Skill 66 Hrs.; Professional Knowledge 08 Hrs	Produce components of high accuracy by cylindrical grinding operations. [Accuracy of + 0.02mm.] NOS:CSC/N0316	**Cylindrical grinding:** 73.External Parallel grinding (Bothholding in chuck/ collet and in between centres. (17 hrs.) 74. Plunge grinding. (04 hrs.)	Procedure for mounting of grinding wheels, balancing of grinding wheels. Dressing, types of dresser. Glazing and Loading of wheels – its Causes and remedies. Roughness values and their symbols. Explain the importance and necessity of quality. (04 Hrs).
		Cylindrical grinding: 75.Internal Parallel grinding (Both holding in chuck/collet). (20 hrs.)	Abrasives - its types, Bond, Grade, Grit, structure. (04 Hrs.)
		76.Grinding of step in Cylindrical grinding machine with an accuracy of ±0.01 mm (15 hrs.) 77.Grinding of external taper in Cylindrical grinding machine with an accuracy of ± 0.01 mm. (10hrs.)	
Professional Skill 30 Hrs.; Professional Knowledge 04 Hrs.	Sharpen different cutter or multipoint cutting tool. [Different cutters – end mill cutter, side & face milling cutter, single angle cutter, Reamer] NOS:CSC/N0316	78.Demonstrate and practice of grinding of end mill cutter of different sizes by using tool & cutter grinding machine. (30 hrs.)	Tool & cutter grinderˆconstruction, use and specification. (04 Hrs.)
Professional Skill 108 Hrs.; Professional Knowledge 28 Hrs.	Develop isometric drawing and solid modelling of mould using CAD & Pro-E. NOS:CSC/N9492	79.Prepare simple mould design drawings with basics of AutoCAD viz., Basic and advanced 2D drafting, draw commands, Constraints, Modify commands, Layers, Line types block, Texts, Attribute, Table, Dimensioning, Isometric, Solid modelling, View port. (58 hrs.)	AutoCAD: Introduction to AutoCAD, creating first drawing, learning the tools trade, organizing the work, drawing the first mould. (14 Hrs.))
		80.Prepare solid modelling of simple mould with Pro-E [Sketch, Part (solid, surface, free style, flexible modelling, sheet metal.), Assembly, Creo direct, Creo simulate]. (25 hrs.) 81.Creating (NC assembly and mould cavity) drawing. (10 hrs.) 82.Part drawing of the universal coupling assembled all the parts and solid modelling and denoted by coloured combination. (15hrs.)	Pro-E: Familiarization of interface/ Windows, Sketching, basic modeling, advanced modeling, assembling, drawing, surface modeling, manufacturing – mould design awareness. (14Hrs.)

Guide lines to work with national skill training institute system including store's procedures

Objectives: At the end of this lesson you shall be able to
- **state what is ITI and brief the objectives of ITI**
- **describe the organizational chart**
- **list out the infrastructure available in ITI**
- **explain the job opportunities and carrier development after completion of courses**
- **brief the examination pattern and soft skills.**

Introduction to ITI

Industrial Training Institutes (ITI) and Industrial Training Centers (ITC) comes under Craftsman training Scheme (CTS) to provide Vocational training in various trades functioning under Directorate General of Training (DGT), Ministry of Skill Development and Entrepreneurship, Government of India.

ITIs and ITCs are one and the same; ITIs are governed by the state/union government, whereas, the ITCs are self-financing institutions to provide same training courses as ITI's. Trade test for ITI and ITC trainees are common and the National Trade Certificate issued by the National Council for Vocational Training (NCVT) is of the same standard.

The objectives of an ITI

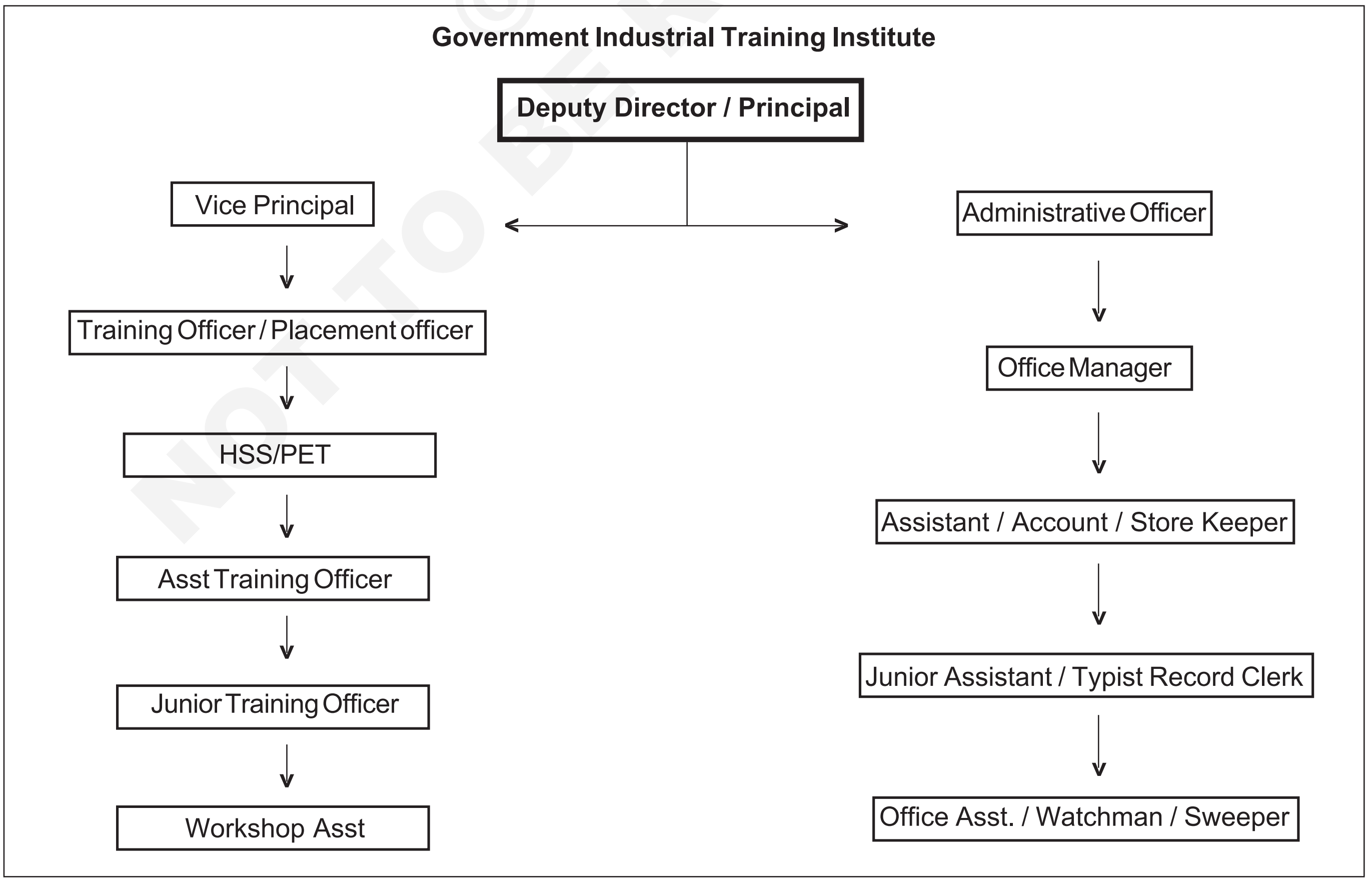
The objective of an ITI is to ensuring a steady flow of skilled workers and to reduce unemployment among the educated youth by training & equipping for suitable industrial employment and as well as for self employment.

The institute imparts training in engineering and non engineering two years/ one year trade courses approved by the Government of India in consultation with the National Council for Vocational Training, New Delhi.

Structure of ITI

The structure of industrial training institute is shown in the following chart 1. It may vary state to state It explain the information /order flow from higher superior officials to the ground level officials. The working hours may differ state to state. The trade master is the overall in-charge for the particular trade .the trainee has to report to trade master.

In every ITI there is a store and the incharge of the store is storekeeper for inward and outward movement of tools, equipment and consumables. The instructor will indent the training requirement for the training purposes.



Infrastructure available in ITI's

To provide 100% practical training to the trainees, tools, equipments, machineries and classroom facilities are available in ITI's. Continuous learning process/ programs are conducted in regular intervals as per the instructions given by the DGT.

The following facilities are available in ITI's

- Hostel facilities
- Libraries
- Soft skills lab/ computer labs
- High end classrooms /smart class.
- Stores
- Sports
- Wifi enabled campus.
- Industrial visit's/ Industrialist guest lecture
- Internship training on the job training
- Apprentice programs
- Campus interview etc

CTS Admission Process

Online counseling is conducted Statewide selection is made on merit basis duly following rules of reservation. The candidates exercise the option of choosing the ITI and trade of their choice.

Students between the age of 14 - 40 are admitted in Industrial Training Institutes. Admission is made during the month of August every year.

Craftsman Training Scheme Exam System

Final Trade Test is conducted on All India basis and the question papers are issued to all Trade Testing Centres on the same day by the NCVT. Passed-out candidates are issued with National Trade Certificate (NTC) under the seal and authority of NCVT by DGT, New Delhi

Job area after completion of training

This highlights the employability aspect on completion of training. The trainee should be aware of various prospects available in present market scenario along with scope for self-employment. For example a trainee with NTC engineering trade may opt for various jobs available in different industries in india and abroad.

After successful completion of training in any one of the engineering trade one can seek appointment in engineering workshop /Factories (Public Sector, Private Sector and Government Industries) in India and abroad as technician / Skilled worker.

Self employment

One can start is own factory / ancillary unit or design products manufacture and become an entrepreneur.

Further learning scope

- Apprentice training in designated trade.
- Craft Instructor certificate course.
- Diploma in relevant Engineering.

Skill competition

All India Skill Competition for Craftsmen scheme at national level was introduced to foster a healthy competition among the trainees of ITIs / ITCs

India skill competition is organized by National Skill Development Corporation., India skill competition the country's biggest skill competition is designed to demonstrate the highest standards of skilling and offers a platform to young people to show their talent at national level and international levels.

The competition is now held every year in 15 trades viz. Instrument Mechanic, Electronic Mechanic, Welder, Fitter, Turner, Machinist, Mechanic Motor Vehicle, Foundry man, Electrician, Cutting & Sewing, Computer Operator & Programming Assistant, Draughtsman (Civil), Draughtsman (Mechanical), Mechanic Diesel and Mechanic Refrigeration & Air- Conditioning.

The best trainee of each of the above trades at the State level competition competes at the All India Skill Competition.

Awards

The best Craftsmen in each of the above 15 trades at the All India level are awarded merit certificates and a cash prize of Rs. 50,000/- each. ITIs whose trainee stands first in the competition at the All India Skill Competition is awarded a merit certificate and is declared as the best ITI.

Approach on soft skills

Soft skills - refer to the cluster of personality traits, social graces, facility with language, personal habits, friendliness, and optimize that make people to varying degrees. The same can also be defined as-ability to interact communicates positively and productively with others. Sometimes called "character skills".

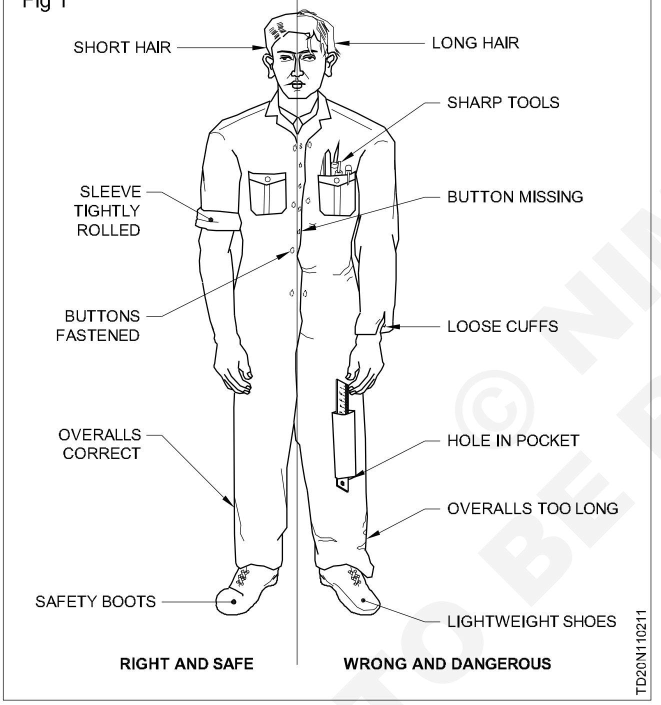
More and more business are considering soft skills as an important job criteria. Soft skills are used in personal and professional life. Hard skills / technical skills do not matter without soft skills.

Safety and general precautions in industry/shop floor

Objectives: At the end of this lesson you shall be able to
- **state the importance of safety**
- **list out the safety precautions to be observed in a industry/shop floor**
- **list out the personal safety precautions to be observed in machine shop**
- **list out the safety precautions to be observed while working on the machines.**

Generally accidents do not happen; they are caused. Most accidents are avoidable. A good craftsman, having a knowledge of various safety precautions, can avoid accidents to himself and to his fellow workers and protect the equipment from any damage. To achieve this, it is essential that every person should follow safety procedure. (Fig 1)

Fig 1



Safety in a workshop can be broadly classified into 3 categories.
- General safety
- Personal safety
- Machine safety

General safety

Keep the floor and gangways clean and clear.

Move with care in the workshop, do not run.

Don't leave the machine which is in motion.

Don't touch or handle any equipment/ machine unless authorised to do so.

Don't walk under suspended loads.

Don't crack practical jokes while on work.

Use the appropriate tools for the job.

Keep the tools at their proper place.

Wipe out split oil immediately.

Replace worn out or damaged tools immediately.

Never direct compressed air at yourself or at your co-worker.

Ensure adequate light in the workshop.

Clean the machine only when it is not in motion.

Sweep away the metal cuttings.

Know everything about the machine before you start it.

Personal safety

Wear a one piece overall or boiler suit.

Keep the overall buttons fastened.

Don't use ties and scarves.

Roll up the sleeves tightly above the elbow.

Wear safety shoes or boots

Cut the hair short.

Don't wear a ring, watch or chain.

Never lean on the machine.

Don't clean hands in the coolant fluid.

Don't remove guards when the machine is in motion.

Don't use cracked or chipped tools.

Don't start the machine until

– the workpiece is securely mounted

– the feed of machinery is in the neutral

– the work area is clear & neat.

Don't adjust clamps or holding devices while the machine is in motion.

Never touch the electrical equipment with wet hands.

Don't use any faulty electrical equipment.

Ensure that electrical connections are made by an authorised electrician only.

Concentrate on your work. Have a calm attitude.

Do things in a methodological way.

Don't engage yourself in conversation with others while concentrating on your job.

Don't distract the attention of others.

Don't try to stop a running machine with hands.

Machine safety

Switch off the machine immediately, if something goes wrong.

Keep the machine clean.

Replace any worn out or damaged accessories, holding devices, nuts, bolts etc as soon as possible.

Do not attempt operating the machine until you know how to operate it properly.

Do not adjust tool or the workpiece unless the power is off.

Stop the machine before changing the speed.

Disengage the automatic feeds before switching off.

Check the oil level before starting the machine.

Never start a machine unless all the safety guards are in position.

Take measurements only after stopping the machine.

Use wooden planks over the bed while loading and unloading heavy jobs.

Safety is a concept, understand it. Safety is a habit, cultivate it.

Approach on soft skills

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

- **state the concept of soft skill**
- **list the important common soft skills**
- **brief the employability aspect of training**
- **brief the further learning scope.**

Concept: Soft skills - refer to the cluster of personality traits, social graces, facility with language, personal habits, friendliness, and optimism that make people to varying degrees. The same can also be defined as an ability to interact communicate positively & productively with others. Sometimes called “character skills”.

More and more business are considering soft skills as important job criteria. Soft skills are used in personal and professional life. Hard skills/technical skills do not matter without soft skills.

Common Soft Skills

- Strong work ethic
- Positive attitude
- Good communication skills
- Interpersonal skills
- Time management abilities
- Problem-solving skills
- Team work
- Initiative, Motivation
- Self-confidence
- Loyalty
- Ability to accept and learn from criticism
- Flexibility, Adaptability
- Working well under pressure

Job area completion of training: This highlights the employability aspect on completion of training. The trainee should be aware of various prospects available in present market scenario along with scope for self-employment. For example a trainee with NTC engineering trade may opt for:

Various job available in different industries in India and Abroad.

After successful completion of ITI training in any one of the engineering trade one can see appointment in engineering workshop/Factories (Public Sector, Private Sector and Government Industries) in India and Abroad as technician/Skilled worker.

Self employment

One can start is own factory/ancillary unit or design products manufacture and became an entrepreneur.

Further learning scope

- Apprentice training in designated trade.
- Craft Instructor certificate course.
- Diploma in relevant Engineering.

Personal Protective Equipment (PPE)

Objectives: At the end of this lesson you shall be able to
- **state what is personal protective equipment and its purpose**
- **name the two categories of personal protective equipment**
- **list the most common type of personal protective equipment**
- **list the conditions for selection of personal protective equipment.**

Personal protective equipment

Personal protective equipment, commonly referred to as "PPE", is equipment worn to minimize exposure to hazards that cause serious workplace injuries and illnesses. These injuries and illnesses may result from contact with chemical, radiological, physical, electrical, mechanical, or other workplace hazards. Personal protective equipment may include items such as gloves, safety glasses and shoes, earplugs or muffs, hard hats, respirators, or coveralls, vests and full body suits.

Categories of PPE-Small's'

Depending upon the nature of hazard, the PPE is broadly divided into the following two categories.

Non- respiratory : Those used for protection against injury from outside the body, i.e. for protecting the head, eye, face, hand, arm, foot, leg and other body parts

Respiratory: Those used for protection from harm due to inhalation of contaminated air.

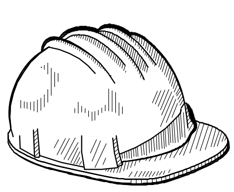
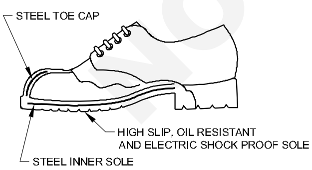
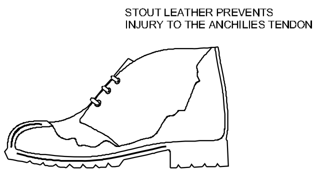
They are to meet the applicable BIS (Bureau of Indian Standards) standards for different types of PPE.

The guidelines on 'Personal Protective Equipment' is issued to facilitate the plant management in maintaining an effective program with respect to protection of persons against hazards, which cannot be eliminated or controlled by engineering methods listed in table 1.

Table 1

No	Title
PPE1	Helmet
PPE2	Safety footwear
PPE3	Respiratory protective equipment
PPE4	Arms and hands protection
PPE5	Eyes and face protection
PPE6	Protective clothing and coverall
PPE7	Ears protection
PPE8	Safety belt harness

Personal protective equipment and their uses and hazards are listed in Table 2

Types of protection	Hazards	PPE to be used
Head protection (Fig 1)  HELMET	1 Falling objects 2 Striking against objects 3 Spatter	Helmets
Foot protection (Fig 2)  INDUSTRIAL SAFETY SHOE  INDUSTRIAL SAFETY BOOT	1 Hot spatter 2 Falling objects 3 Working wet area	Leather leg guards Safety shoes Gum boots

<table>
<tr><th>Types of protection</th><th>Hazards</th><th>PPE to be used</th></tr>
<tr><td>Nose (Fig 3)

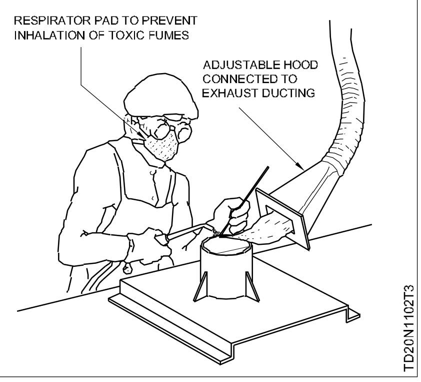
Fig 3

</td><td>1 Dust particles
2 Fumes/gases/ vapours</td><td>Nose mask</td></tr>
<tr><td>Hand Protecion (Fig 4)

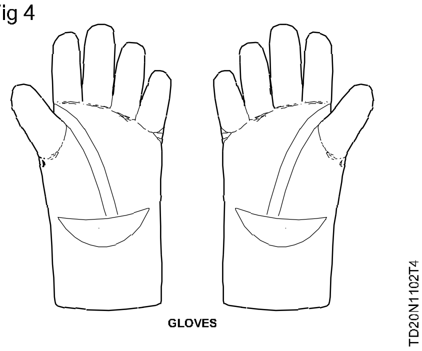
Fig 4

</td><td>1 Heat burn due to direct contact
2 Blows spark moderate heat
3 Electric shock</td><td>Hand gloves</td></tr>
<tr><td>Eye protection (Fig 5 & Fig6)

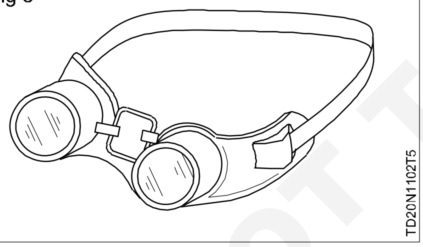
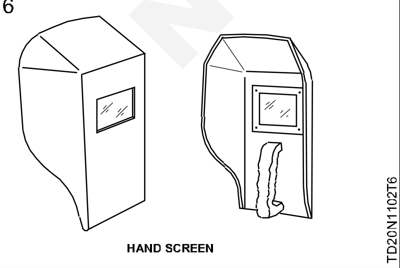
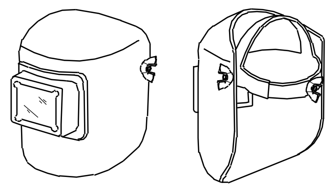
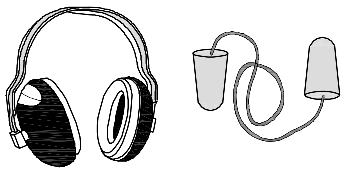
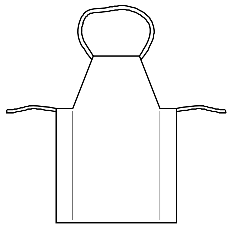
Fig 5


Fig 6

</td><td>1 Flying dust particles
2 UV rays, IR rays heat and
High amount of visible</td><td>Goggles
Face shield
radiation
Hand shield
Head shield</td></tr>
</table>

Types of protection	Hazards	PPE to be used
Face protection (Fig 6 &Fig 7) Fig 7 	1 Spark generated during Welding, grinding 2 Welding spatter striking 3 Face protection from UV rays	Face shield Head shield with or without ear muff Helmets with welders Screen for welders
Ear protection (Fig 8s) Fig 8 	1 High noise level Ear muff	Ear plug
Body protection (Fig 8, & Fig 9)  APRON LEATHER APRONS 	1 Hot particles	Leather aprons

Quality of PPE's

PPE must meet the following criteria with regard to its quality-provide absolute full protection against possible hazard and PPE's be so designed and manufactured out of materials that it can withstand the hazards against which it is intended to be used.

Selection of PPE's requires certain conditions

- Nature and severity of the hazard
- Type of contaminant, its concentration and location of contaminated area with respect to the source of reparable air
- Expected activity of workman and duration of work, comfort of workman when using PPE
- Operating characteristics and limitation of PPE
- Easy of maintenance and cleaning
- Conformity to Indian / International standards and availability of test certificate.

Proper use of PPEs

Having selected the proper type of PPE, it is essential that the workman wears it. Often the workman avoids using PPE. The following factors influence the solution to this problem.

- The extent to which the workman understands the necessity of using PPE
- The ease and comfort with which PPE can be worn with least interference in normal work procedures
- The available economic, social and disciplinary sanctions which can be used to influence the attitude of the workman
- The best solution to this problem is to make wearing of PPE' mandatory for every employee.
- In other places, education and supervision need to be intensified. When a group of workmen are issued PPE for the first time.

First-aid

Objectives: At the end of this lesson you shall be able to
- **state what is first aid**
- **list the important guide lines for the first aid**
- **explain the ABC of the first aid**
- **brief how to give first-aid for a victim who need first aid.**

Basic first aid: Basic first aid refers to the initial process of assessing and addressing the needs of someone who has been injured or is in physiological distress due to choking, a heart attack, allergic reactions, drugs or other medical emergencies. Basic first aid allows one to quickly determine a person's physical condition and the correct course of treatment.

Golden hours: India have best of technology made available in hospitals to treat devastating medical problem viz. head injury, multiple trauma, heart attack, strokes etc, but patients often do poorly because they don't gain access to that technology in time. The risk of dying from these conditions, is greatest in the first 30 minutes, often instantly. This period is referred to as Golden period. By the time the patient reach hospitals, they would have passed that critical period. First aid care come handy to save lives. It helps to get to the nearest emergency room as quickly as possible through safe handling and transportation. The shorter that time, the more likely the best treatment applied.

Important guideline for first aiders

Evaluate the situation: Are there things that might put the first aider at risk. When faced with accidents like fire, toxic smoke, gasses, an unstable building, live electrical wires or other dangerous scenario, the first aider should be very careful not to rush into a situation, which may prove to be fatal.

Remember A-B-Cs

The ABCs of first aid refer to the three critical things the first aiders need to look for.

- Airway - Does the person have an unobstructed airway?
- Breathing - Is the person breathing?
- Circulation - Does the person show a pulse at major pulse points (wrist, carotid artery, groin)

Avoid moving the victim: Avoid moving the victim unless they are in immediate danger. Moving a victim will often make injuries worse, especially in the case of spinal cord injuries.

Call emergency services: Call for help or tell someone else to call for help as soon as possible. If alone in at the accident scene, try to establish breathing before calling for help, and do not leave the victim alone unattended.

Determine responsiveness: If a person is unconscious, try to rouse them by gently shaking and speaking to them.

> **If the person remains unresponsive, carefully roll them on the side (recovery position) and open his airway.**

- Keep head and neck aligned.
- Carefully roll them onto their back while holding his head.
- Open the airway by lifting the chin. (Fig 1)

Fig 1

Look, listen and feel for signs of breathing

Look for the victim's chest to raise and fall, listen for sounds of breathing.

If the victim is not breathing, see the section below

- If the victim is breathing, but unconscious, roll them onto their side, keeping the head and neck aligned with the body. This will help drain the mouth and prevent the tongue or vomit from blocking the airway.

Check the victim's circulation: Look at the victim's colour and check their pulse (the carotid artery is a good option; it is located on either side of the neck, below the jaw bone). If the victim does not have a pulse, start CPR.- If you are trained.

Treat bleeding, shock and other problems as needed

After establishing that the victim is breathing and has a pulse, next priority should be to control any bleeding. Particularly in the case of trauma, preventing shock is the priority.

- **Stop bleeding:** Control of bleeding is one of the most important things to save a trauma victim. Use direct pressure on a wound before trying any other method of managing bleeding.
- **Treat shock:** Shock, a loss of blood flow from the body, frequently follows physical and occasionally psychological trauma. A person in shock will frequently have ice cold skin, be agitated or have an altered mental status, and have pale colour to the skin around the face and lips. Untreated, shock can be fatal. Anyone who has suffered a severe injury or life-threatening situation is at risk for shock.
- **Choking victim:** Choking can cause death or permanent brain damage within minutes.
- **Treat a burn:** Treat first and second degree burns by immersing or flushing with cool water. Don't use creams, butter or other ointments, and do not pop blisters. Third degree burns should be covered with a damp cloth. Remove clothing and jeweler from the burn, but do not try to remove charred clothing that is stuck to burns.
- **Treat a concussion:** If the victim has suffered a blow to the head, look for signs of concussion. Common symptoms are: loss of consciousness following the injury, disorientation or memory impairment, vertigo, nausea, and lethargy.
- **Treat a spinal injury victim:** If a spinal injury is suspected, it is especially critical, not move the victim's head, neck or back unless they are in immediate danger.

Stay with the victim until help arrives: Try to be a calming presence for the victim until assistance can arrive.

Unconsciousness (COMA): Unconscious also referred as Coma, is a serious life threatening condition, when a person lie totally senseless and do not respond to calls, external stimulus. But the basic heart, breathing, blood circulation may be still intact, or they may also be failing. If unattended it may lead to death.

The condition arises due to interruption of normal brain activity. The causes are too many.

- Shock (Cardiogenic, Neurogenic)
- Head injury (Concussion, Compression)
- Asphyxia (obstruction to air passage)
- Extreme of body temperature (Heat, Cold)
- Cardiac arrest (Heart attack)
- Stroke (Cerebra-vascular accident)
- Blood loss (Hemorrhage)
- Dehydration (Diarrhea & vomiting)
- Diabetes (Low or high sugar)
- Blood pressure (Very low or very high)
- Over dose of alcohol, drugs
- Poisoning (Gas, Pesticides, Bites)
- Epileptic fits (Fits)
- Hysteria (Emotional, Psychological)

The following symptoms may occur after a person has been unconscious:

- Confusion
- Drowsiness
- Headache
- Inability to speak or move parts of his or her body (see stroke symptoms)
- Light headedness
- Loss of bowel or bladder control (incontinence)
- Rapid heartbeat (palpitation)
- Stupor

First aid

- Call EMERGENCY number.
- Check the person's airway, breathing, and pulse frequently. If necessary, begin rescue breathing and CPR.
- If the person is breathing and lying on the back and after ruling out spinal injury, carefully roll the person onto the side, preferably left side. Bend the top leg so both hip and knee are at right angles. Gently tilt the head back to keep the airway open. If breathing or pulse stops at any time, roll the person on to his back and begin CPR.
- If there is a spinal injury, the victims position may have to be carefully assessed. If the person vomits, roll the entire body at one time to the side. Support the neck and back to keep the head and body in the same position while you roll.
- Keep the person warm until medical help arrives.
- If you see a person fainting, try to prevent a fall. Lay the person flat on the floor and raise the level of feet above and support.
- If fainting is likely due to low blood sugar, give the person something sweet to eat or drink when they become conscious.

DO NOT

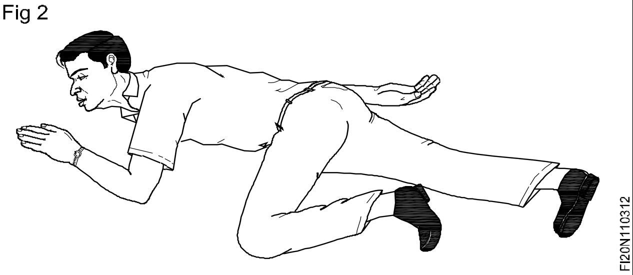
- Do not give an unconscious person any food or drink.
- Do not leave the person alone.
- Do not place a pillow under the head of an unconscious person.
- Do not slap an unconscious person's face or splash water on the face to try to revive him.

Loss of consciousness may threaten life if the person is on his back and the tongue has dropped to the back of the throat, blocking the airway. Make certain that the person is breathing before looking for the cause of unconsciousness. If the injuries permit, place the casualty in the recovery position with the neck extended. Never give anything by mouth to an unconscious casualty.

How to diagnose an unconscious injured person

- **Consider alcohol:** look for signs of drinking, like empty bottles or the smell of alcohol.
- **Consider epilepsy:** are there signs of a violent seizure, such as saliva around the mouth or a generally disheveled scene?

Fig 2



- **Think insulin:** might the person be suffering from insulin shock (see 'How to diagnose and treat insulin shock")?
- **Think about drugs:** was there an overdose? Or might the person have under dosed - that is not taken enough of a prescribed medication?
- **Consider trauma:** is the person physically injured?
- **Look for signs of infection:** redness and/ or red streaks around a wound.
- **Look around for signs of Poison:** an empty bottle of pills or a snakebite wound.
- **Consider the possibility of psychological trauma:** might the person have a psychological disorder of some sort?
- Consider stroke, particularly for elderly people.
- Treat according to what you diagnose.

Shock: A severe loss of body fluid will lead to a drop in blood pressure. Eventually the blood's circulation will deteriorate and the remaining blood flow will be directed to the vital organs such as the brain. Blood will therefore be directed away from the outer area of the body, so the victim will appear pale and the skin will feel ice cold.

Disposal of waste material

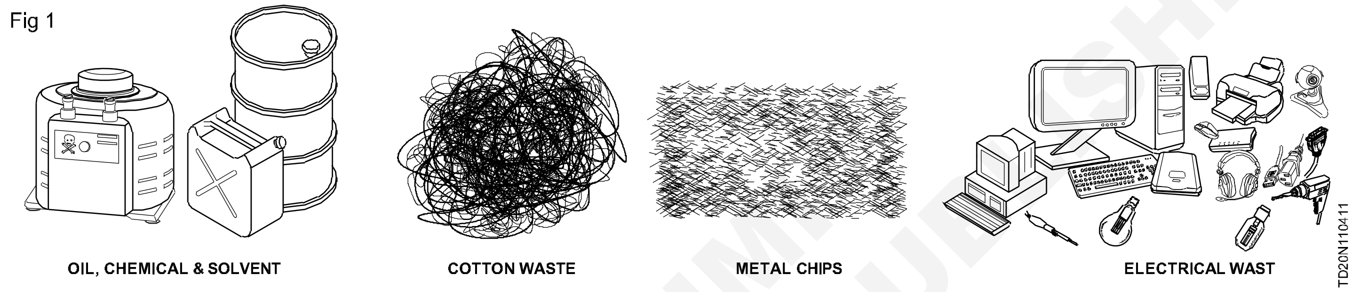
Objectives: At the end of this lesson you shall be able to
- **state what is waste material**
- **list the waste materials in a work shop**
- **explain the methods of disposal of waste material.**
- **state advantage of disposal of waste material.**
- **state colour code for bins for waste segregation.**

Waste material: industrial waste is the waste produced by industrial activity such as that of factories, mills and mines.

List of waste material (Fig 1)

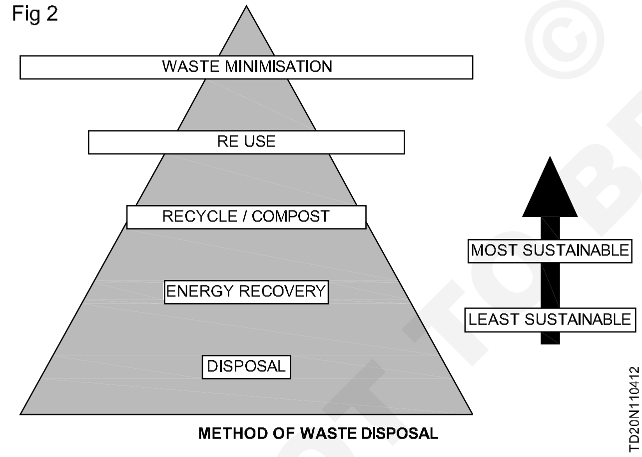
- Cotton waste
- Metal chips of different material.
- Oily waste such as lubricating oil, coolant etc.
- Other waste such electrical, glass etc.

Fig 1



Methods of waste disposal (Fig 2)

Fig 2



Recycling: Recycling is one of the most well known method of managing waste. It is not expensive and can be easily done by you. If you carry out recycling. you will save a lot of energy, resources and thereby reduce pollution.

Composting: This is a natural process that is completely free of any hazardous by-products. This process involves breaking down the materials into organic compounds that can be used as manure.

Landfills: Waste management through the use of landfills involves the use of a large area. This place is dug open and filled with the waste.

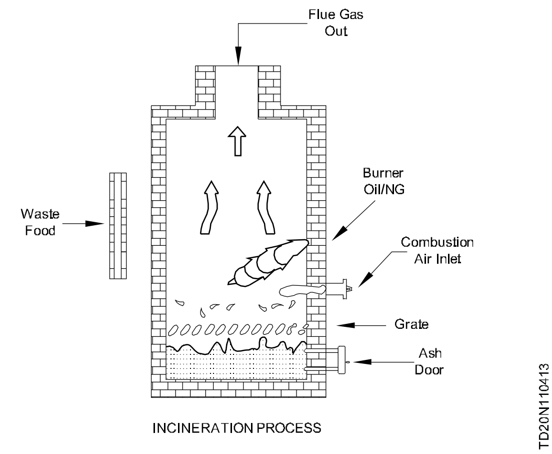
Burning the waste material: If you cannot recycle or if there are no proper places for setting up landfills, you can burn the waste matter generated in your household. Controlled burning of waste at high temperatures to produce steam and ash is a preferred waste disposal technique.

Advantage of waste disposal:

- Ensures workshop neat & tidy
- Reduces adverse impact on health
- Improves economic efficiency
- Reduce adverse impact on environment

Incineration (Fig 3)

Fig 3



It is the process of controlled combustion of garbage to reduce it to incombustible matter, ash, waste gas and heat. It is treated and released into the environment (Fig 3). This reduced 90% volume of waste, some time the heat generated used to produce electric power.

Waste compaction: The waste materials such as cans and plastic bottles compact into blocks and send for recycling. This process space need, thus making transportation and positioning easy.

Colour code for bins for waste segregation given in Table 1

Table 1

Sl.No.	Waste Material	Color code
1	Paper	Blue
2	Plastic	Yellow
3	Metal	Red
4	Glass	Green
5	Food	Black
6	Others	Sky blue

Occupational safety and health

Objectives: At the end of this lesson you shall be able to
- **describe occupational safety and its importance at work place to prevent unsafe act and conditions in work-related activities**
- **brief the Environmental guidelines, legislations & regulations in India, framed to protect workplace health and safety.**
- **list the Occupational safety and health Tips.**

Occupational safety, and health

Occupational safety, and health means actions or working conditions which are safe from any cause resulting in danger to life, physique, mentality or health arising out of or related to working environment. OSH includes the laws, standards and programs that are aimed at making the workplace better for workers, along with co-workers, family members, customers, and other stakeholders.

The goal of Occupational safety and health

The goal of Occupational safety and health program is to foster a safe and healthy occupational environment. OSH also protects all the general public who may be affected by the occupational environment.

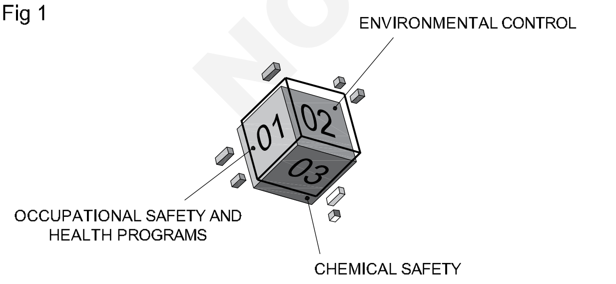
Environmental safety

Environmental safety is defined by the guidance, policies, and practices enforced in order to ensure that the surrounding environment is free from hazards that will warrant the safety and well-being of workers and employees, residents near industrial operations, as well as the prevention of accidental environmental damage

The surrounding areas include industrial facilities, work areas, and laboratories. Environmental safety is a crucial issue for any industrial activity as negligence and non-compliance heighten the risk resulting in injuries, illnesses, and accidental environmental releases.

Environmental safety is usually divided into three subcategories: (fig1) Occupational safety and Health Programs, Environmental Control, and Chemical Safety. (Fig 1)

Fig 1



ENVIRONMENTAL SAFETY SEGMENTS

In order to protect the workers against work related sickness, disease and injury. The International labour organization(ILO) came up with an official order on OSH.

Similarly government of India is enacted the following acts

- The legislation for labour welfare, known as the Factories Act 1948, was enacted with the prime objective of protecting workmen employed in factories against industrial and occupational hazards. There are number of Acts enacted by the government of India and amended from time to time; among them the following are the most important ones in this regard:
- Factories Act, 1948,
- Mines Act, 1952,
- Dock workers (Safety, Health and welfare) Act, 1986,
- Building and other Construction workers (Regulation of Employment and conditions of service) Act, 1996,
- Plantation Labour Act, 1951,
- Contract Labour (Regulation and Abolition) Act, 1970
- The Child labour (Prohibition and Regulation) Act, 1986, etc.

Constitutional provisions form the basis of workplace safety and health laws in India by imposing a duty on the State governments to implement policies that promote the safety and health of workers at workplaces. In addition, safety and health statutes for regulating occupational safety and health (OSH) of persons at work exist in different sectors, namely manufacturing, mining, ports, and construction sector.

The health and safety at work Act,1974 states employers are responsible for protecting the safety of their employees at work by preventing potential dangers in the workplace. It places general duties on employers to ensure the health, safety and welfare of all persons while at work.

Legislation is a directive proposed by a legislative body while a regulation is a specific requirement within legislation. Legislation is broader and more general while regulation is specific and details how legislation is enforced.

The difference between legislation and regulation is that legislation is the act of process of making certain laws while regulation is maintaining the law or set of rules that govern the people. It is a government-driven or ministerial order having the force of law.

The ILO's primary goal is to promote opportunities for women and men to obtain decent and productive work in conditions of freedom, equity, security and human dignity. In 2003 the ILO adopted a global strategy to improve preventive standards on occupational safety and health to provide essential tools for governments, employers, and workers to establish safe practices and health culture for providing maximum safety at work.

The four important aims of health and safety legislation is to

i secure the safety, health and welfare of employees and other people at work;

ii protect the public from the safety and health risks of business activities;

iii amend statutes relating to safety aspects of substances, equipment and environment;

iv eliminate workplace risks at the source.

Occupational safety and health Tips:

- Be aware of your surroundings.
- Maintain a correct posture.
- Take break regularly.
- Use Equipment properly.
- Locate Emergency Exits.
- Report Unsafe conditions.
- Practice Effective Housekeeping.
- Make use of mechanical aids.
- Wear the correct Safety equipment.
- Reduce workplace stress.

Safety signs

Objective : At the end of this lesson you shall be to
- **state the safety attitude and list the four basic categories of safety signs.**

Safety signs: As you go about your work on a construction site you will see a variety of signs and notices. Some of these will be familiar to you - a 'no smoking' sign for example; others you may not have seen before. It is up to you to learn what they mean - and to take notice of them. They warn of the possible danger, and must not be ignored.

Safety signs fall into four separate categories. These can be recognised by their shape and colour. Sometimes they may be just a symbol; other signs may include letters or figures and provide extra information such as the clearance height of an obstacle or the safe working load of a crane.

The four basic categories of signs are as follows:

- prohibition signs (Fig 1 & Fig 5)
- mandatory signs (Fig 2 & Fig 6)
- warning signs (Fig 3 & Fig 7)
- information signs (Fig 4)

Fig 1

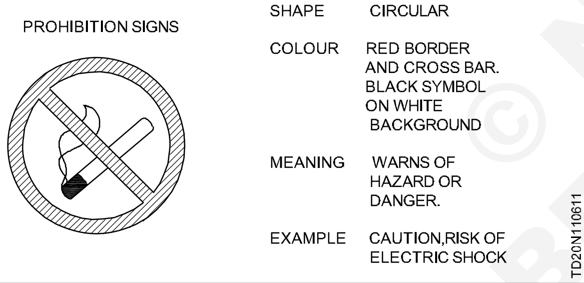


Fig 2

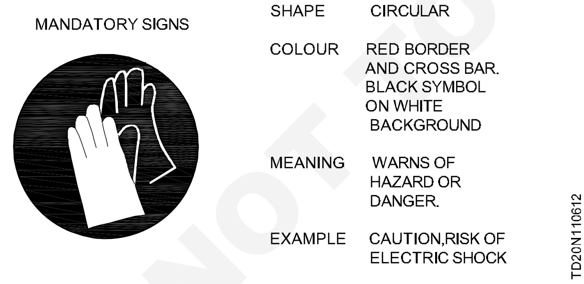


Fig 3

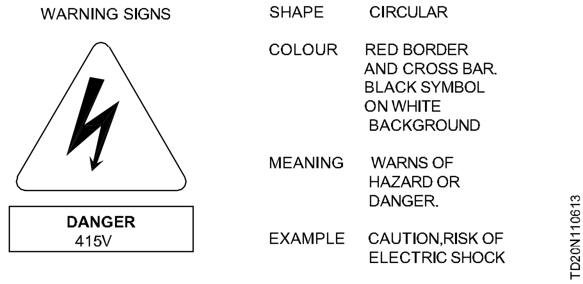


Fig 4

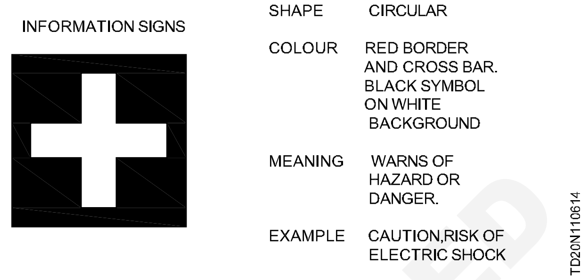
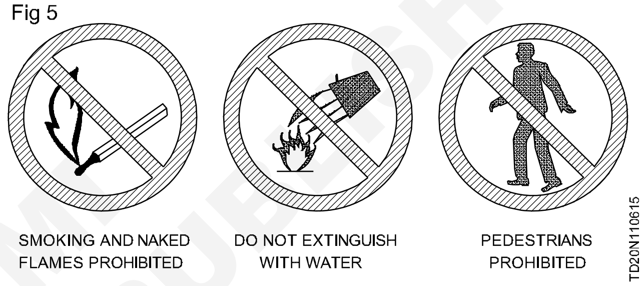


Fig 5

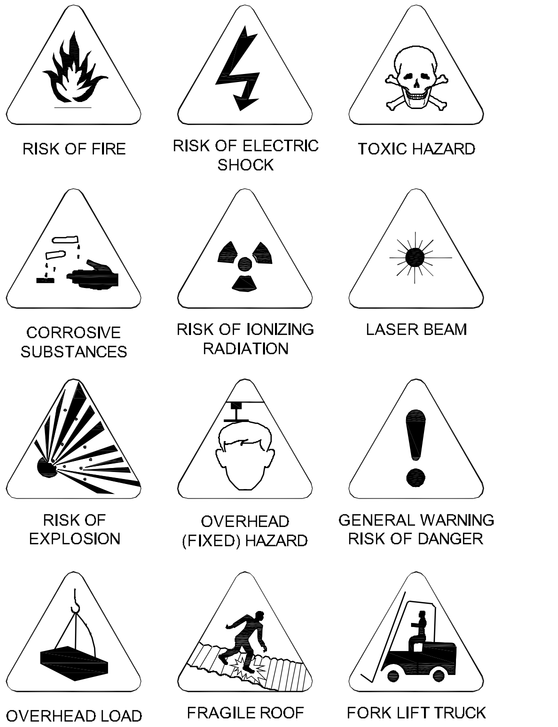


Mandatory signs

Fig 6

Warning signs

Fig 7



WARNING SIGNS

Question about your safety

Do you know the general safety rules that cover your place of work?

Are you familiar with the safety laws that govern you particular job?

Do you know how to do your work without causing danger to yourself, your workmates and the general public?

Are the plant, machinery and tools that you use really safe? Do you know how to use them safely and keep them in a safe condition?

Do you wear all the right protective clothing, and have you been provided with all the necessary safety equipment?

Have you been given all the necessary safety information about the materials used?

Have you been given training and instruction to enable you to do your job safely?

Do you know who is responsible for safety at your place of work?

Do you know who are the appointed 'Safety Representatives'?

Response to emergencies

Objectives: At the end of this lesson you shall be able to
- **respond incase of power failure, system failure and fire**
- **report an emergency.**

Power failure, System failure & Fire

1 If there is a power failure, start the emergency generator. This provides power to close the shutter, which is the first priority. The generator will also keep the UPSs and the cryogenic compressors running,

- Get a flash light.
- Look out for power transfer switch and switch over to normal power to emergency power by pressing the latch.
- Check the fuel valves open or not - Open the valves.
- Check to see that the main breaker switch ON the generator is in OFF position.
- Move the starter switch of the generator to run position. The engine will start at once.
- Allow few minutes to warm up the engine.
- Check all the gauges, pressure, temperature, voltage and frequency.
- Check the "AC line" and "Ready" green light on the front panel.

2 System failure

- If the bug or virus, invades the system. The system failure happens.
- Several varieties of bugs are there

1 Assasin bug

2 Lightening bug

3 Brain bug

For more details refer instruction manual for "System failure".

3 Fire

When fire alarm sounds in your buildings

- Evacuate to outside immediately.
- Never go back
- Make way for fire fighters and their trucks to come
- Never use an elevator
- Do not panic

Report an emergency

Reporting an emergency is one of those things that seems simple enough, until actually when put to use in emergency situations. A sense of shock prevail at the accident sites. Large crowd gather around only with inquisitive nature, but not to extend helping hands to the victims. This is common in road side injuries. No passer by would like to get involved to assist the victims. Hence first aid managements is often very difficult to attend to the injured persons. The first aiders need to adapt multitask strategy to control the crowd around, communicate to the rescue team, call ambulance etc, all to be done simultaneously. The mobile phones helps to a greater deal for such emergencies. Few guidelines are given below to approach the problems.

Assess the urgency of the situation. Before you report an emergency, make sure that the situation is genuinely urgent. Call for emergency services if you believe that a situation is life-threatening or otherwise extermely disruptive.

- A fire - If you're reporting a fire, describe how the fire started and where exactly it is located. If someone has already been injured, missing, report that as well.
- A life - threatening medical emergency, explain how the incident occurred and what symptoms the person currently displays.

Call emergency service

The emergency number varies - 100 for Police & Fire, 108 for Ambulance.

Report your location

The first thing the emergency dispatcher will ask where you are located, so the emergency services can get there as quickly as possible. Give the exact street address, if you're not sure of the exact address, give approximate information.

Importance of house keeping

Objectives: At the end of this lesson you shall be able to
- **list the steps involves in house keeping**
- **state good shop floor practices followed in industry**

Housekeeping

The following activities to be performed for better up keep of working environment:

- **Cleaning of shop floor:** Keep clean and free from accumulation of dirt and scrap daily
- **Cleaning of Machines :** Reduce accidents to keep machines cleaned well
- **Prevention of Leakage and spillage:** Use splash guards in machines and collecting tray
- **Disposal of Scrap-** Empty scrap, wastage, sward from respective containers regularly
- **Tools Storage-** Use special racks, holders for respective tools
- **Storage Spaces:** Identify storage areas for respective items. Do not leave any material in gangway
- **Piling Methods-** Do not overload platform, floor and keep material at safe height.
- **Material handling:** Use forklifts, conveyors and hoist according to the volume and weight of the package.

Good shop floor practices followed in industry

Good Shop floor practices are motivating action plans for improvement of the manufacturing process.

- All workers are communicated with daily target on manufacturing, activities.
- Informative charts are used to post production, quality and safety results compared to achievements.
- Workers are trained on written product quality standards.
- Manufactured parts are inspected to ensure adherence to quality standards.
- Production processes are planned by engineering to minimize product variation.
- 5s methods are used to organize the shop floor and production lines.
- Workers are trained on plant safety practices in accordance with Occupational Safety Health (OSH) standards.
- Workers are trained on "root cause" analysis for determining the causes of not following.
- A written preventive maintenance plan for upkeep of plant, machinery & equipment
- Management meets with plant employees regularly to get input on process improvements.
- Process Improvement Teams are employed to implement "best practices"

Introduction to 5S concept and its application

Objectives: - At the end of this lesson you shall be able to
- **stat what is 5S**
- **state the general benefits of implementing 5S**
- **explain the terms in 5S and its concept of implementation.**

Introduction

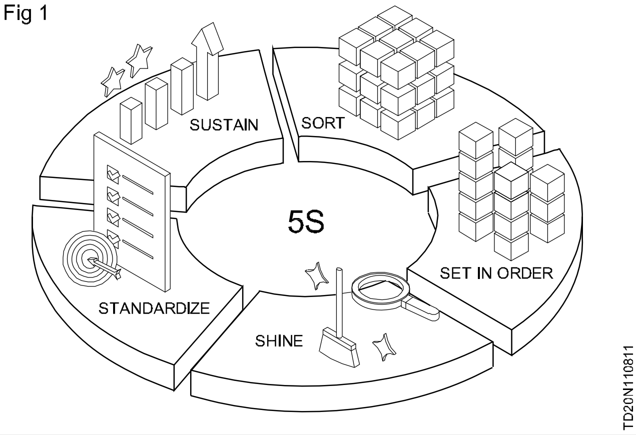
5S is a philosophy and a way of organizing and managing the workspace and work flow with the intent to improve efficiency by eliminating waste, improving flow and reducing process unreasonableness. There are five steps in the system, each starting with the letter S:

1 Sort 2 Set in order 3 Shine
4 Standardize 5 Sustain

The Steps of 5S (Fig 1)

5S was created in Japan, and the original "S" terms were in Japanese, so English translations for each of the five steps may vary. The basic ideas and the connections between them are easy to understand, though.

Fig 1

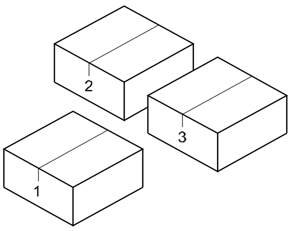


Step Name	Japanese term	Explanation
1	Sort Seri (tidiness)	Remove unnecessary items from each area
2	Set In Order	Seiton (orderliness) Organize and identify storage for efficient use
3	Shine Seiko (cleanliness)	Clean and inspect each area regularly
4	Standardize	Seiketsu (standardization) Incorporate 5S into standard operating procedures
5	Sustain Shinseki (discipline)	Assign responsibility, track progress, and continue the cycle

Step 1 Sort

The first step in the 5S process is Sort, or "seiri," which translates to "tidiness." The goal of the Sort step is to eliminate clutter and clear up space by removing things that don't belong in the area. (Fig 2)

Fig 2

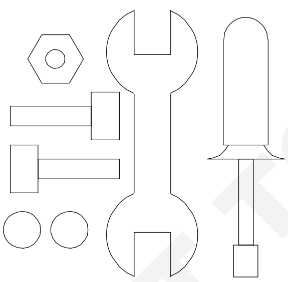


TD20N110812

Step 2: Set In Order

The second step, Set in Order, was originally called "seiton," which translates to "orderliness." A variety of names have been used in English: "Systematic Organization," "Straightening Out," and "Simplify," for example. No matter what it's called, the goal of this step is to organize the work area. Each item should be easy to find, use, and return: a place for everything, and everything in its place. (Fig 3)

Fig 3



TD20N110813

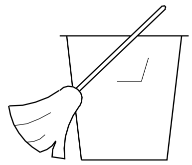
Implementation steps of Set in order

- Draw up a map, and then implement it
- Physically arrange the workplace first, and then map it out
- Map as you go, testing ideas and writing down what works well

Step 3: Shine

The third step of 5S is Shine, or "seiso," which means "cleanliness." While the first and second steps cleared up space and arranged the area for efficiency, this step attacks the dirt and grime that inevitably builds up underneath the clutter, and works to keep it from coming back.(Fig 4)

Fig 4

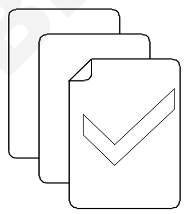


TD20N110814

Step 4: Standardize

The fourth step is Standardize, or "seiketsu," which simply means standardization. By writing down what is being done, where, and by whom, you can incorporate the new practices into normal work procedure. This paves the way for long-term change.(Fig 5)

Fig 5



TD20N110815

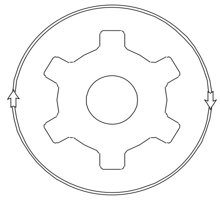
Tools for Standardizing

- 5S checklists
- Job cycle charts
- Procedure labels and signs

Step 5: Sustain

The fifth step of a 5S program is Sustain, or "shitsuke," which literally means "discipline." The idea here is continuing commitment. It's important to follow through on the decisions that you've made and continually return to the earlier steps of 5S, in an ongoing cycle. (Fig 6)

Fig 6



TD20N110816

Sustaining a 5S program can mean different things in different work places, but there are some elements that are common in successful programs.

- Management support
- Department tours
- Updated training
- Progress audits
- Performance evaluations

Operation of electrical mains

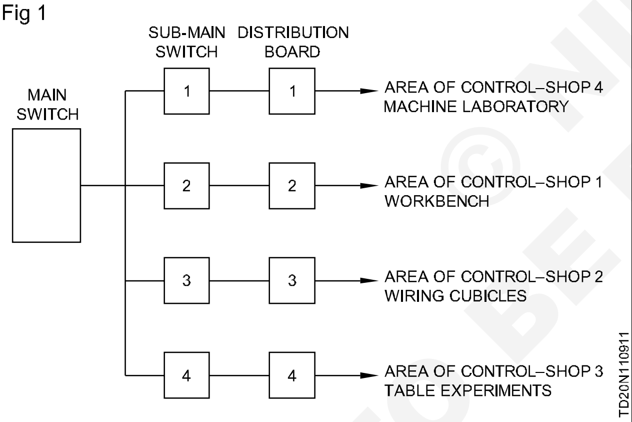
Objectives : At the end of this lesson you shall be able to
- **explain the term 'emergency'**
- **explain the need to switch off the circuit during emergency**
- **explain the method of locating the area sub-main and switches in the shop floor**
- **explain the position of handle with respect to ON & OFF in case of iron clad switches, MCB and ordinary house hold stitches.**

An emergency is an unexpected occurrence and requires immediate action. In a place like a workshop such a situation can arise when a person gets a shock due to electrical current or a person gets injured by the rotating part of a machine.

In such situations, switching off the supply will be the first and best solution to avoid further damage to the victim. For this, every person involved in the workshop should know which switch controls the area where the victim of shock remains.

Normally the total wiring in a workshop is controlled by a main switch and the different areas within the workshop may have two or more sub-main switches as shown in Fig.1.

Fig 1



To ascertain the area of the sub-main control, switch off one of the sub-main switches and try to switch 'on' the lights, fans and power points in that suspected area. If they do not work, then the area covered by the fan, light and power points are controlled by the sub-main switch. One after another, switch off the sub-main switches and locate their area of control. Mark the area of control of the switch in the plan of the wireman's section.

In a well organised workshop, the main switch, the sub main switches and distribution ways will have clear marking to show their area of control. (Fig 1) If this is not found, do this now. However, If you are not sure about the area of control the sub-main of the switches it is always better to switch 'off' the main switch itself.

The handle of iron clad switches and the knob of MCB should be pushed down to switch 'off ' the circuits as shown in Fig 2. whereas in the ordinary switches, the switch off the circuit should be done by pushing the switch to upward position. (Fig 3)

Fig 2

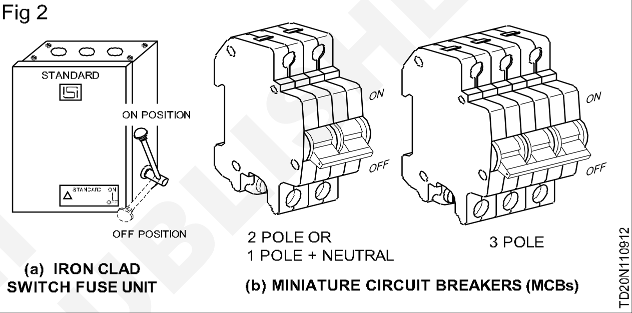
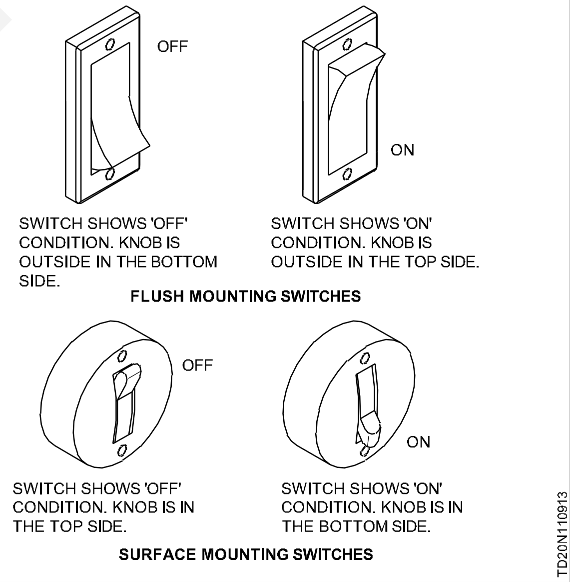


Fig 3



The emergency situations could happen even at home Hence, identify the area of control of the switch and mark them in the main/sub-main/ distribution bound of your house switch board as a safety measure. Educate the intimates of the house how to switch off the circuit in case of any emergency.

Electrical safety

Objectives : At the end of this lesson you shall be able to
- **explain the necessary of adopting the safety rules**
- **list the safety rules and follow them.**

Safety rules

Necessity of safety rules: Safety consciousness is one of the essential attitudes required for any job. A skilled electrician always should strive to form safe working habits. Safe working habits always save men, money and material. Unsafe working habits always end up in loss of production and profits, personal injury and even death. The safety hints given below should be followed by Electrician to avoid accidents and electrical shocks as his job involves a lot of occupational hazards.

The listed safety rules should be learnt, remembered and practised by every electrician. Here a electrician should remember the famous proverb, "Electricity is a good servant but a bad master".

Safety rules

- Only qualified persons should do electrical work
- Keep the workshop floor clean, and tools in good condition.
- Do not work on live circuits, if unavoidable, use rubber gloves rubber mats, etc.
- Use wooden or PVC insulated handle screwdrivers when working on electrical circuits.
- Do not touch bare conductors.
- When soldering, place the hot soldering irons in their stand. Never lay switched 'ON' or heated soldering iron on a bench or table as it may cause a fire to break out.
- Use only correct capacity fuses in the circuit. If the capacity is less it will blow out when the load is connected. If the capacity is large, it gives no protection and allows excess current to flow and endangers men and machines, resulting in loss of money.
- Replace or remove fuses only after switching off the circuit switches.
- Use extension cords with lamp guards to protect lamps against breakage and to avoid combustible material coming in contact with hot bulbs.
- Use accessories like sockets, plugs and switches and appliances only when they are in good condition and be sure they have the mark of BIS (ISI). (Necessity using BIS (ISI) marked accessories is explained under standardisation.
- Never extend electrical circuits by using temporary wiring.
- Stand on a wooden stool, or an insulated ladder while repairing live electrical circuits/appliances or replacing fused bulbs. In all the cases, it is always good to open the main switch and make the circuit dead.
- Stand on rubber mats while working/ operating switch panels, control gears etc.
- Position the ladder, on firm ground.
- While using a ladder, ask the helper to hold the ladder against any possible slipping.
- Always use safety belts while working on poles or high rise points.
- Never place your hands on any moving part of rotating machine and never work around moving shafts or pulleys of motor or generator with loose shirt sleeves or dangling neck ties.
- Only after identifying the procedure of operation, operate any machine or apparatus.
- Run cables or cords through wooden partitions or floor after inserting insulating porcelain tubes.
- Connections in the electrical apparatus should be tight. Loosely connected cables will heat up and end in fire hazards.
- Use always earth connection for all electrical appliances along with 3-pin sockets and plugs.
- While working on dead circuits remove the fuse grips; keep them under safe custody and also display 'Men on line' board on the switchboard.
- Do not meddle with inter locks of machines/switch gears
- Do not connect earthing to the water pipe lines.
- Do not use water on electrical equipment.
- Discharge static voltage in HV lines/equipment and capacitors before working on them.

Games to improve your memory

Objectives : At the end of this lesson you shall be able to
- **list the benefit of playing memory games**
- **describe what is a crossword pussle**
- **brief what is sudoku game**
- **highlights the importance of chess game.**

As an organic storage device, the brain compares pretty favourably to digital drives. It's believed to be able to retain up to 2.5 pet bytes of information-that's akin to nearly three million hours of television. Being able to retrieve all of that knowledge, however, is a different matter. To help the process along, we've compiled a list of games that can help sharpen both your long- and short-term memory.

Do a Crossword-Just Not Too Often

Crossword puzzles remain the go-to cognitive exercise, with studies showing a strong correlation between the habit and delayed onset of dementia. But the problem with crosswords as a memory and overall brain booster is that you can get too good at them: If you're able to put your pen down in record time, you're probably not giving yourself enough of a challenge. Try alternating crosswords with other word-search games to keep your brain engaged.

Sudoku

This number game has been a staple of newspapers for years, and for good reason. By having to keep a series of numbers in your head while mentally "rehearsing" their placement in the nine-space grids, you're relying heavily on working memory. But bear in mind, Sudoku is believed to be most effective early on, before your brain has gotten used to organizing the numbers. If it gets easier, it's time to try something new.

Chess

Chess is one of the most intellectually challenging games around, though newer players often rely on short-term memory in order to analyze the board and plot their next move on the spot. More experienced players have committed strategies to their long-term memory, shifting the exercise to retaining information for the long haul.

Motivation

Objectives : At the end of this lesson you shall be able to
- **define the concept of motivation**
- **list the types of motivation in teaching and training**
- **analyse the result of reinforcement in learning motivation**
- **explain the motivational functions of instructor in modifying the behaviour of learners.**

The terms ‘motivation’ is derived from the word “motive” which may be defined as needs, wants, drives or impulses within an individual.

Motivation is the hidden force within us which impels us to behave in a particular way. The inducement to the people to contribute effectively and efficiently as possible is called motivation.

Human motives are based on needs, desire, wishes, drives, interests inclination, purpose etc. Thus, motivation is an act of energising and activating the people to satisfy their needs.

Motivation is the important factor in the learning process. It is important for an instructor to know the general principles of motivation.

In the field of teaching and training two fundamentally different types of motives may be distinguished.

“Intrinsic” motives which drives for knowledge and activity without the consideration of the result or consequence.

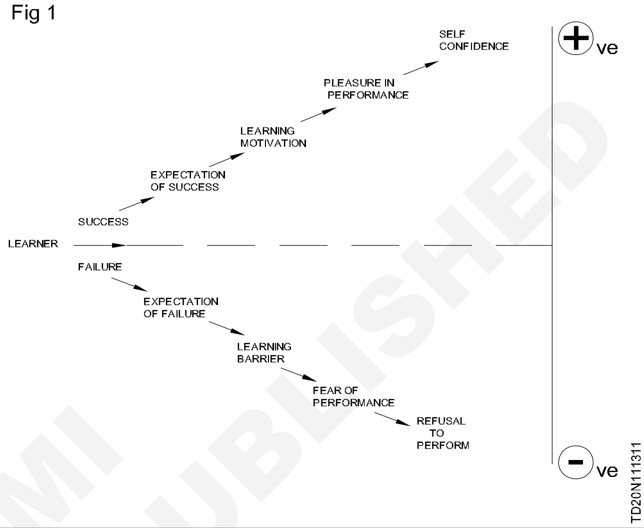
“Extrinsic” motives which are directed towards the achievement of goal such as learning a skill in order to earn money.

Trainer have a tendency to value intrinsic motivation more highly than extrinsic motivation. Extrinsic motivation, however, cannot be fully neglected and should not be looked separately.

Reinforcement

Reinforcement plays an important role in influencing the learning process. Reinforcement occurs when the learner can see that his efforts have led to success. A person who has such a pleasant experience of success will understandably have a tendency to repeat the activity again and again. In this way successful learning can lead to a continuous renewed willingness to learn. A person whose efforts have met with failure for long duration and frequently will have an expectation of failure before every new learning situation. This negative reinforcement acts as a barrier to learning. (Fig 1)

Fig 1



Behaviour modification

The behaviour modification techniques that a teacher or instructor can adopt includes.

- **Extinction :** Ignoring failures to reinforce “undesirable forms of behaviour or mistakes” by not noticing them.
- **Counter conditioning :** “Undesirable form of behaviour or mistakes” leads to an unpleasant experience for the learner. (Punishment, scolding, reprimands, marking mistakes with thick red pencils / pen, condemnation etc.)
- **Learning by imitation :** The teacher or instructor is himself a model or provides a behavioural model in some other way.

Motivational function of the trainer / instructor

From the available theory and data of the educational psychology, the teacher should perform four functions to motivate the learners, which are as follows:

Arousal function

It is to arouse and maintain learner’s interest. It involves the initial responsibility of winning the learner’s attention (readiness to receive the lesson) and the continuing responsibility of regulating the level of arousal to avoid both sleep and emotional eruption. To meet this, the trainer should make the lesson interesting to the learners by bringing them within the learner’s intellectual range and helping them to understand that it is worth and valuable. Changing the position, tone of voice, mood of the trainer, teaching valuable from one technique to another etc, all leads to increased motivation of learner.

Expectance function

It is to maintain or modify the learner’s expectation of success or failure in reaching the instructional objectives. It requires the trainer to describe concretely the learners what they will be able to do after the lesson.

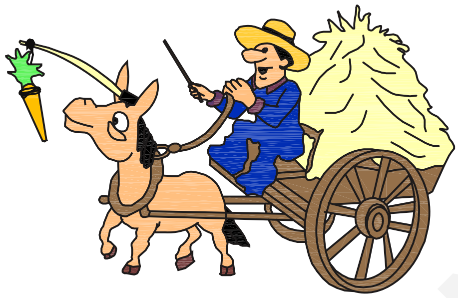
Incentive function

It is to encourage learner in his further effort in the pursuit of instructional objectives. Feedback of test results, spoken or written praise or blame, grading, competition and co-operation are some of the established methods as successful incentives for learners which increase learner's vigour in learning. However, care must be taken that these incentives do not come as discouragement to other groups of learners.

Disciplinary function

It is to control the deviant behaviour of the learners through the use of reward and punishment (Fig 2). A punishment procedure which involves both the suppression of undesired response (or behaviour) and the provision of an alternative reward for desired response (or behaviour) may be a most effective procedure. This artful combination of punishment and reward as a disciplinary technique is called "restitution". This technique should be adopted by the trainer in a non-threatening manner.

Fig 2



TD20N111312

Techniques of motivation

In order to create effective motivation, the trainer or instructor should avoid the precipitous path of negative reinforcement. In general attention should be paid to the consistent use of reinforcement to promote positive types of behaviour and achievement through encouragement. In appropriate behaviour and errors should be largely ignored. In addition, the following steps may be taken to create effective motivation:

- Unambiguous objectives and a clear view of the subject as a whole makes the learner's motivation increased.
- Connection with the learner's own interests and elucidation of ways in which the results of his learning can be put to practical use.
- Setting of tasks with a moderate degree of difficulty and a flexible range of difficulty. This results in a challenge to make an effort as well as the experience of success in learning.
- Interesting and challenging formulation of questions:
    a Creating surprise
    b Producing uncertainty
    c Raising objectives,
    d Provoking learners by taking an extreme point of view
    e Arousing doubts, setting conflicting requirements
- Stimulation of independent learning by extending the periods of independent activity during learning process. Appropriate methods include goup discussion, individual and group work, projects, role-playing etc.
- Feed back concerning learning success which encourages positive reinforcement of learning.

Learning starts with what learner knows, and not what the instructor / teacher knows or starts.

"Learning is change in behaviour"

- What learning means.
- Focal area in psychology of learning.
- Needs of learner.
- Learning materials.
- Learning methods.
- Readiness to learn.
- Methods of Instruction.
- Learning by interaction.
- Environment.
- Psychological factors.
- Types of learning.
- Factors affecting learning process.

What learning means

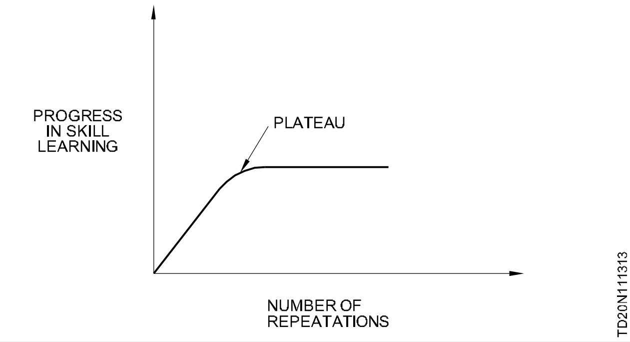
Learning is a process by which an individual acquires various habits, knowledge, skill and attitude that are required for meeting certain objectives. Learning changes the behaviour of individuals.

For simple job initial learning is fast and the learning curve shows steep rise (Fig 3). The curve becomes flat as learning slow down. For simple tasks, steep initial rise in performance and study of flattening of performance is the main objective for complex skill, there may be number of flattening which refers to hold up or fatigue in the process.

Learning situation

Learning situation that provides necessary atmosphere required for the acquisition of knowledge, attitudes and skills. If any one of these is not satisfactory it affects the effective learning. The learning situation provides opportunity for learning. The learning situation determines the quality and speed to provide good learning situation to the learner.

Fig 3



Needs of learner

Learning can take place in response to the needs of the learner. When the learner knows the needs of learning, the learning will be more effective. The instructor should display their needs, then only they can make them to start learning.

Learning materials (content)

It is the content to be learnt by the learner. It should be according to the mental level of the trainees. The content should be presented in a language that could be understood by the learner. If the content is in their own mother tongue / Regional language the learning will be more effective.

The learning methods

Comprehensive (understanding) learning is better that cursory (Hasty) learning. Whole learning is better than part learning. Self evaluation, periodical revision, repetition of learning and altering modes (manner) of learning are much important. Learning by trial and error needs good amount of energy. The method of learning also depends on the age maturity.

Readiness for learning

For learning readiness is the most important factor. Learning is based on physical, social, intellectual and emotional growth. It is very important that the learners readiness should be ascertained before starting learning process. The instructor should know the maturity back ground of the learners for certain kind of learnings. In mass education teaching, this is not possible to achieve. Readiness of all trainees may vary from each other as such some may be ready and some may not be and this kind of individual differences can't be avoided.

Method of instruction

The instructional methods affects the learning process. If the instructional method is not related to the requirements of learner, the learning will be ineffective. The method of instruction should be in cognizance (knowledgeable) to the instructional objectives. The method of teaching should not be mechanical. The active participation of the trainees in the teaching, accelerates the absorption of what trainees learnt. For effective and faster learning teaching and training aids may be used. Training aids help the learner to retain better from what the trainees have learnt. Remedial instructions helps the backward trainers.

Learning by interaction

When a learner feels the needs of the objectives to be learn and to achieve the goal set for themself they learn by interacting in their learning situation. This process of responding to the situation enables them feeling satisfaction for what they have learnt.

Environment

Physical, residential and social environment is very important. Outside the institute the trainee must have facility to apply the knowledge and skill. It is necessary for instructor and the institute administration to develop and create healthy and congenial environment which can bring about good qualities and character in the trainees. The environment helps the trainees in future to become good craftsman and better citizens.

Psychological factors

Learning is a process involving communication between the trainee and instructor. Cordial relationship provide security and trainees attention. Psychological factor helps improving self-expression, self ascertain and satisfaction. Personality requirements are quite important for efficient learning. Discipline, attitude of instructor, ability of administrators and involvement of all these connections are equally important for providing better learning conditions.

Bench vice

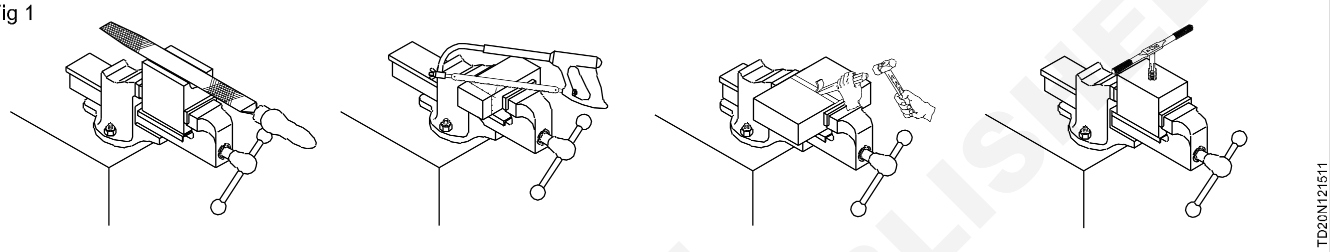
Objectives: At the end of this lesson you shall be able to
- **state the uses of vice clamps**
- **explain the size of the bench vice**
- **state the parts of the bench vice**
- **explain the care and maintenance of vices.**

Vice

Vices are used for holding workpieces. They are available in different types. The vice used for bench work is the bench vice. (Engineer's vice)

A bench vice is made of cast iron or cast steel and it is used to hold work for filing, sawing, threading and other hand operations. (Fig 1)

Fig 1



The size of the vice is stated by the width of the jaws.

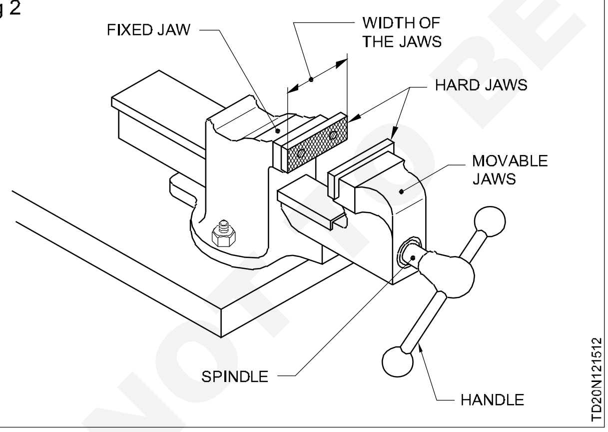
Parts of a bench vice (Fig 2)

The following are the parts of a vice.

Fixed jaw, movable jaw, hard jaws, spindle, handle, box nut and spring.

The box nut and the spring are the internal parts.

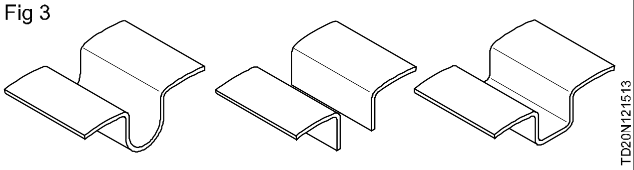
Fig 2



Vice clamps or soft jaws (Fig 3)

To hold a finished work use soft jaws (vice clamps) made of aluminium over the regular hard jaws. This will protect the work surface from damage.

Fig 3



Do not over-tighten the vice as, otherwise, the spindle may be damaged.

Care and maintenance of vices

- Always keep all threaded and moving parts clean by wiping the vice with a cloth after each use.
- Make sure to oil and lubricate the joints and sliding parts.
- To oil the sliding section, open the jaws completely and apply a layer of grease to the screen.
- Remove the rust if appears on the vice using rust remover chemical.
- When the vice is not in use bring the jaws lightly gap together and place the handle in a vertical position.
- Avoid striking the handle of the vice by a hammer for tightening fully, otherwise the handle will become bend or damaged.

Positioning of bench vice

Vices are mounted rigidly on a work bench with the fixed jaw in line with the bench edge to permit a long work to be clamped in a vertical position (Fig 4).

For the convenience of working, the vice should be held at a correct height i.e. when the fist is pressed against the chin the elbow should touch the top of the vice. For further height adjustments, wooden platforms can be used.

The correct height of a bench vice is shown in Fig 5.

Precautions

Clamp the work as low as possible on the vice. Do not give extra leverage while tightening the work.

Lubricate the spindle and the box-nut periodically. Do not tighten the jaws of he vices without any work in between

Do not hammer on the vices for levelling metal

Fig 5

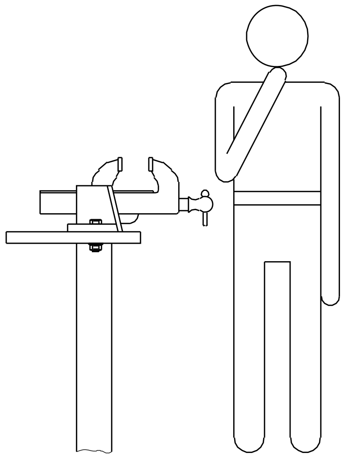
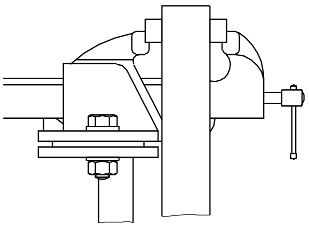


Fig 4



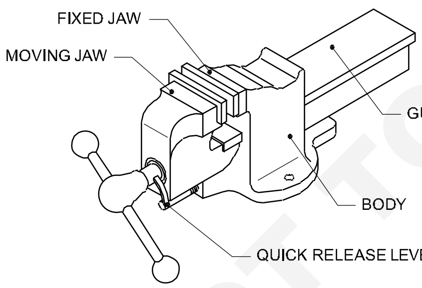
Types of vices

Objectives: At the end of this lesson you shall be able to
- **state the different types of vices**
- **state the uses of quick relasing vice, pipe vice, hand vice, pin vice and leg vice.**

There are different types of vices used for holding work-pieces. They are quick releasing vice, pipe vice, hand vice, pin vice and toolmaker's vice.

Quick releasing vice (Fig 1)**:** A quick releasing vice is similar to an ordinary bench vice but the opening of the movable jaw is done by using a trigger (lever). If the trigger at the front of the movable jaw is pressed, the nut disengages the screw and the movable jaw can be set in any desired place quickly.

Fig 1



Pipe vice (Fig 2)**:** A pipe vice is used for holding round sections of metal, tubes and pipes. In the vice, the screw is vertical and movable. The jaw works vertically.

The pipe vice grips the work at four points on its surface. The parts of a pipe vice are shown in Fig 2.

Hand vice (Fig 3)**:** Hand vices are used for gripping screws, rivets, keys, small drills and other similar objects which are too small to be conveniently held in the bench vice. A hand vice is made in various shapes and sizes. The length varies from 125 to 150 mm and the jaw width from 40 to 44 mm. The jaws can be opened and closed using the wing nut on the screw that is fastened to one leg, and passes through the other.

Fig 2

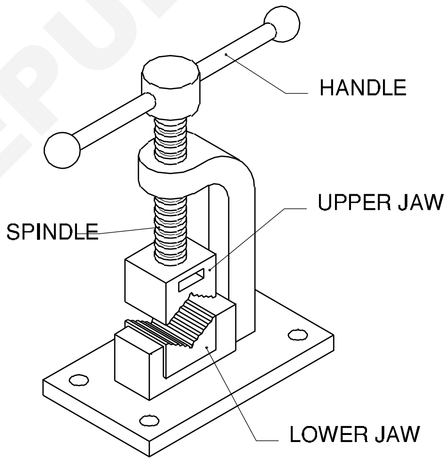
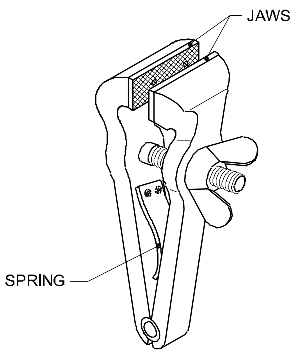
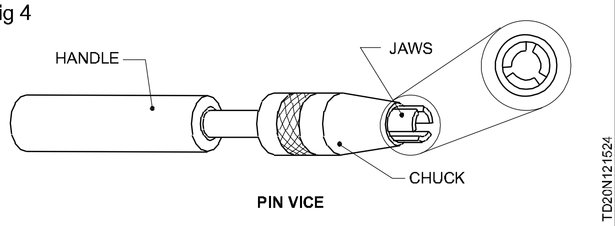


Fig 3



Pin vice (Fig 4)**:** The pin vice is used for holding small diameter jobs. It consists of a handle and a small collet chuck at one end. The chuck carries a set of jaws which are operated by turning the handle.

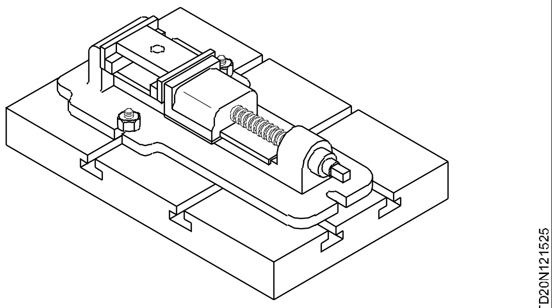
Fig 4



Toolmaker's vice (Fig 5)**:** The toolmaker's vice is used for holding small work which requires filing or drilling and for marking of small jobs on the surface plate. This vice is made of mild steel.

Toolmaker's vice is accurately machined.

Fig 5



Leg vice

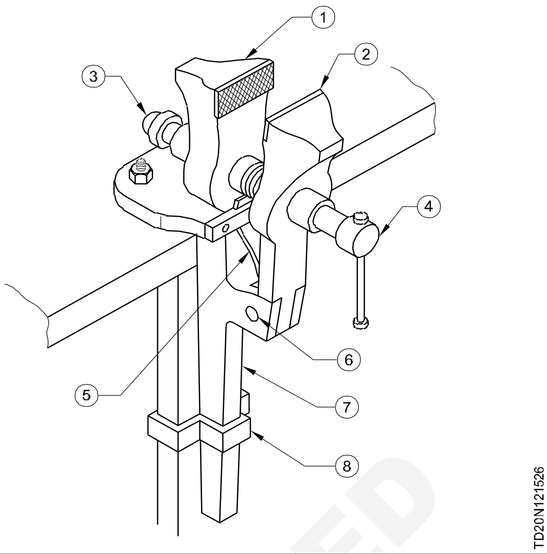
A leg vice is a holding device generally used in a forge shop for bending and forging work. It is made fo mild steel to avoid breakage while hammering.

Main pats of a leg vice (Fig 6**)**

The following are the main parts of a leg vice.

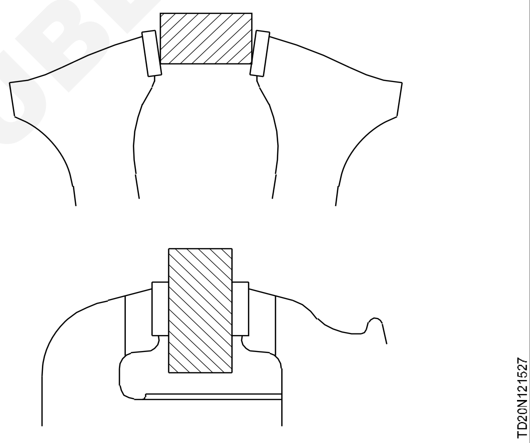
1	Solid jaw	2	Movable jaw
3	Threaded jaw	4	Spindle
5	Spring	6	Pivot
7	Leg	8	Clamp

Fig 6



Since the hinged jaw moves in a radial path, the job held in this vice is not gripped properly because of the line contact. (Fig 7) Hence a work which can be carried out on a bench vice is not held on a leg vice. Jobs which require hammering only are held on a leg vice.

Fig 7



Hacksaw frames and blades

Objectives: At the end of this lesson you shall be able to
- **explain the different types of hacksaw frames**
- **explain the different type of hacksaw blades**
- **describe the method of sawing**

Hacksaw frame: A hacksaw frame is used along with a blade to cut metals of different sections, and is specified by the type and maximum length of the blade that can be fixed.

Example

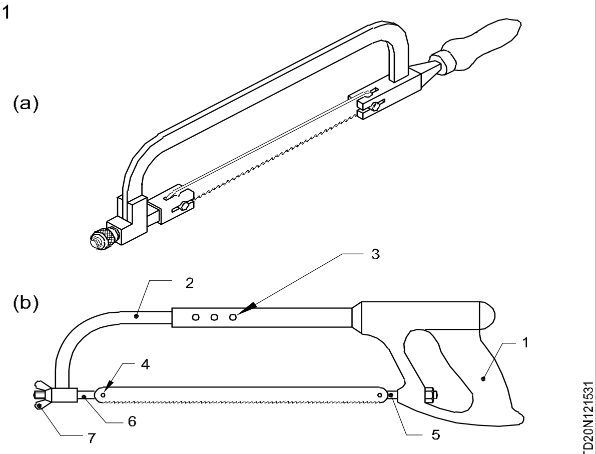
Adjustable hacksaw frame - tubular - 250 - 300mm or 8" - 12"

Types of hacksaw frames

Solid frame (Fig 1a)**:** Only a blade of a particular standard length can be fitted to this frame. e.g 300 mm or 250 mm.

Adjustable frame (flat type)**:** Different standard lengths of blades can be fitted to this frame i.e. 250 mm and 300 mm.

Fig 1



Adjustable frame (tubular type) (Fig 1b)**:** This is the most commonly used type. It gives a better grip and control, while sawing.

Parts of a hacksaw frame

1 Handle

2 Frame

3 Tubular frame with holes for length adjustment

4 Retaining pins

5 Fixed blade-holder

6 Adjustable blade-holder

7 Wing-nut

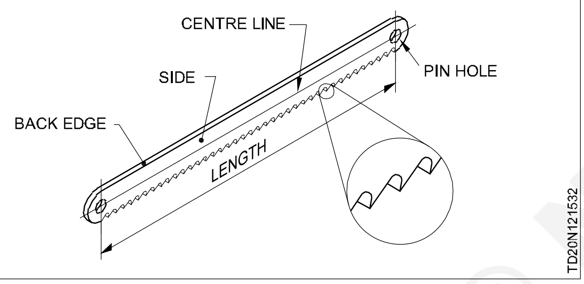
A hacksaw blade is made of either low alloy steel (LA) or high speed steel (HSS), and is available in standard lengths of 250 mm and 300mm. (Fig 2)

Parts of a hacksaw blade (Fig 2)

1 Back edge 2 Side

3 Centre line 4 Pin holes

Fig 2



Type of hacksaw blades

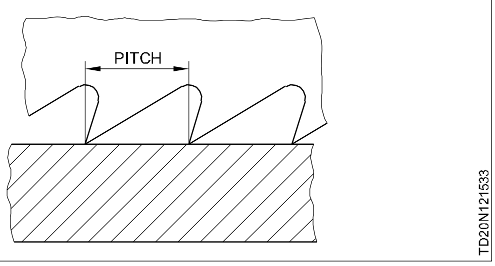
All-hard blade: The full length of the blade between the pins is hardened and it is used for harder metals such as tool steel, die steel and HCS.

Flexible blade: Only the teeth are hardened. Because of their flexibility these blades are useful for cutting along curved lines. Flexible blades should be thinner than all-hard blades.

Pitch of the blade (Fig 3)**:** The distance between adjacent teeth is known as the 'pitch' of the blade.

Classification	Pitch
Coarse	1.8 mm
Medium	1.4 mm & 1.0 mm
Fine	0.8 mm

Fig 3



Specification: Hacksaw blades are specified by the length, pitch and type of material. (The width and thickness of blade is standardised)

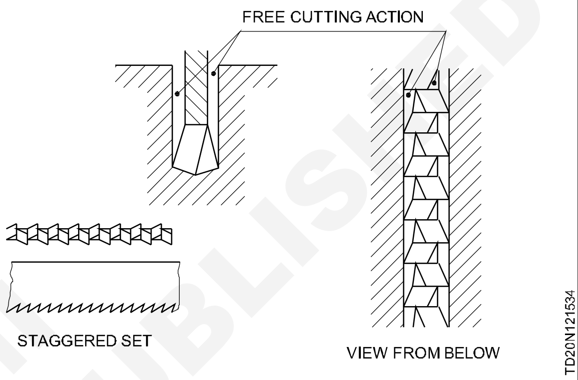
Example

300 x 1.8 mm pitch LA all-hard blade.

To prevent the hacksaw blade binding when penetrating into the material, and to allow free movement of the blade, the cut is to be broader than the thickness of the hacksaw blade. This is achieved by the setting of the hacksaw teeth. There are two types of hacksaw teeth settings.

Staggered set (Fig 4)**:** Alternate teeth or groups of teeth are staggered. This arrangement helps for free cutting, and provides for good chip clearance.

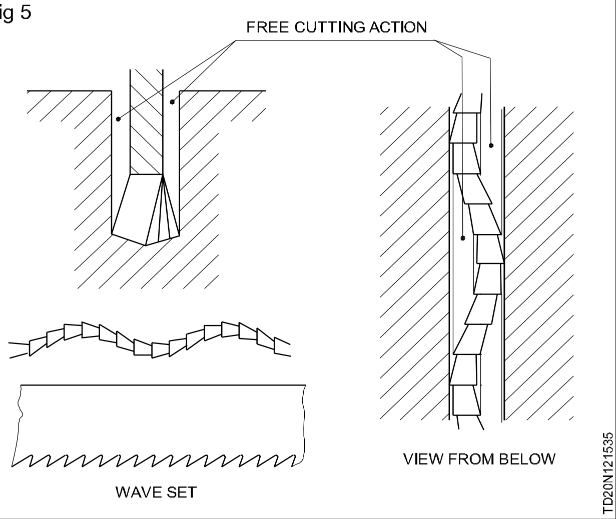
Fig 4



Wave set (Fig 5)**:** In this, the teeth of the blade are arranged in a wave-form. The types of sets for different pictures are as follows:

Pitch	Type of set
0.8 mm	Wave-set
1.0 mm	Wave-set or staggered
Over 1.0 mm	Staggered

Fig 5

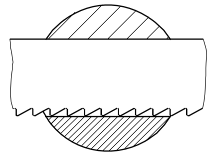


For the best results, the blade with the right pitch should be selected and fitted correctly.

Selection of blade: The selection of the blade depends on the shape and hardness of the material to be cut.

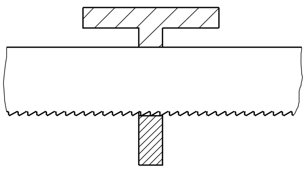
Pitch selection (Fig 6)**:** For soft materials such as bronze, brass, soft steel, cast iron, heavy angles etc. use a 1.8 mm pitch blade.

Fig 6



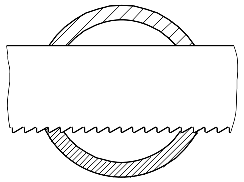
For tool steel, high carbon, high speed steel etc. use a 1.4 mm pitch. For angle iron, brass tubing, copper, iron pipe etc. use a 1 mm pitch blade. (Fig 7)

Fig 7



For conduit and other thin tubing, sheet metal work etc. use a 0.8 mm pitch. (Fig 8)

Fig 8



Method of sawing

Select the correct blade for the material to be cut.

HSS - Blades are used for tough resistant materials

High Carbon Steel - General cutting

Select the correct number of teeth / inch the general rule is that atleast 3 teeth should extend across the surface of the material to be cut.

The hand holds the hacksaw handle, and the index finger is support the handle and also points in the direction of cutting.

The other hand holds the frame, near the wing nut. Cutting/ sewing should be carried out close to the jaws of the vice. This ensures that the metal does not flex or bend under the force of the hacksaw and the sawing motion.

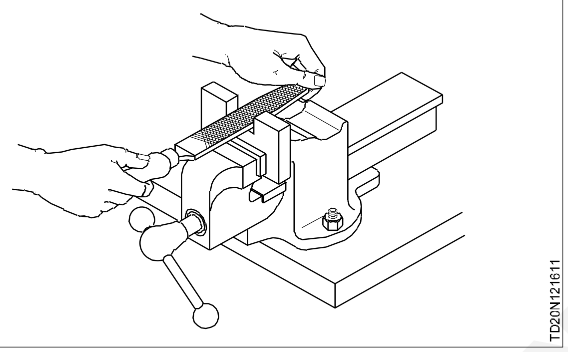
Elements of a file

Objectives: At the end of this lesson you shall be able to
- **state what is file in fitting shop**
- **state the parts of a file**
- **state the material of a file.**

File: A File is a hand cutting tool used to remove fine amount of material from a workpiece. It is commonly used in woodand metal working.

Filing is a method for removing excess material from a workpiece by using a file which acts as a cutting tool. Fig 1 shows how to hold a file. Files are available in many shapes and sizes.

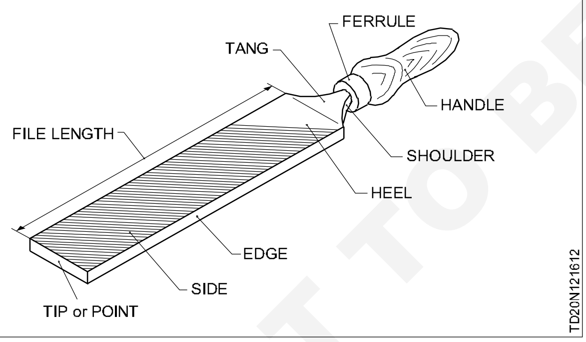
Fig 1



Parts of a file (Fig 2)

The parts of a file can be seen in Fig 2, are

Fig 2



Tip or Point

the end opposite to tang

Face or side

The broad part of the file with teeth cut on its surface

Edge

The thin part of the file with a single row of parallel teeth

Heel

The portion of the broad part without teeth

Shoulder

the curved part of the file separating tang from the body

Tang

The narrow and thin part of a file which fits into the handle

Handle

The part fitted to the tang for holding the file

Ferrule

A protective metal ring to prevent cracking of the handle.

Materials

Generally files are made of high carbon or high grade cast steel. The body portion is hardened and tempered. The tang is however not hardened.

Types of files

Objectives: At the end of this lesson you shall be able to
- **explain the different shape of files(types)**
- **state the uses of flat files, Hand files square, round, half round, triangular and knife-edge files**
- **state the correct shape of files for filing different profiles.**

For filing and finishing different profiles, files of different shapes are used

The shape of files is stated by its cross section.

Common files of different shapes: Flat file, Hand file, Square file, Round file, Half round file, Triangular file and Knife-edge file.

Flat file (Fig 1)

These files are of a rectangular cross section. The edges along the width of these files are parallel up to two-thirds of the length, and then they taper towards the point. The faces are double cut, and the edges single cut. These files are used for general purpose work. They are useful for filing and finishing external and internal surfaces.

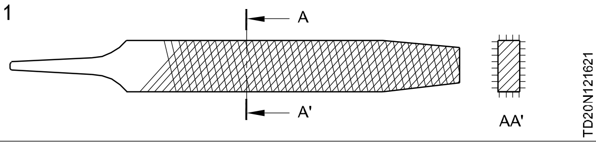


Fig 1

Hand file (Fig 2)

These files are similar to the flat files in their cross section. The edges along the width are parallel throughout the length. The faces are double cut. One edge is single cut whereas the other is safe edge. Because of the safe edge, they are useful for filing surfaces which are at right angles to surfaces already finished.

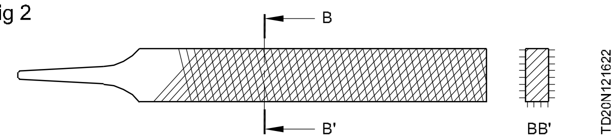


Fig 2

> **Flat files are general purpose files. They are available in all grades. Hand files are particularly useful for filing at right angles to athe finished surface.**

Square File: The square file is square in its cross section. It is used for filing square holes, internal square corners, rectangular openings, keyways and splines. (Fig 3)

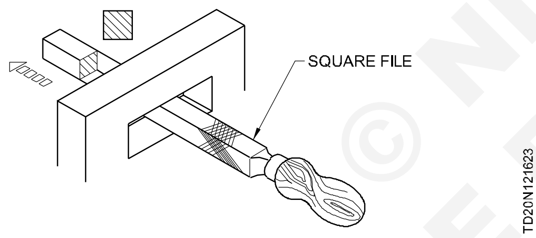


Fig 3

Round file: A round file is circular in its cross section. It is used for enlarging the circular holes and filing profiles with fillets. (Fig 4)

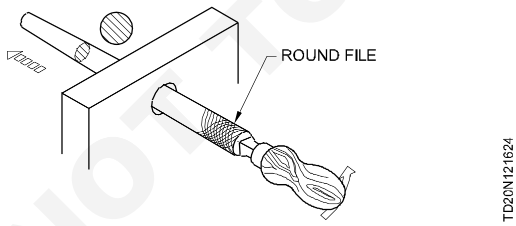


Fig 4

Half round file: A half round file is in the shape of a segment of a circle. It is used for filing internal curved surfaces. (Fig 5)

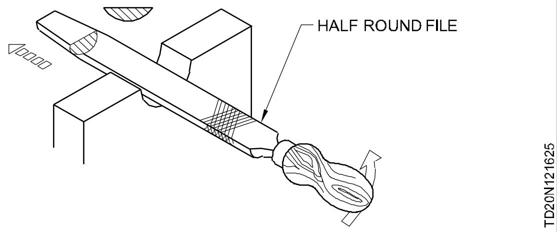


Fig 5

Knife edge file: A knife edge file has the cross section of a sharp triangles. It is used for filing narrow grooves and angles above 10° (Fig 6)

The above files have one third of their lengths tapered. They are available both single and double cuts.

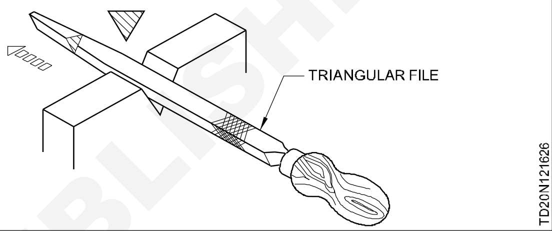


Fig 6

Triangular file: A triangular file is of a triangular cross section. It is used for filing corners and angles which are more than 60°. (Fig 7)

Square, round, half-round and triangular files are available in lengths of 100, 150, 200, 250, 300 and 400mm. These files are made in bastard, second cut and smooth grades.

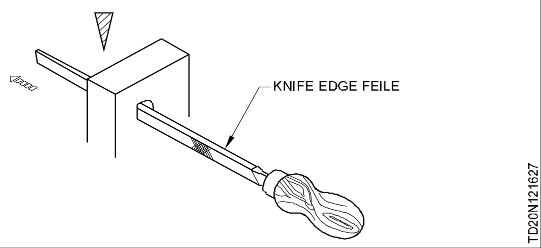


Fig 7

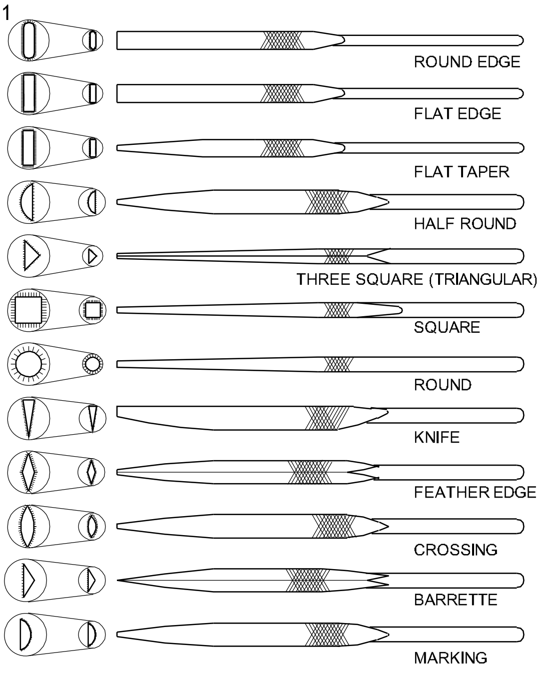
Needle files

Objectives: At the end of this lesson you shall be able to
- **explain the different shapes of needle files**
- **state the needle files as per BIS.**

Needle files are usually available in sets with assorted shapes. These types of files are used for delicate, light kinds of work. These files are available in bastard and smooth grade.

Shapes: The common shapes of needle files are shown in figure 1. The shapes are round edge, flat edge, flat taper, half round, triangular, square, round, knife, feather edge, crossing, barret and marking. (Fig 1)

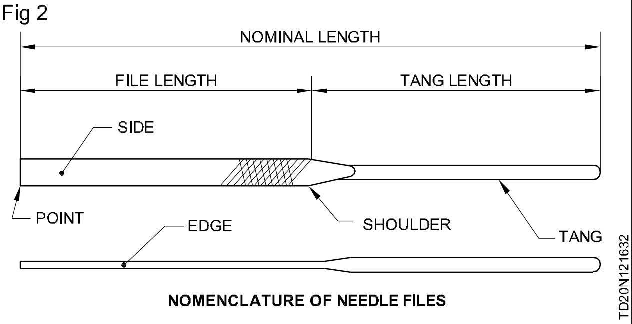
Fig 1



Nomenclature of needle files. (Fig 2)

Length: These files are available in a nominal length of 120mm to 180mm.

Fig 2



Grades: The grades of cut may be identified by the cut number as follows

- bastard - Cut 0.
- smooth - Cut 2.

Designation of needle files: The needle files are designated by their names

- grade of cut
- nominal length
- BIS number

Example

A flat edge needle file with grade of cut bastard, having a nominal length of 160mm shall be designated as flat edge needle file bastard, 160 IS 3152

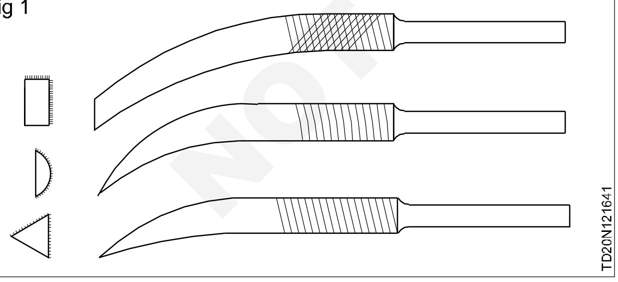
Special files

Objectives: At the end of this lesson you shall be able to
- **explain the different types of special files**
- **state the uses of each type of special files.**

In addition to the common type of files, files are also available in a variety of shapes for 'special' applications. These are as follows.

Riffler files (Fig 1): These files are used for die-sinking, engraving and in silversmith's work. They are made in different shapes and sizes and are made with standard cuts of teeth.

Fig 1



Mill saw files (Fig 2): Mill saw files are usually flat and have square or rounded edges. These are used for sharpening teeth of wood-working saws, and are available in single cut.

Crossing file (Fig 3): This file is used in the place of a half round file. Each side of the file has different curves. It is also known as 'fish back' file.

Fig 2

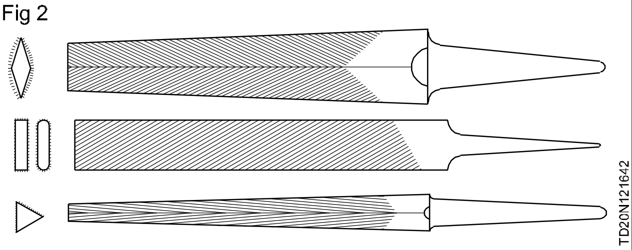
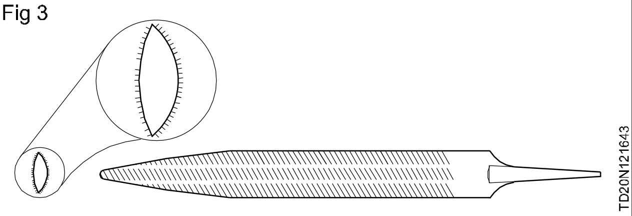


Fig 3



Barrette file (Fig 4): This file has a flat, triangular face with teeth on the wide face only. It is used for finishing sharp corners.

Tinker's file (Fig 5): This file has a rectangular shape with teeth only at the bottom face. A handle is provided on the top. This file is used for finishing automobile bodies after tinkering.

Fig 4

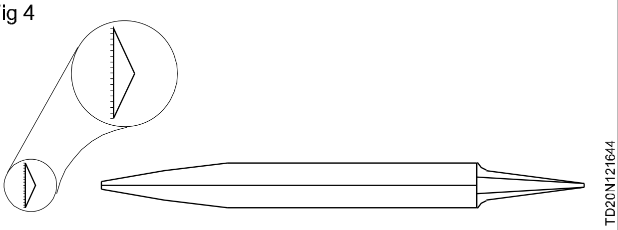
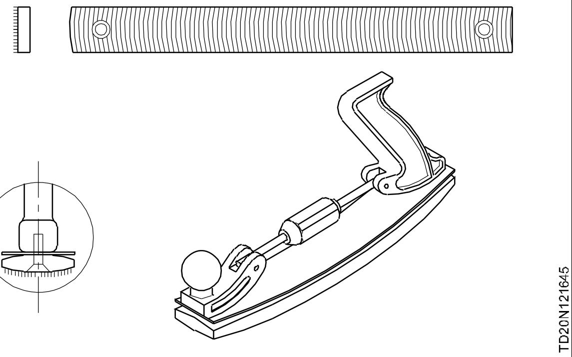


Fig 5



Rotary files (Fig 6): These files are available with a round shank. They are driven by a special machine with a portable motor and flexible shaft. These are used in diesinking and mould-making work.

Machine files for hand filing machine (Fig 7)**:** Machine files are of double cut, having holes or projections to fix to the holder of the filing machine. The length and shape will vary according to the machine capacity. These files are suitable for filing the inner and outer surfaces, and are ideal for diesinking and other tool-room work.

Fig 6

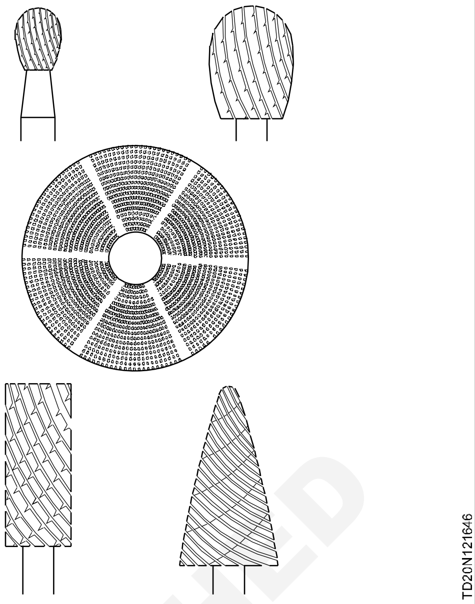
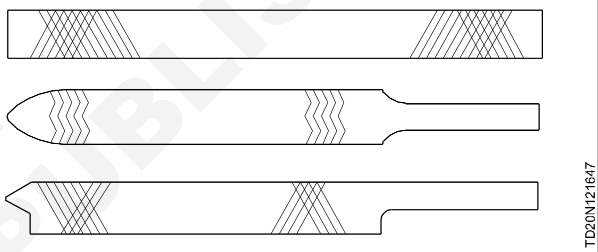


Fig 7



Pinning of files

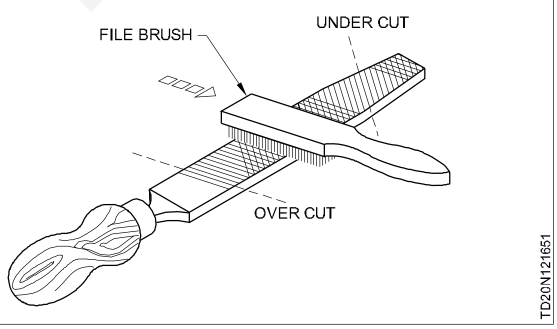
Objective: At the end of this lesson you shall be able to
- **explain the pinning of file.**

During filing, sometimes the metal chips (filings) will clog between the teeth of files. This is known as ‘pinning’ of files.

Files which are pinned will produce scratches on the surface being filed, and also will not bite well.

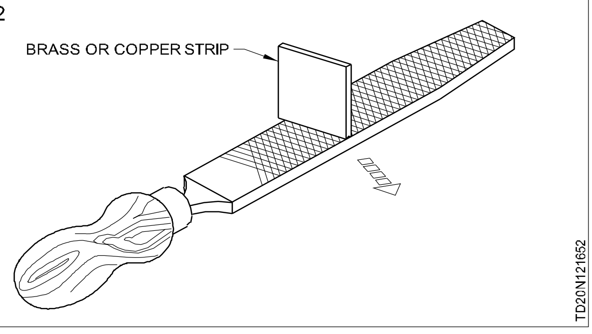
Pinning of the files is removed by using a file brush also called a file card, (Fig 1) with either forward or backward stroke.

Fig 1



Filings which do not come out easily by the file card should be taken out with a brass or copper strip. (Fig 2)

Fig 2



For new files, use only soft metal strips (brass or copper) for cleaning. The sharp cutting edges of the files will wear out quickly if a steel file card is used. When filing a workpiece to a smooth finish more ‘pinning’ will take place because the pitch and depth of the teeth are less.

Application of chalk on the face of the file will help reduce the penetration of the teeth and 'pinning'.

Clean the file frequently in order to remove the filings embedded in the chalk powder.

Care and maintenance of file

Objective: At the end of this lesson you shall be able to
- **state the care and maintenance of file.**

- Do not use files having the blunt cutting edge
- Remember that files cut on the push stroke. Never apply the pressure on the pull stroke, or you could crush the file teeth, blunt them or cause them to break off.
- Prevent from pinning.
- Giving your files teeth a light brush with oil during long storage.
- Normally do not apply any oil while filing.
- Files should be stored separatelly so that their faces cannot rub against each other or against other tools.

Cut of files

Objectives: At the end of this lesson you shall be able to
- **explain the different cuts of files**
- **state the uses of each type of cut.**

The teeth of all file are formed by cuts made on its face. Files have cuts of different types. Files with different cuts have different uses.

Types of cuts

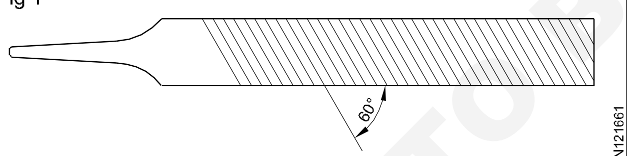
Basically there are four types.

Single cut, Double cut, Rasp cut and Curved cut.

Single cut file (Fig 1)

A single cut file has rows of teeth cut in one direction across its face. The teeth are at an angle of 60^0 to the centre line. It can cut chips as wide as the cut of the file. Files with this cut are useful for filing soft metals like brass, aluminium, bronze and copper.

Fig 1

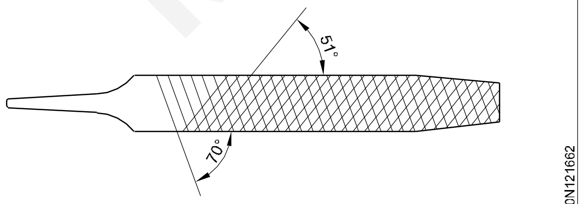


SINGLE CUT FILE

Single cut files do not remove stock as fast double cut files, but the surface finish obtained is much smoother.

Double cut file (Fig 2)

Fig 2



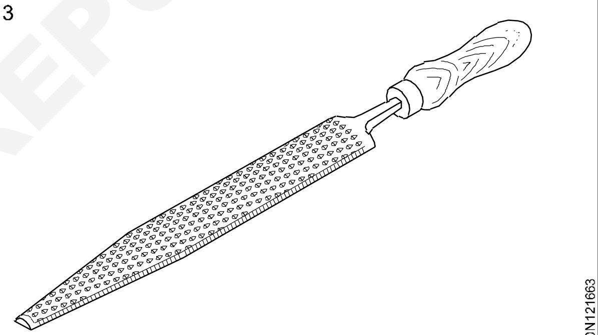
DOUBLE CUT FILE

A double cut file has two rows of teeth cut diagonal to each other. The first row of teeth is known as OVERCUT and they are cut at an angle of 70^0. The other cut, made diagonal to this, is known as UPCUT, and is at an angle of 51^0. This removes stock faster than the single cut file.

Rasp cut file (Fig 3)

The rasp cut has individual, sharp, pointed teeth in a line, and is useful for filing wood, leather and other soft materials. These files are available only in half round shape.

Fig 3

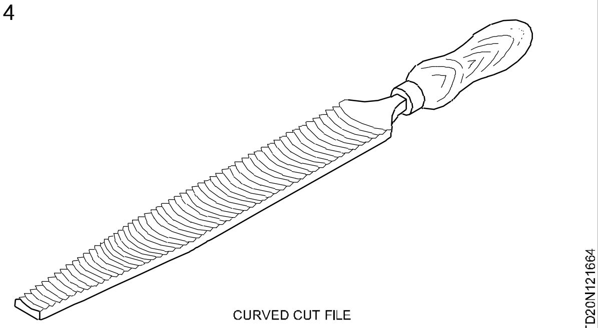


RASP CUT FILE

Curved cut file (Fig 4)

These files have deeper cutting action and are useful for filing soft materials like - aluminium, tin, copper, and plastic.

Fig 4



CURVED CUT FILE

The curved cut files are available only in a flat shape.

The selection of a file with a particular type of cut is based on the material to be filed. Single cut files are used for filing soft materials. But certain special files, for example, those used for sharpening saws, are also of single cut.

File specifications and grades

Objectives: At the end of this lesson you shall be able to
- **state how files are specified**
- **explain the different grades of files**
- **state the application of each grade of file.**

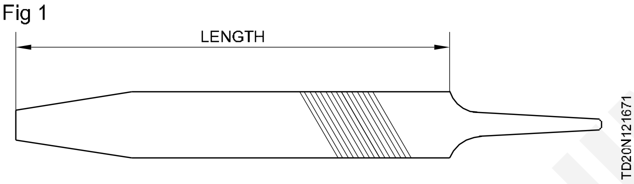
Files are manufactured in different types and grades to meet the various needs.

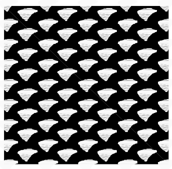
Files are specified according to their length, grade, cut and shape.

Length is the distance from the tip of a file to the heel.

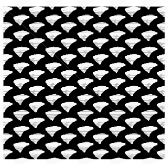
File grades are determined by the spacing of the teeth.

Fig 1

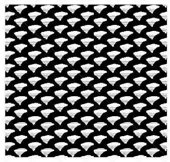




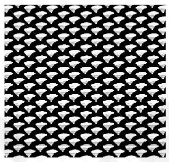
A **rough file** is used for removing rapidly a larger quantity of metal. It is mostly used for trimming the rough edges of soft metal castings.



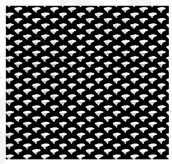
A **bastard file** is used in cases where there is a heavy reduction of material



A **second cut file** is used to give a good finish on metals. It is excellent to file hard metals. It is useful for bringing the jobs close to the finishing size.



A **smooth file** is used to remove small quantity of material and to give a good finish.



A **dead smooth** file is used to bring the material to accurate size with a high degree of finish.

The most used grades of files are bastard, second cut, smooth and dead smooth.These are the the grades recommended by the bureau of indian standards (BIS)

Different sizes of files with the same grade will have varying sizes of teeth. In longer files, the teeth will be coarser.

The number of cutting edge in rows in each of the above grades over a Length of 10mm as shown in Table (1).

Table (1)

Grade of files (Number of cuts over the length of 10mm)					
Length of file	**Rough**	**Bastard**	**Second cut**	**Smooth**	**Deadsmooth**
150mm	8	13	17	24	33
200mm	7	11	16	22	31
250mm	6	10	15	20	30
300mm	5	9	14	19	28

Hammers

Objectives: At the end of this lesson you shall be able to
- **state the uses of an engineer's hammer**
- **identify the parts of an engineer's hammer**
- **name the types of engineer's hammer**
- **specify the engineer's hammer.**

An engineer's hammer is a hand tool used for striking purposes while punching, bending, straightening, chipping, forging or rivetting.

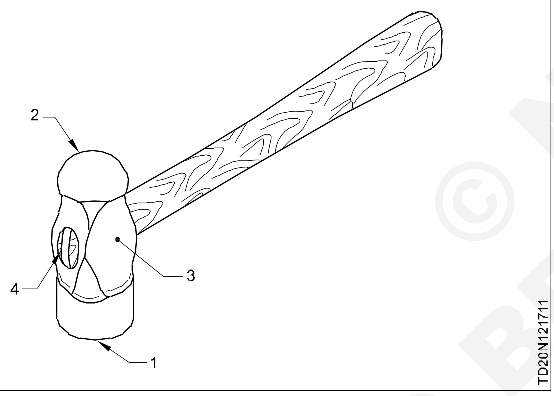
Major parts of a hammer: The major parts of a hammer are the head and the handle.

Hammer is made of drop - forged carbon steel, while the wooden handle must be capable of absorbing shock.

The parts of a hammer - head are face (1), pein (2) cheek (3) and the eyehole (4).

Face: The face is the striking portion. A slight convexity is given to it to avoid digging of the edge. It is used for striking while chipping, bending, punching, etc.

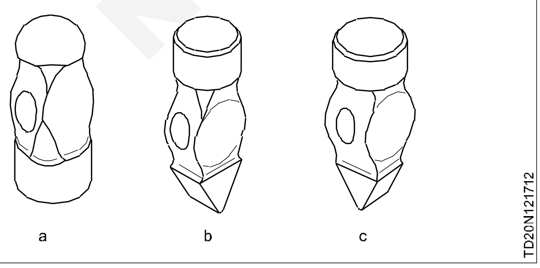
Fig 1



Pein: The pein is the other end of the head. It is used for shaping and forming work like riveting and bending.The pein is of different shapes such as:

- ball pein (Fig.2a)
- cross - pein (Fig.2b)
- straight pein. (Fig 2c)

Fig 2



The face and the pein are case hardened.

Cheek: The cheek is the middle portion of the hammer head. The weight of the hammer is stamped here.

This portion of the hammer - head is left soft.

Eyehole: The eyehole is meant for fixing the handle. It is shaped to fit the handle rigidly. The wedges fix the handle in the eyehole. (Figs 3 and 4)

Fig 3

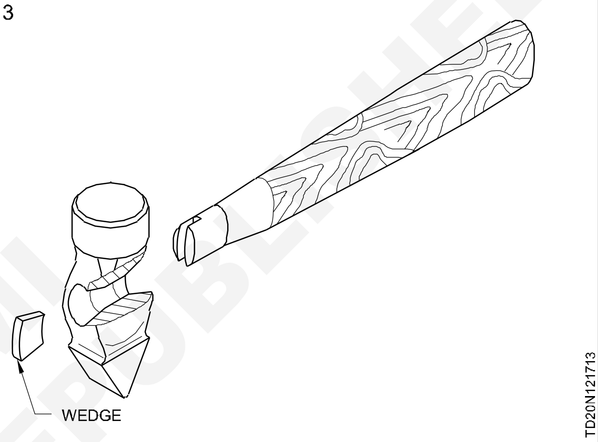
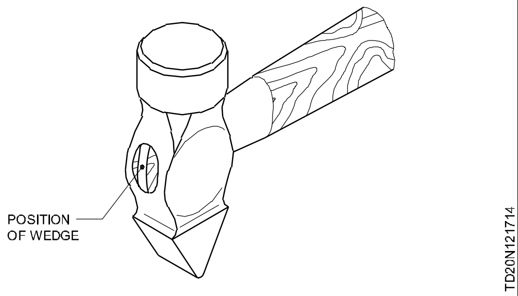
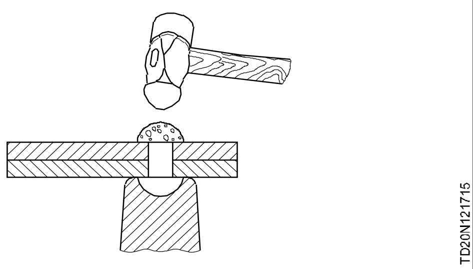


Fig 4



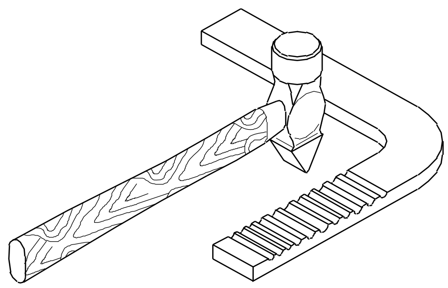
Application of hammer pein: The ball pein is used for riveting. (Fig 5)

Fig 5



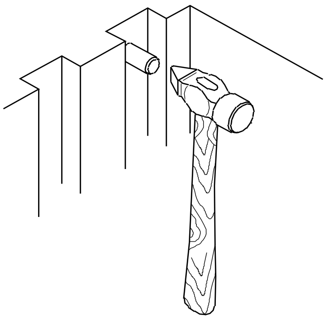
The cross- pein is used for spreading the metal in one direction. (Fig 6)

Fig 6



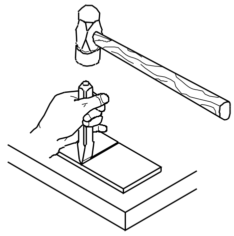
The straight pein is used at the corners. (Fig 7)

Fig 7



The ball pein hammer is used for driving a chisel in parting metal. (Fig 8)

Fig 8



Specification: An engineer's hammers are specified by their weight and the shape of the pein. Their weight varies from 125 gms to 750 gms.

The weight of an engineer's hammer, general work in a machine/ fitting shop.

The ball pein hammers are used for general work in a machine/ fitting shop.

Before using a hammer

- make sure the handle is properly fitted
- select a hammer with the correct weight suitable for the job
- check the hammer head and handle whether any crack is there
- ensure that the face of the hammer is free from oil or grease.

Screwdriver

Objectives: At the end of this lesson you shall be able to
- **state the different types of screwdrivers and their uses**
- **specify a screwdriver**
- **list the precautions to be observed while using screwdrivers.**

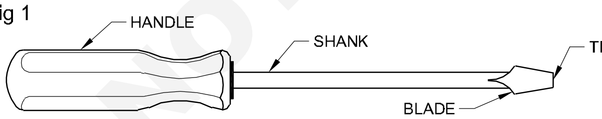
Screwdrivers are used to tighten or loosen screws and are available in various lengths.

Hand-held screwdrivers are of the following types.

Standard screwdriver (Light duty) (Fig 1)

It is made of a round shank/blade with metal, wood or moulded insulated material handle.

Fig 1



Standard screwdriver (Heavy duty) (Fig 2)

The shank is square section for applying extra twisting force with the end of a spanner. (Fig 3)

Heavy duty screwdriver (London pattern) (Fig 4)

It has a flat blade and is mostly used by carpenters for fixing and removing wood screws.

Fig 2

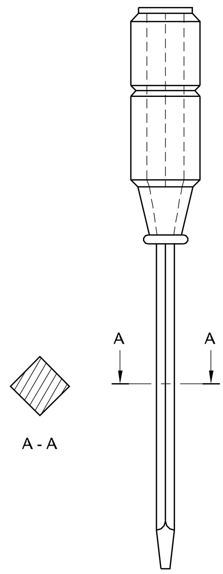


Fig 3

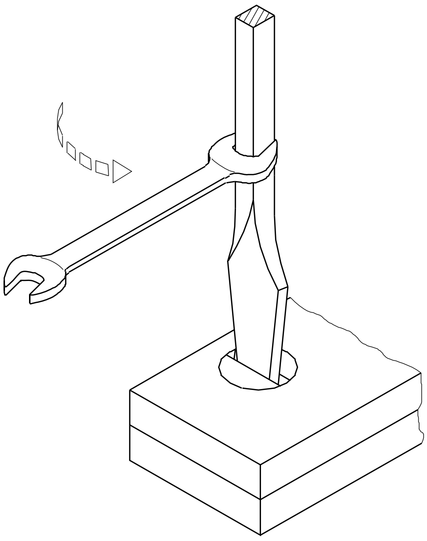
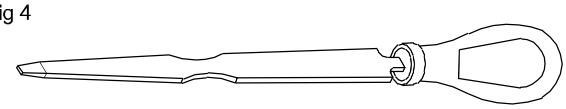
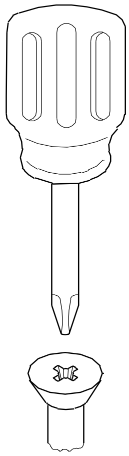


Fig 4



Philips screwdriver (Fig 5)

Fig 5



These are made with cruciform (Fig 6) tips that are unlikely to slip from the matching slots. (Fig 7) Philips recess head screws are shown in Fig 8.

Fig 6

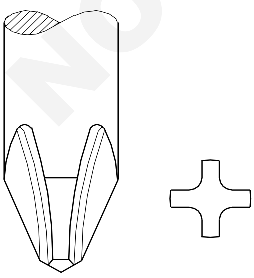


Fig 7

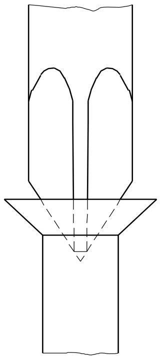
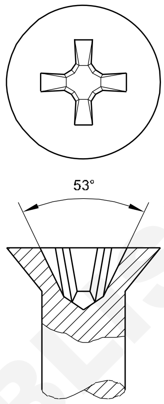


Fig 8



The sizes of philips scredrivers are specified by point size 1,2,3 and 4.

Offset scerwdrivers (Fig 9)

These are useful in some situations (Fig 10) where the normal screwdriver cannot be used because of the length of the handle. They are also useful for applying greater turning force.

Fig 9

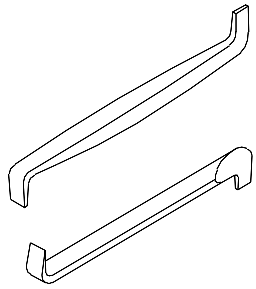
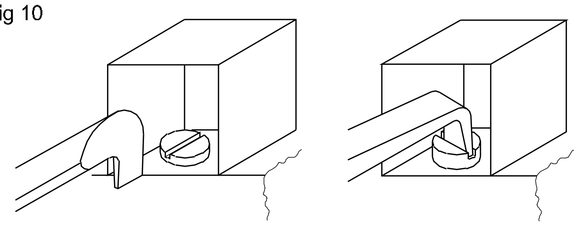
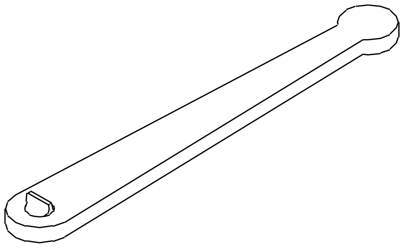


Fig 10



For quicker application ratchet offset screwdrivers are also available with renewable tips. (Fig 11)

Fig 11

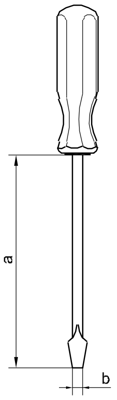


Specification

Screwdrivers (Fig 12) are specified according to the

- length of the blade
- width of the tip

Fig 12



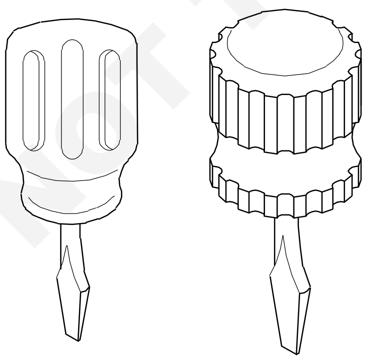
Normal blade length: 45 to 300 mm. Width of blade: 3 to 10 mm

The blades of screwdrivers are made of carbon steel or alloy steel, hardened and tempered.

Screwdrivers for special uses

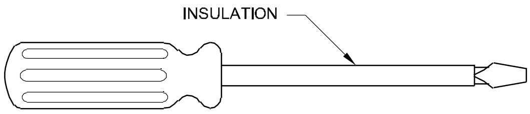
Small sturdy screwdrivers ar available for use wher there is limited space. (Fig 13)

Fig 13



Srewdrivers with blades sheathed in insulation are available for the use of electricians. (Fig 14)

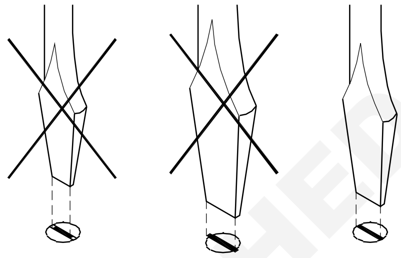
Fig 14



Precautions

Use screwdrivers with tips correctly fitting into the screw slot. (Fig 15)

Fig 15



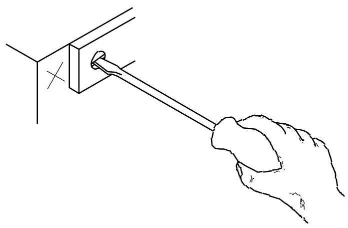
Make sure your hand and the handle are dry.

Hold the screwdriver axis in line with the axis of the screw.

While using a Philips scredriver apply more downward pressure.

Keep your hand away to avoid injury due to slipping of the screwdriver. (Fig 16)

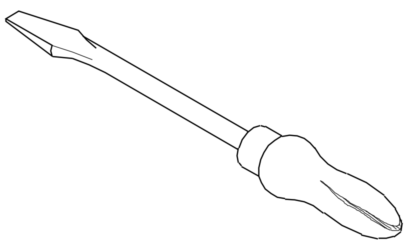
Fig 16



Do not use screwdrivers with split or defective handles. (Fig 17)

In the case of damaged screwdrivers, the blades can be ground (the faces will be parallel with the sides of the screw slot) and used. While grinding endure the end of the tips is as thick as the slot of the screw.

Fig 17

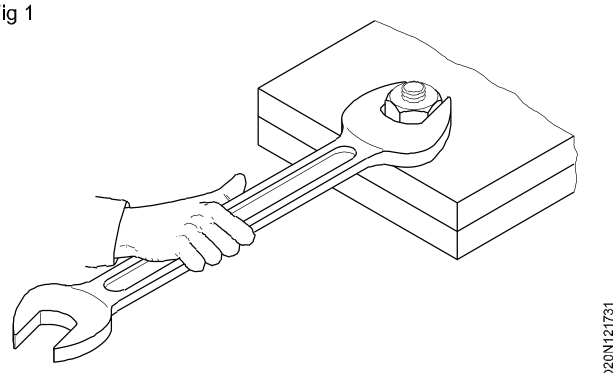


Spanners

Objectives: At the end of this lesson you shall be able to
- **state the uses of spanners of different sizes**
- **identify the size of a spanner.**

A spanner is a hand tool with jaws or opening or a ring at one end or at both ends or at both ends for tightening or slackening nuts and bolts and screw heads. (Fig 1) It is made of drop forged, high tensile or alloy steel and heat treated for strength.

Fig 1



Types of spanners

- Open and spanners
- Ring spanners

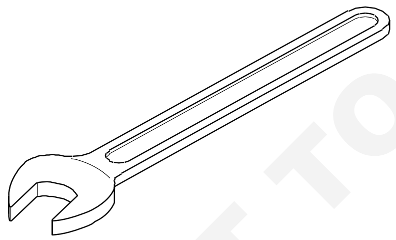
Open and spanners

They can be single-ended or double-ended.

Single-ended spanners (Fig 2)

These are general purpose spanners. Single-ended spanners are mostly supplied with machine tools for a specific purpose.

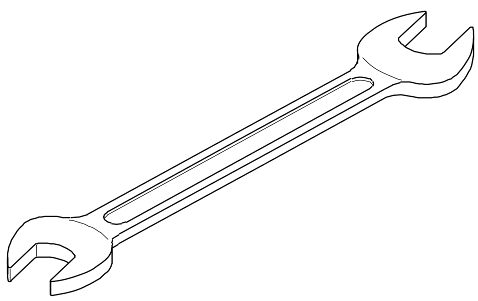
Fig 2



Double-ended spanners (Fig 3)

Double-ended spanners are standard spenners having two different size openings. Some spanners are made of chrome vanadium steel.

Fig 3



They are available in a set of 8, Nos 8 to 27 mm. (Fig 3) 8x10, 9x11, 12x13, 14x15, 16x17, 18x19, 20x22 and 24x27 mm.

Open and spanners bigger than 27 mm size are also available.

Ring spanners (Fig 4,5 & 6)

Fig 4

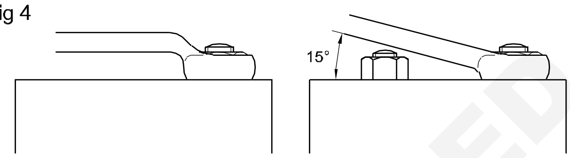


Fig 5

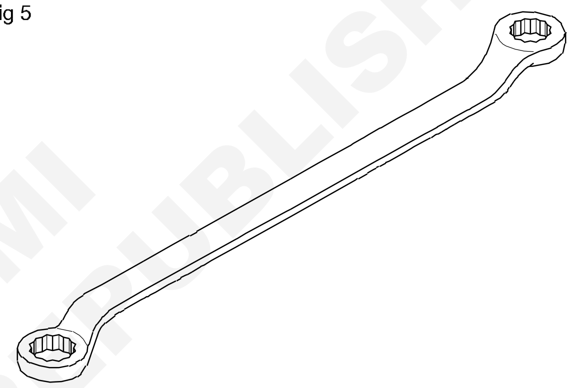
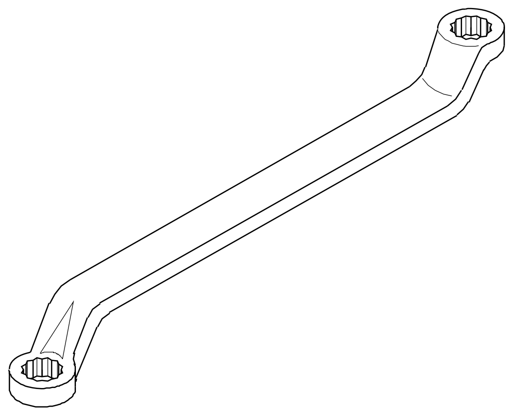


Fig 6



These types of spanners are used where obstruction close to the side of a nut prevails (Fig 4) and application of open-ended spanners is not possible.

These are available in a set of 8 Nos. (8 to 27 mm) 8x9, 10x11, 12x13, 14x15, 16x17, 18x19, 20x22 and 24x27 mm.

Sizes and identification of spanners

Spanners for metric bolts, nuts and screws are marked with the size across the jaw opening in mm.

Special purpose spanners

- Tube or tubular box spanners (Figs 7 & 8)

Fig 7

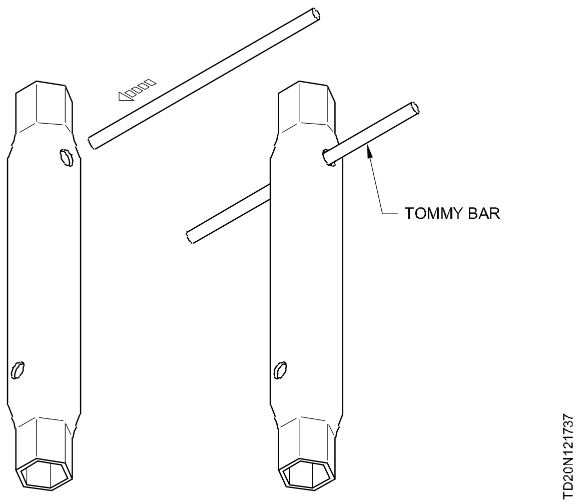
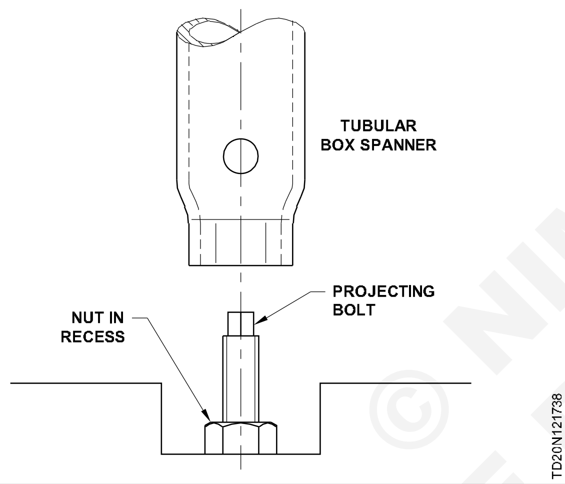


Fig 8



- Socket spanners (Fig 9)
- Adjustable spanners (Figs 10&11)
- Hook spanners (C-spanner) (Figs 12 & 13)

Fig 9

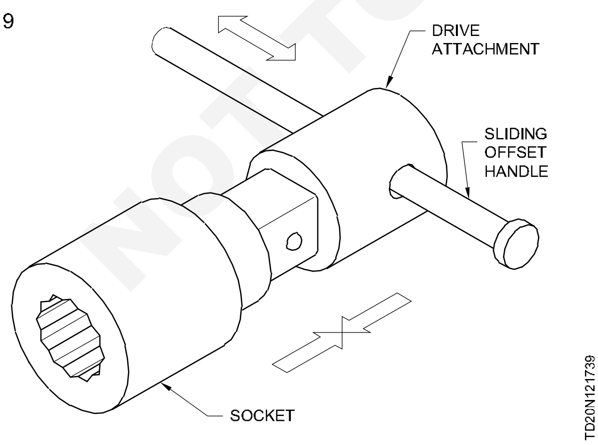


Fig 10

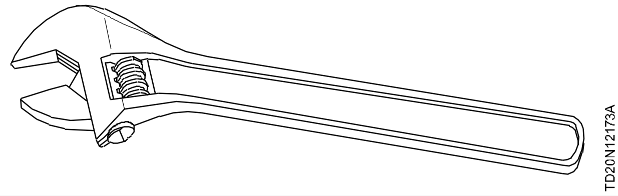


Fig 11

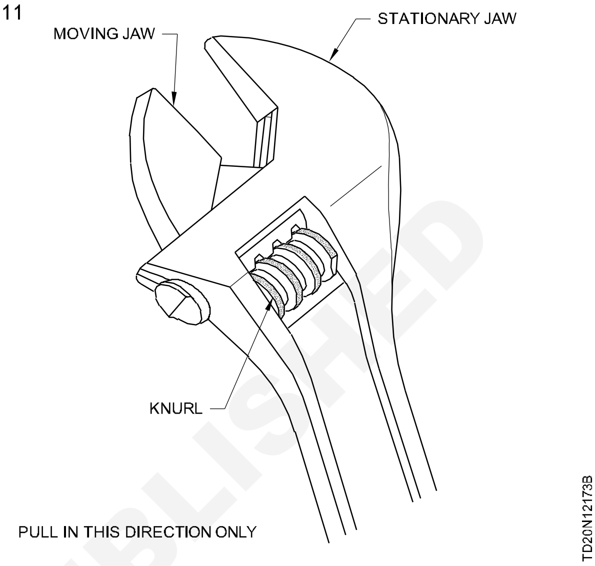


Fig 12

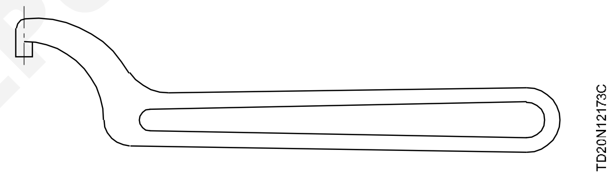
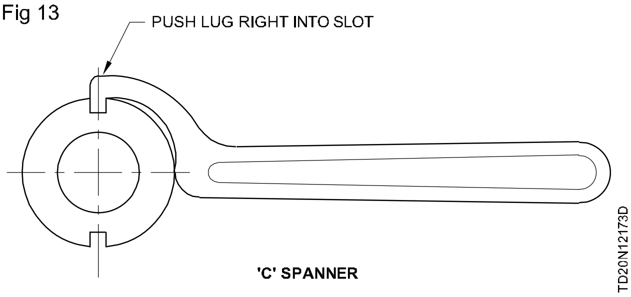


Fig 13



Linear measurement

Objectives: At the end of this lesson you shall be able to
- **explain the base unit of linear measurement as per the international system of units of measurement (SI)**
- **describe the measuring standards of english and metric units**
- **state the multiples of a metre and their values**
- **state the purpose of steel rule**
- **state the types of steel rule**
- **state the precautions to be followed while using a steel rule.**

Necessity

All physical quantities are to be measured in terms of standard quantities.

Unit

A unit is defined as a standard or fixed quantity of one kind used to measure other quantities of the same kind.

Classification

Fundamental units and derived units are the two classifications.

Fundamental units

Units of basic quantities of length, mass and time.

Derived units

Units which are derived from basic units and bear a constant relationship with the fundamental units.

Ex : Area, Volume, Pressure, Force, etc.

System of units

F.P.S. system is the British system in which the basic units of length, mass and time are foot, pound and second respectively.

C.G.S. system is the metric system in which the basic units of length, mass and time are centimetre, gram and second respectively.

M.K.S system is another metric system in which the basic units of length, mass and time are metre, kilogram and second respectively.

S.I. units is referred to as Systems International units which is again of metric and the basic units, their names and symbols are Listed in table - 1

Fundamental units and derived units are the two classification of units.

Length, mass and time are the fundamental units in all the systems (ie) F.P.S, C.G.S, M.K.S and S.I systems.

Table 1

Basic Quantity	Metric Unit		British unit	
	Name	Symbol	Name	Symbol
Length	Metre	m	Foot	F
Mass	Kilogram	kg	Pound	P
Time	Second	S	Second	S
Current	Ampere	A	Ampere	A
Temperature	Kelvin	K	Farenheit	F°
Light intensity	Candela	Cd	Candela	Cd

When we measure an object , we are actually comparing it with a known standard of measurement.

The base unit of length as per SI units is METRE.

Length - SI UNITS and MULTIPLES

Base unit

The base unit of length as per the International Systems of units (SI) is metre. The table given below lists some multiples of a metre.

METRE(m) = 1000 mm

CENTIMETRE (μm) = 10 mm

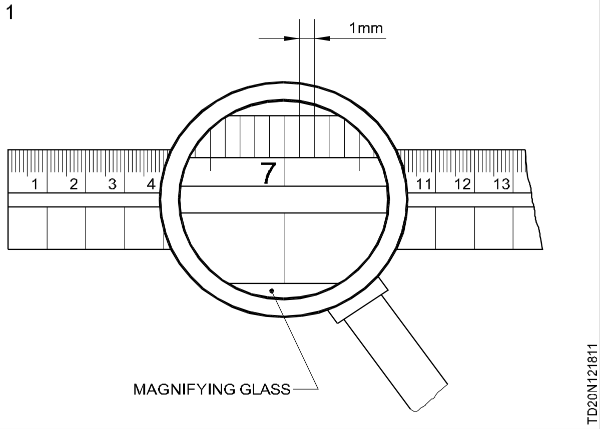
MILLIMETRE (mm) = 1000 μm

MICROMETRE (μm) = 0.001 mm

Measurement in engineering practice

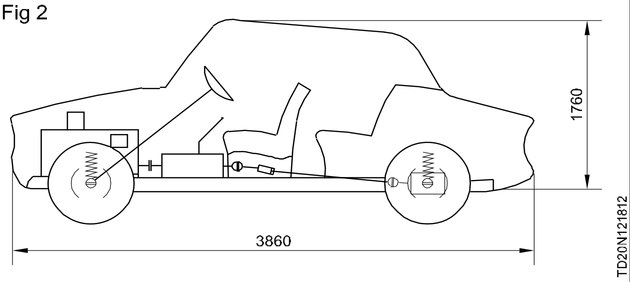
Usually, in engineering practice, the preferred unit of length measurement is millimetre. (Fig 1)

Fig 1



Both large and small dimensions are stated in millimetres. (Fig 2)

Fig 2

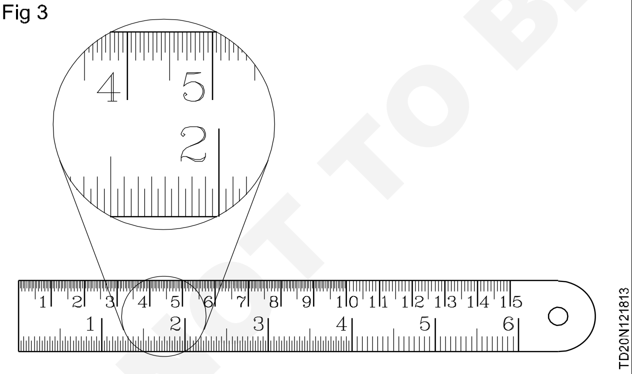


The British system of length measurement

An alternative system of length measurement is the British system. In this system, the base unit is the Imperial Standard yard. Most countries, including Great Britain itself, have, however, in the last few years, switched over to SI units.

Engineer's steel rule (Fig 3) are used to measure the dimensions of work pieces.

Fig 3



Steel rules are made of spring steel or stainless steel. These rules are available in length 150mm, 300mm and 600mm. The reading accuracy of steel rule is 0.5 mm and 1/64 inch.

For accurate reading it is necessary to read vertically to avoid errors arising out of parallax. (Fig 4)

Steel rule in English measure, they can also be available with metric and English graduation in a complete range of sizes 150, 300, 500 and 1000 mm. (Fig 5)

Fig 4

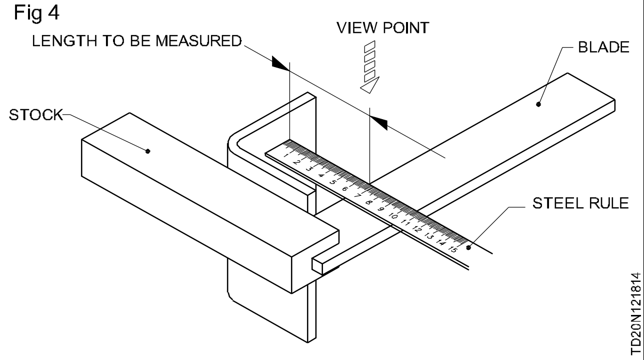
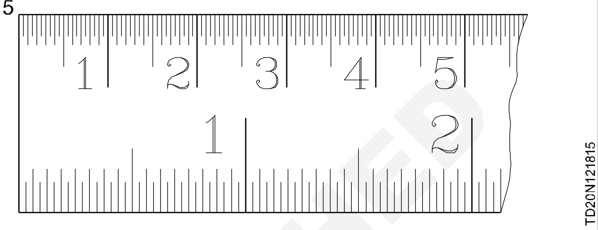


Fig 5



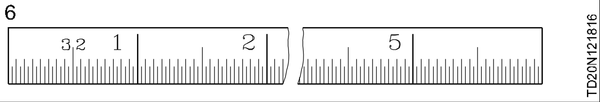
Other types of rule

- Narrow steel rules
- Short steel rules
- Full flexible steel rule with tapered end.

Narrow steel rule

Narrow steel rule is used to measure the depth of keyways and depth of smaller dia, blind holes of jobs, where the ordinary steel rule can not reach. Its width is approximately 5 mm and thickness 2 mm. (Fig 6)

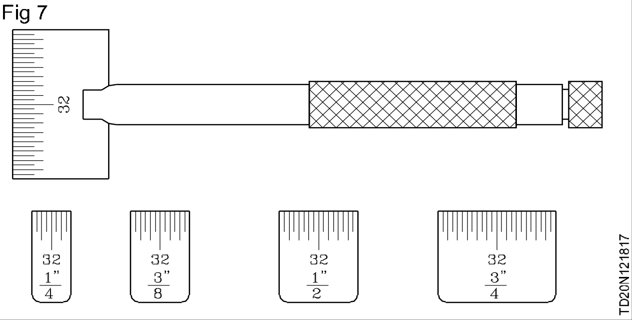
Fig 6



Short steel rule (Fig 7)

This set of five small rules together with a holder is extremely usefull for measurements in confined or hard to reach locations which prevent the use of ordinary steel rules. It is used suitably for measuring grooves, short shoulder, recesses, key ways etc. In machining operation on shapers, millers and tool and die work.

Fig 7

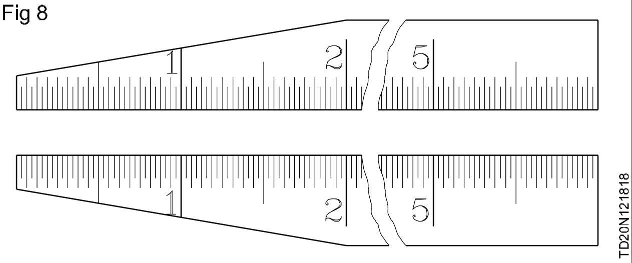


The rules are easily inserted in the slotted end of the holder and are rigidly clamped in place by a slight turn of the knurled nut at the end of the handle. Five rule lengths are provided 1/4", 3/8" and 1" and each rule is graduated in 32^{nds} on one side and 64ths on the reverse side.

Steel rule with tapered end

This rule is a favourate with all mechanics since its tapered end permits measuring of inside size of small holes, narrow slots, grooves, recesses etc. This rule has a taper from 1/2 inch width at the 2 inch graduation to 1/8 inch width at the end. (Fig 8)

Fig 8



For maintaining the accuracy of a steel rule, it is important to see that its edges and surfaces are protected from damage and rust.

> **Do not place a steel rule with other cutting tools. Apply a thin layer of oil when not in use.**

Angular measurement

Angular measurement of angles of an object is usually expressed in degrees, minutes and seconds. One degree is divided into 60 minutes and one minute is to 60 seconds.

Measurements of fundamental, derived units

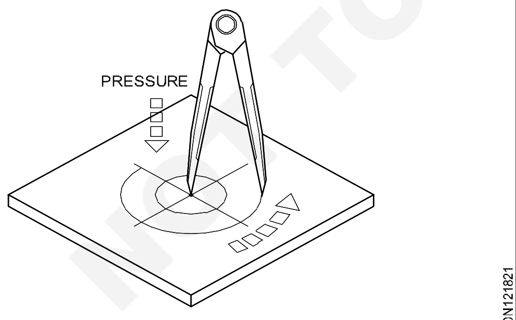
Metric	British
Micron 1μ = 0.001 mm	Thousandth of an inch = 0.001”
Millimetre 1 mm = 1000μ	Inch = 1”
Centimetre 1 cm = 10 mm	Foot 1 ft = 12”
Decimetre 1 dm = 10 cm	Yard 1 yd = 3 ft
Metre 1 m = 10 dm	1 furlong 1 fur = 220 yds
Decametre 1 dam = 10 metre	1 mile = 8 furlong

Dividers

Objectives : At the end of this lesson you shall be able to
- **state the parts of a divider**
- **state the uses of dividers**
- **state the specifications of divider**
- **state the important hints on divider points.**

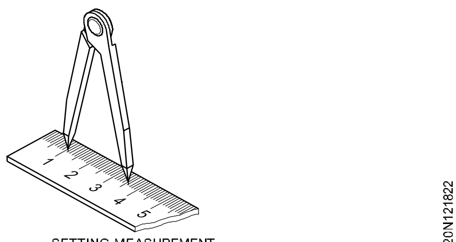
Dividers are used for scribing circles, arcs and for transferring and stepping off distances. (Fig 1, 2 and 3)

Fig 1



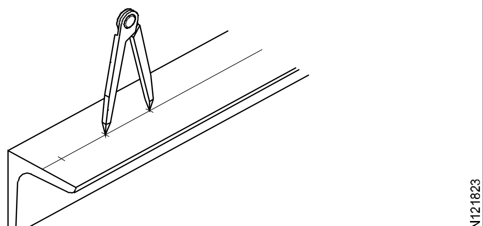
SCRIBING CIRCLES

Fig 2



SETTING MEASUREMENT

Fig 3



STEPPING DISTANCES

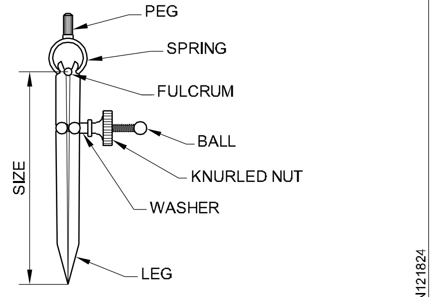
Dividers are available with firm joints and spring joints. (Figs 1 & 4). The measurements are set on the dividers with a steel rule. (Fig 2)

The sizes of dividers range between 50mm to 200mm.

The distance from the point to the centre of the fulcrum roller (pivot) is the size of the divider. (Fig 4)

For the correct location and seating of the divider point prick punch marks of 30° are used.

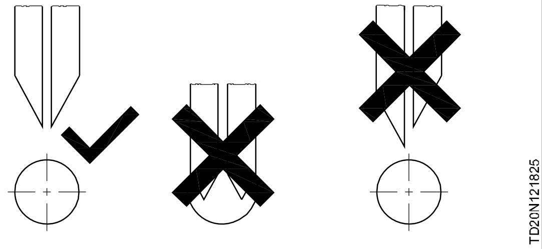
Fig 4



SPRING JOINT DIVIDER

The two legs of the divider should always be of equal length. (Fig 5) Dividers are specified by the type of their joints and length.

Fig 5



The divider point should be kept sharp in order to produce fine lines. Frequent sharpening with an oilstone is better than sharpening by grinding. Sharpening with an oilstone is better than sharpening by grinding. Sharpening by grinding will make the points soft.

Calipers

Objectives: At the end of this lesson you shall be able to
- **state the commonly used calipers**
- **state the advantages of spring joint calipers.**

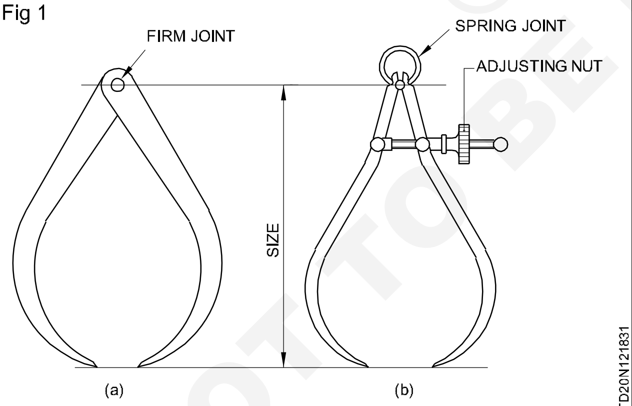
Calipers are indirect measuring instruments used for transferring measurements from a steel rule to a job, and vice versa.

Calipers are classified according to their joints and their legs.

Joint

- Firm joint calipers (Fig 1a)
- Spring joint calipers (Fig 1b)

Fig 1



Legs

- Inside caliper for internal measurement.(Fig 2)
- Outside caliper for external measurement. (Fig 3)

Calipers are used along with steel rules, and the accuracy is limited to 0.5 mm. Parallelism of jobs etc. can be checked with higher accuracy by using calipers with sensitive feel.

Fig 2

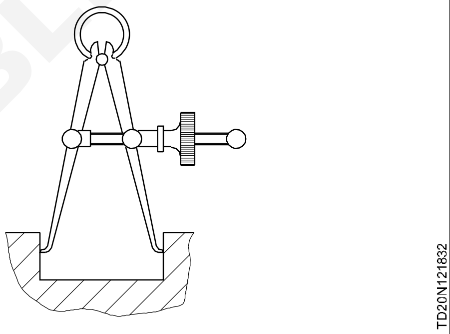
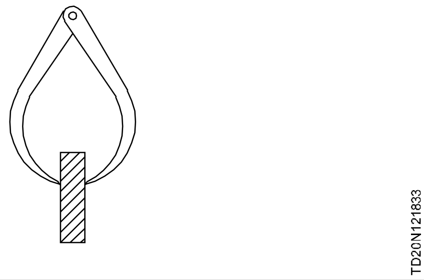


Fig 3



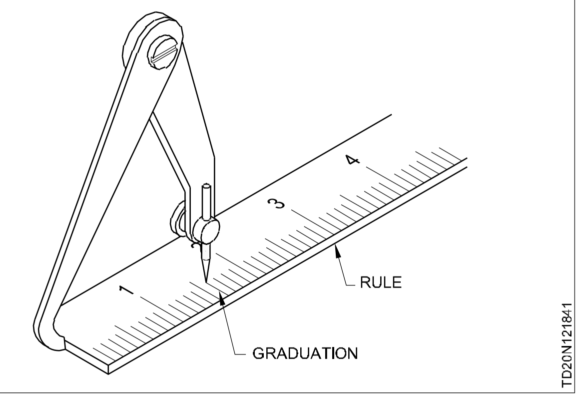
Spring joint calipers have the advantage of quick setting with the help of an adjusting nut. For setting a firm joint caliper, tap the leg lightly on a wooden surface.

Jenny calipers

Objectives: At the end of this lesson you shall be able to
- **state the uses of a jenny caliper**
- **state the two types of legs of a jenny caliper.**

Jenny calipers have one leg with an adjustable divider point, while the other is a bent leg. (Fig 1) These are available in sizes of 150 mm, 200 mm, 250 mm and 300mm.

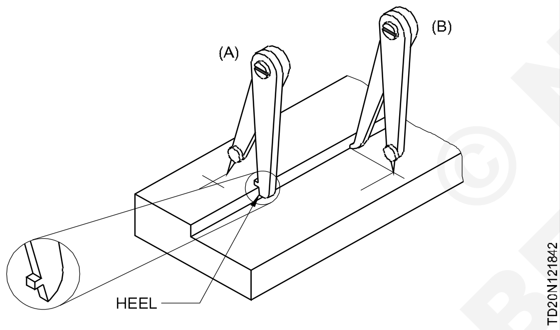
Fig 1



Jenny calipers are used

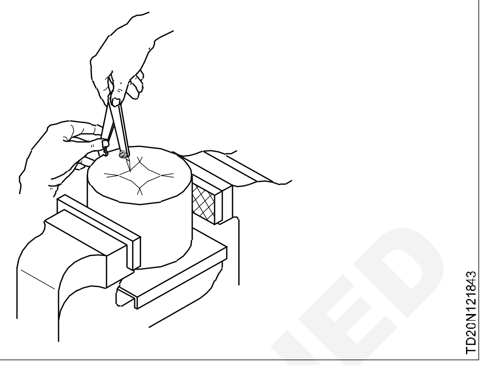
- for marking lines parallel to the inside and outside edges (Fig 2)

Fig 2



- for finding the centre of round bars (Fig 3)

Fig 3



These calipers are available with the usual bent leg or with heel.

Calipers with bent leg (Fig 2B) are used for drawing lines parallel along an inside edge. and the heel type (Fig 2A) is used for drawing parallel lines along the outer edges.

The other names for this caliper are:

- heremaphrodite calipers
- leg and point calipers
- odd leg caliper

Types of marking punches

Objectives: At the end of this lesson you shall be able to
- **explain the different punches in marking**
- **state the features of each punch and its uses.**

Punches are used in order to make certain dimensional features of the layout permanent. There are two types of punches. They are centre punch and prick punch made of high carbon steel, hardened and ground.

Centre Punch: The angle of the point is 90° in a centre punch. This punch mark made by this is wide and not very deep. This punch is used for locating centre of the holes. The wide punch mark gives a good seating for starting the drill. (Fig 1a)

Prick Punch / Dot punch: The angle of the prick punch is 30° or 60°. (Fig 1b) The 30° point punch is used for making light punch marks needed to position dividers. The divider point will get a proper seating in the punch mark. The 60° punch is used for marking witness marks and called as dot punch. (Fig 2)

Fig 1

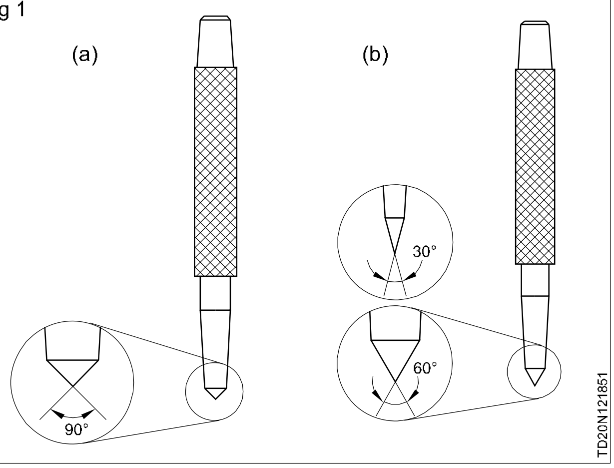
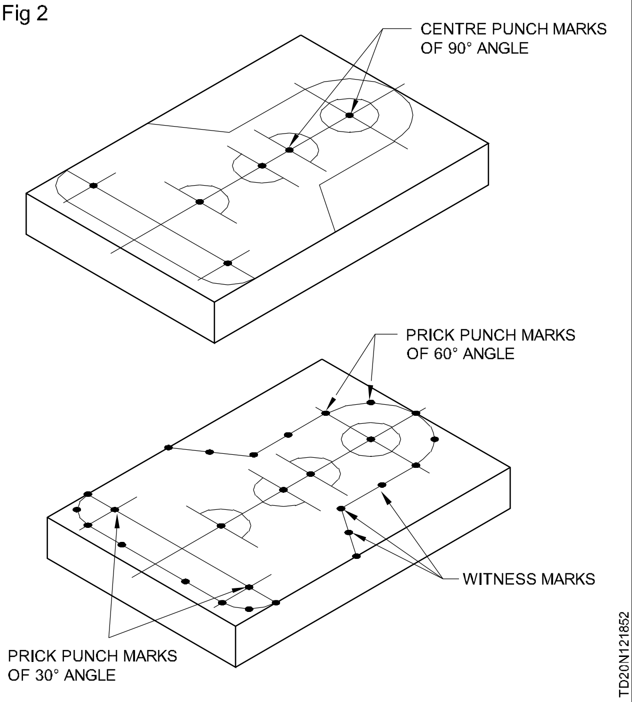


Fig 2



The witness marks should not be too close to one another.

Marking off and marking off table

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

Marking off

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

Witness marks

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

Marking off table (Figs 1 and 2)

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

Fig 1

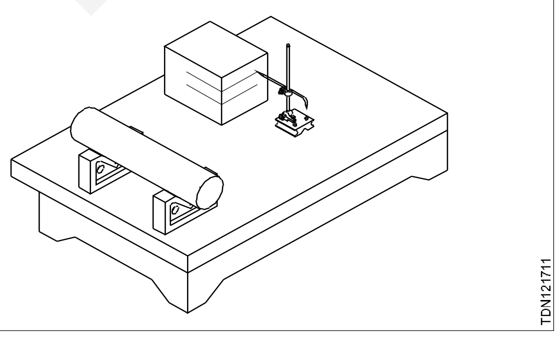
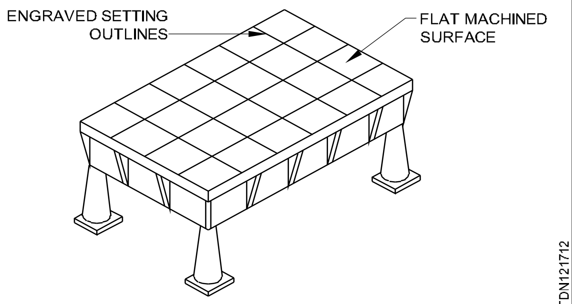


Fig 2



Marking tables are of 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.

Care and maintenance

A marking table is very precise 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.

Visual inspection of raw materials

Objective: At the end of this lesson you shall be able to
- **explain rusting, scaling, corrosion**

Rusting

Rusting is a chemical process which is common with metals contaning iron.

In the presence of oxygen and moisture or water iron under goes this chemical process and form a series of iron oxide

The reddish - brown color compound is known as rust.So rust contains hydrated iron (III) oxide $Fe2O_3$

Ways to prevent rust

1 Using rust resistant alloys

2 Galvanization

3 Bluing

4 Organic coating

5 Regular maintenance

Scalling

Scalling occurs when minerals like calcium, carbonate which can buidup on surfaces.

Ways to prevent scalling

1 Use of protective/antiscale coating.

Corrosion

Corrosion is a process where the metal corrodes. Corrosion is a natural process and in the presence of a moist atmosphere, chemically active metals get corroded. Contrary to the popular belief, rusting and corrosion are not the same.

Rusting is the process where iron corrodes due to exposure to the atmosphere. Corrosion is a process where the water or the moisture on the surfaceof the metal oxidizes with the atmospheric oxygen,it is an oxidation reaction. The main circumstance of corrosion occurs with iron because it is a structural material in construction, bridges, buildings, rail transport, ship, etc. Aluminium is also an important structural metal, but even aluminium goes under oxidation reactions. However, aluminium doesn't corrode or oxidize as rapidly as its reactivity suggests. An alloys of aluminium or any other metal like magnesium can make aluminium stronger,stiffer and harder.

The alkali metals like sodium need to be stored in oil as they corrode quickly. Less reactive metals like lead and copper are used to roof situations. Copper (Cu) corrodes and forms a basic green carbonate and lead corrodes to form a white lead oxide or carbonate.

Corrosion types and prevention method of corrosion prevent

- Alloy's
- Galvanizing
- Electroplating

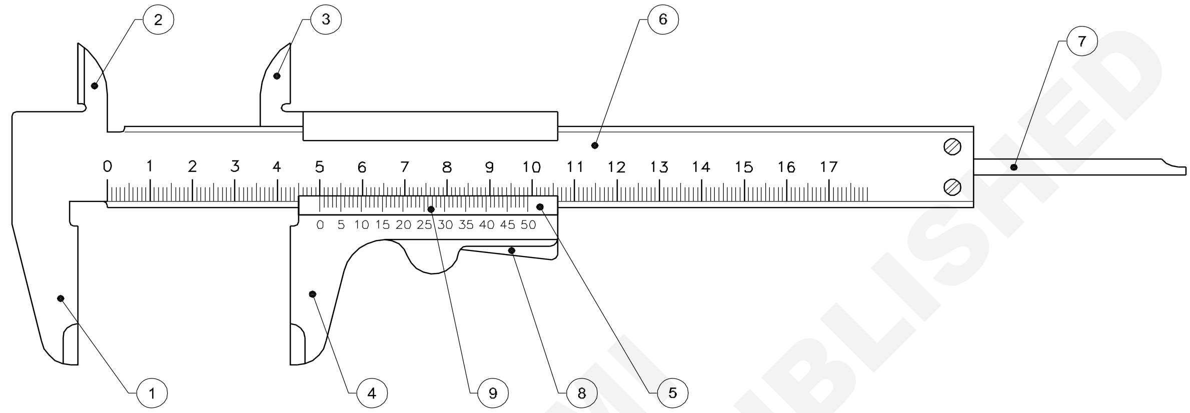
Vernier calipers

Objectives: At the end of this lesson you shall be able to
- **explain the parts of a vernier caliper**
- **state the constructional features of a vernier caliper**
- **state the uses of a vernier caliper.**

A vernier caliper is a precision measuring instrument. It is used to measure up to an accuracy of 0.02 mm. (Fig 1)

Parts of a vernier caliper (Numbers as per Fig 1)

Fig 1

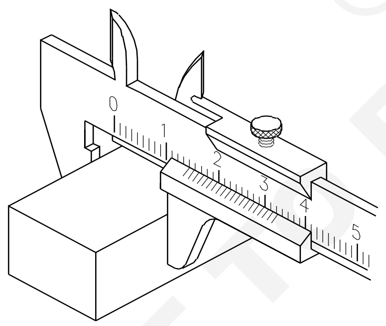


TD20N122011

Fixed jaws (1 and 2)

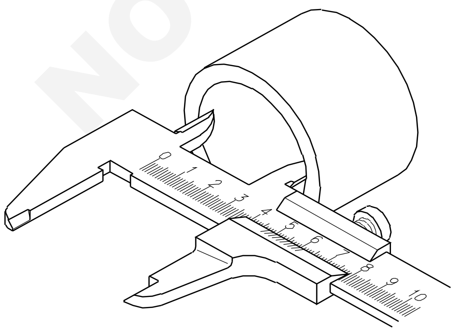
Fixed jaws are part of the beam scale. One jaw is used for taking external measurements, and the other for taking internal measurements.

Fig 2



TD20N122012

Fig 3



TD20N122013

Movable jaws (3 and 4)

Movable jaws are part of the vernier slide. One jaw is used for external measurements, and the other for internal measurements. (Figs 2 and 3)

Vernier slide (5)

A vernier slide moves over the beam and can be set in any position by means of a spring-loaded thumb lever.

Beam (6)

The vernier slide and the depth bar attached to it slide over the beam. The graduations on the beam are called the main scale divisions.

Depth bar (7)

The depth bar is attached to the vernier slide and is used for measurement of depth.

Thumb lever (8)

The thumb lever is spring-loaded which helps to set the vernier slide in any position on the beam scale.

Vernier scale (9)

The vernier scale is graduated and marked on the vernier slide. The divisions of this scale are called vernier divisions.

Main scale

The main scale graduations or divisions are marked on the beam.

Sizes

Vernier calipers are available in sizes of 150, 200, 250, 300 and 600 mm. The selection of the size depends on the measurements to be taken. Vernier calipers are precision instruments, and therefore, extreme care should be taken while handling them.

Never use a vernier caliper for any purpose other than measuring. Vernier calipers should be used only to measure machined or filed surfaces.

They should never be mixed with any other tools.
Clean the instrument after use, and store it in a box.

Graduations and reading of vernier calipers

Objectives: At the end of this lesson you shall be able to
- **explain the least count of a vernier caliper**
- **state how graduations are made on a vernier caliper with 0.02 mm least count**
- **expalin how to read vernier caliper measurements.**

Vernier calipers

Vernier calipers are available with different accuracies. The selection of the vernier caliper depends on the accuracy needed and the size of the job to be measured.

This accuracy/least count is determined by the graduations of the main scale and the vernier scale divisions.

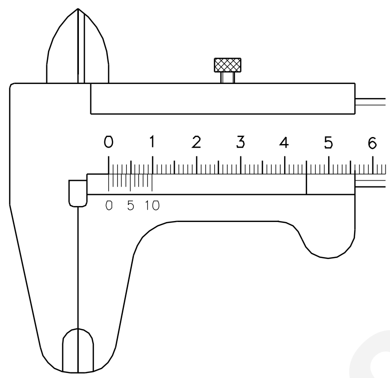
Vernier Principle

The vernier principle states that two different scales are constructed on a single known length of line and the difference between them is taken for fine measurements.

Determining the least count of vernier calipers

In the vernier caliper shown in Fig 1 the main scale divisions (9 mm) are divided into 10 equal parts in the vernier scale.

Fig 1



i.e. One main scale division (MSD) = 1mm

One vernier scale division (VSD) = 9/10 mm

Least count = 1 MSD – 1 VSD

= 1 mm – 9/10 mm = 0.1 mm.

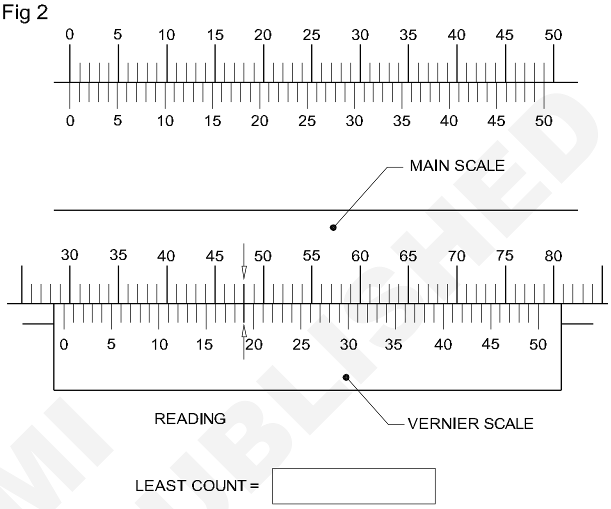
The difference between one MSD and one VSD = 0.1 mm.

Reading vernier measurements

Vernier calipers are available with different graduations and least counts. For reading measurements with a vernier caliper, the least count should be determined first. (The least count of calipers is sometimes marked on the vernier slide.)

Fig 2 shows the graduations of a common type of vernier caliper with a least count of 0.02 mm. In this, 50 divisions of the vernier scale occupy 49 divisions (49 mm) on the main scale.

Fig 2



Example

Calculate the least count of the vernier given in Figure 3.

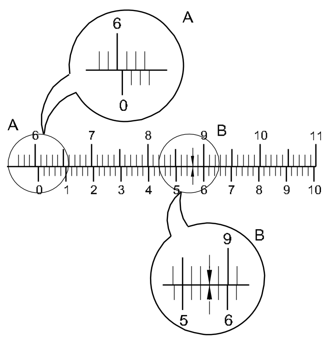
Least count = 1 mm – 49/50 mm

= 1/50 mm

= 0.02 mm.

Example for reading vernier caliper (Fig 3)

Fig 3



Main scale reading = 60 mm

The vernier division coinciding with the main scale is the 28th division, value = 28 x 0.02

= 0.56 mm

Reading = 60 + 0.56 = 60.56 mm.

Outside micrometer

Objectives: At the end of this lesson you shall be able to
- **explain the parts of an outside micrometer**
- **state the functions of the main parts of an outside micrometer.**

A micrometer is a precision instrument used to measure a job, generally within an accuracy of 0.01 mm.

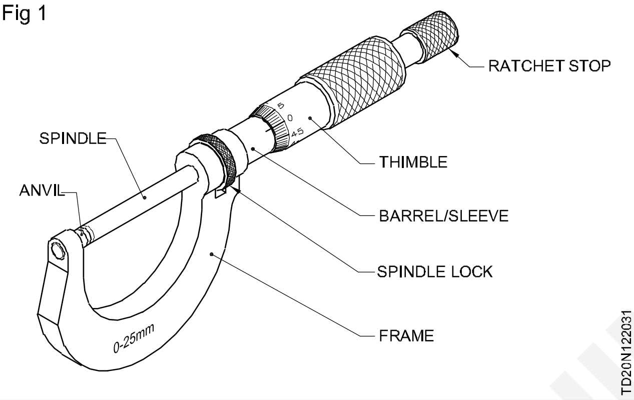
Micrometers used to take the outside measurements are known as outside micrometers. (Fig 1)

The parts of a micrometer are listed here.

Frame

The frame is made of drop-forged steel or malleable cast iron. All other parts of the micrometer are attached to this.

Fig 1



Barrel/sleeve

The barrel or sleeve is fixed to the frame. The datum line and graduations are marked on this.

Thimble

On the bevelled surface of the thimble also, the graduation is marked. The spindle is attached to this.

Spindle

One end of the spindle is the measuring face. The other end is threaded and passes through a nut. The threaded mechanism allows for the forward and backward movement of the spindle.

Anvil

The anvil is one of the measuring faces which is fitted on the micrometer frame. It is made of alloy steel and finished to a perfectly flat surface.

Spindle lock-nut: The spindle lock-nut is used to lock the spindle at a desired position.

Ratchet stop

The ratchet stop ensures a uniform pressure between the measuring surfaces.

Graduations of metric outside micrometer

Objectives: At the end of this lesson you shall be able to
- **state the principle of a micrometer**
- **explain the least count of an outside micrometer**

Working principle

The micrometer works on the principle of screw and nut. The longitudinal movement of the spindle during one rotation is equal to the pitch of the screw. The movement of the spindle to the distance of the pitch or its fractions can be accurately measured on the barrel and thimble.

Graduations (Fig 1)

In metric micrometers the pitch of the spindle thread is 0.5 mm.

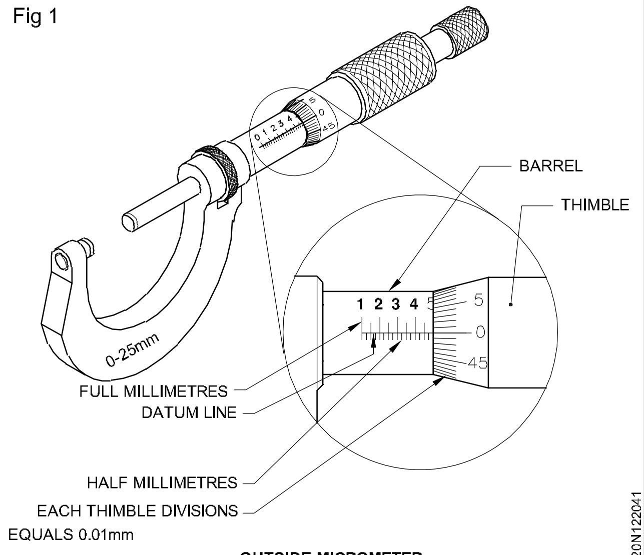
Thereby, in one rotation of the thimble, the spindle advances by 0.5 mm.

On the barrel a 25 mm long datum line is marked. This line is further graduated to millimetres and half millimeters (ie., 1 mm & 0.5 mm). The graduations are numbered as 0,5,10,15, 20 & 25 mm.

The circumference of the bevel edge of the thimble is graduated into 50 divisions of the thimble = 0.5 x 1/50 = 0.01 mm.

> **Accuracy or the least count of a metric outside micrometer is 0.01 mm.**

Fig 1



OUTSIDE MICROMETER

Reading dimensions with outside micrometers

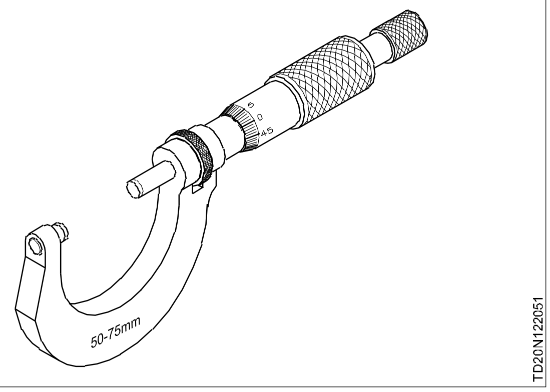
Objectives: At the end of this lesson you shall be able to
- **state the required range of a micrometer**
- **explain how to micrometer measurements.**

Ranges of outside micrometer

Outside micrometers are available in ranges of 0 to 25 mm, 25 to 50 mm, 50 to 75 mm, 75 to 100 mm, 100 to 125 mm and 125 to 150 mm.

For all ranges of micrometers, the graduation marked on the barrel is only 0-25 mm. (Fig 1)

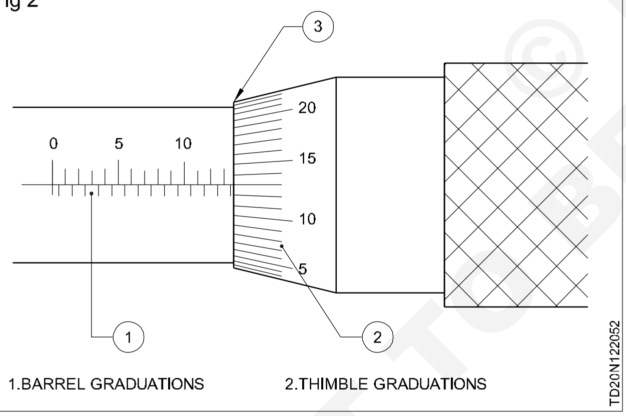
Fig 1



Reading micrometer measurements

How to read a measurement with an outside micrometer? (Fig 2)

Fig 2



First note the minimum range of the outside micrometer. While measuring with a 50 to 75 mm micrometer, note it as 50 mm.

Then read the barrel graduations. Read the value of the visible lines on the left of the thimble edge.

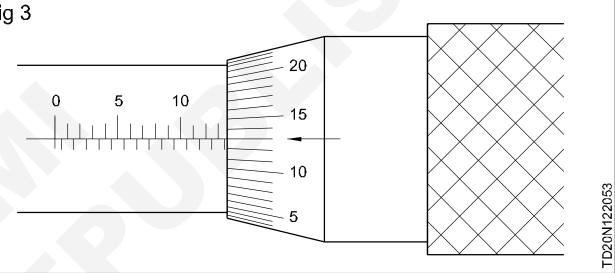
	13.00 mm
+	00.50 mm
	13.50 mm

Barrel graduation

Next read the thimble graduations.

Read the thimble graduations in line with the barrel datum line, 13th div. (Fig 3)

Fig 3



Multiply this value with 0.01 mm (least count). 13 x 0.01 mm = 0.13 mm)

Add

Minimum range	50.00 mm
Barrel reading	13.50 mm
Thimble reading	00.13 mm
Total	63.63 mm

The micrometer reading is 63.63 mm.

Vernier micrometer graduation and reading

Objectives: At the end of this lesson you shall be able to
- **state the graduations of a vernier micrometer (metric)**
- **state vernier micrometer.**

Vernier micrometer

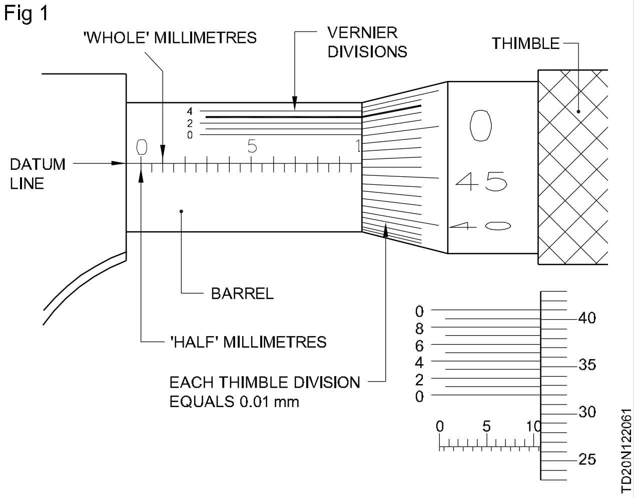
Ordinary metric micrometers can measure only to an accuracy of ±.01mm.

For taking more accurate measurements, vernier micrometers are useful. Vernier micrometers can measure to an accuracy of ±.001 mm.

Construction and graduation

Vernier micrometers are very similar to the ordinary micrometers in construction. The difference is in the graduation. These micrometers have additional, equally spaced graduations (vernier graduations) given above the datum line. There are ten such vernier graduation lines marked parallel above the datum line. (Fig 1) The space between these 10 lines is equal to 9 divisions in the thimble. (Fig 1)

Fig 1



The value of 10 vernier divisions is

0.0 1 mm X 9 = 0.09 mm

The value of a vernier division

$$= \frac{0.09mm}{10} = 0.009 \text{ mm.}$$

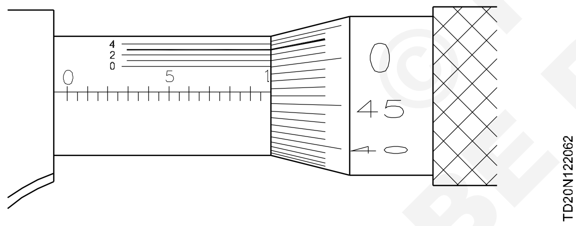
The least count =

1 thimble division – 1 Vernier division =

0.01 – 0.009mm = 0.001 mm or 1/1000 mm.

Reading a vernier micrometer (Fig 2)

Fig 2



Example

After measuring, read the full mm divisions visible on the barrel.

9 full divisions	9 mm

Note the half divisions, if any, visible on the barrel.

1 half division	0.5 mm

Read the thimble divisions below the datum line. (Fig 2)

46 divisions	0.46 mm

Note the vernier division coinciding with the thimble division.

3 divisions	0.003 mm

Add up all the readings together.

Calculation

The range of micrometer is 0 to 25 mm.

1	Full mm division visible before the thimble edge	9	=	9.000 mm
2	Half mm division visible after the full mm divsion on thimble	1	=	0.500 mm
3	Thimble division below the index line	46	=	0.460 mm
4	Vernier division coinciding with thimble division	3	=	0.003 mm
	Reading			9.963 mm

Vernier height gauge

Objectives: At the end of this lesson you shall be able to
- **name the parts of a vernier height gauge**
- **state the functions of each part**
- **list out the specific uses of a vernier height gauge.**

Specific uses of a vernier height gauge

Accurate measurements are important in layout (marking off) and inspection work. (Figs 1 & 2)

Vernier height gauges are particularly suitable for marking off accurate distances, and centre locations.

The graduations and readings are the same as those of a vernier caliper.

Fig 1

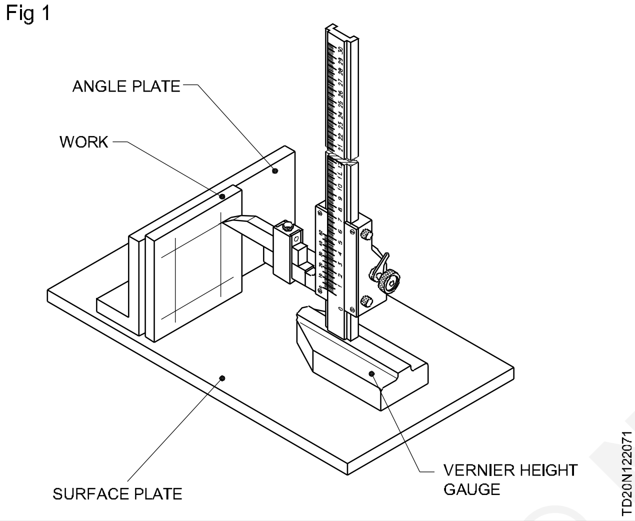
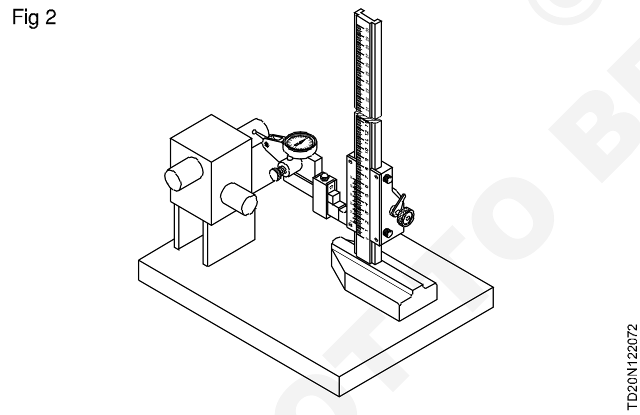


Fig 2



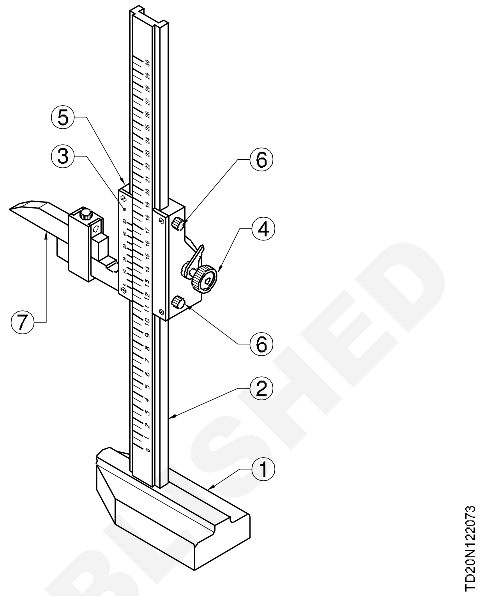
Parts of a vernier height gauge and their functions

The main parts of a vernier height gauge and their functions are given here. (Fig 3)

Base (1)

This is the datum from which measurements and settings are made. The underside of the base is hardened, ground and lapped.

Fig 3



Beam (2)

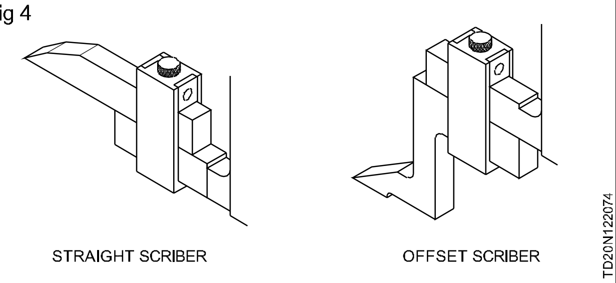
This is similar to the beam scale of a vernier caliper and is attached to the base.

Vernier slide (3)

This unit slides on the beam and carries the vernier plate (5), locking screws (6), fine setting device (4) and scriber (7). Some vernier height gauges are provided with a rack and pinion arrangement for moving the slide along the beam.

Vernier height gauges are provided with both straight and offset scribers.(Fig 4)

Fig 4



Zero setting of the vernier height gauge.

The offset scriber permits zero setting of the instrument from the datum surface.

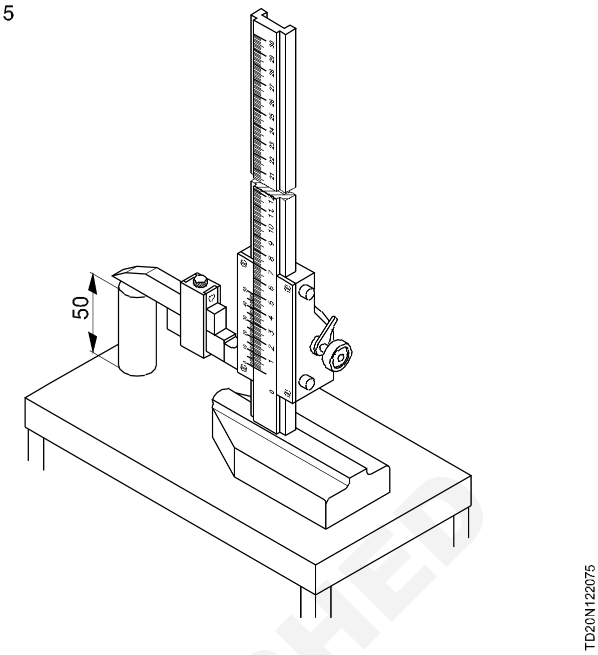
While using a straight scriber, the zero setting of the instrument is at a level above the datum surface. In this case the zero setting is to be checked using the precision round block, supplied along with the instrument.

Vernier height gauges with which we can measure from the datum surface without the special offset scribers are also available.(Fig 5)

The size of the vernier height gauge is stated by the height of the beam. The most commonly used size has a beam of 300 mm height.

Vernier height gauges are used with surface plates or other accurate flat surfaces.

Fig 5



Scribers

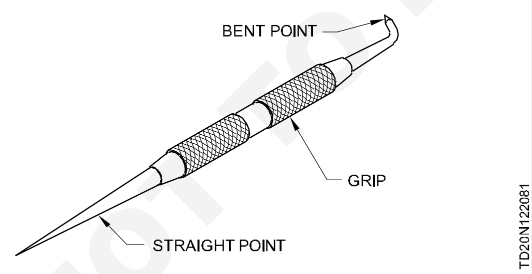
Objectives: At the end of this lesson you shall be able to
- **state the features of scribers**
- **state the uses of scribers.**

Scribers

In lay out work it is necessary to scribe lines to indicate the dimensions of the workpiece to be filed or machined. The scriber is a tool used for this purpose. It is made of high carbon steel and is hardened. For drawing clear and sharp lines, the point should be ground and honed frequently for maintaining its sharpness.

Scribers are available in different shapes and sizes. The most commonly used one is the plain scriber. (Fig 1)

Fig 1

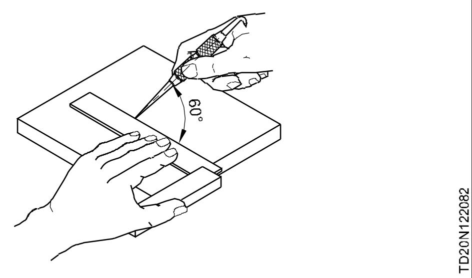


While scribing lines, the scriber is used like a pencil so that the lines drawn are close to the straight edge. (Fig 2)

Scriber points are very sharp; therefore, do not put the plain scriber in your pocket.

> **Place a cork on the point when not in use to prevent accidents.**

Fig 2



Marking out

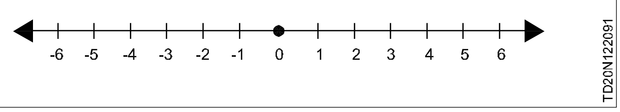
Objectives: At the end of this lesson you shall be able to
- **state what is co-ordinate system**
- **list and brief the different co-ordinate system used in marking**
- **brief the datum line in marking..**

Co-ordinate system

A coordinate system is a system consisting of a set of points, lines or surfaces with each point having a unique location or coordinate that is assigned to it.

A number line is a line on which positive and negative whole numbers are represented. It typically start at 'O' and goes up to positive infinity or negative infinity depending on the direction. (Fig 1)

Fig 1



Types of coordinate system

There are many types of coordinate system, but some are commonly used.

They are

1 Cartesian coordinate system

2 rectangle coordinate system

3 polar coordinate system

Cartesian coordinate system

The Cartesian coordinate system is the most commonly used coordinate system. It uses a coordinate plane with two perpendicular axes, X and Y, to locate points. The x-axis runs vertically.(Fig 2) Points are located by their X and Y coordinates which are measured in units of distance from the origin (Fig 3).

Fig 2

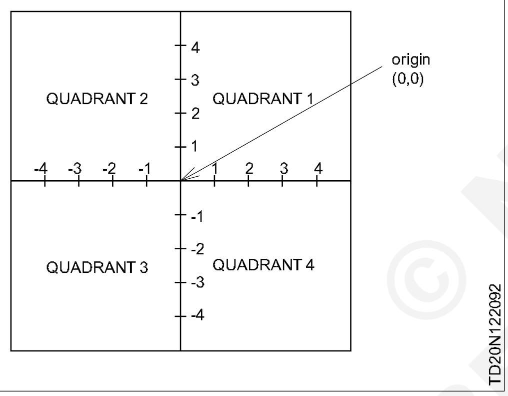
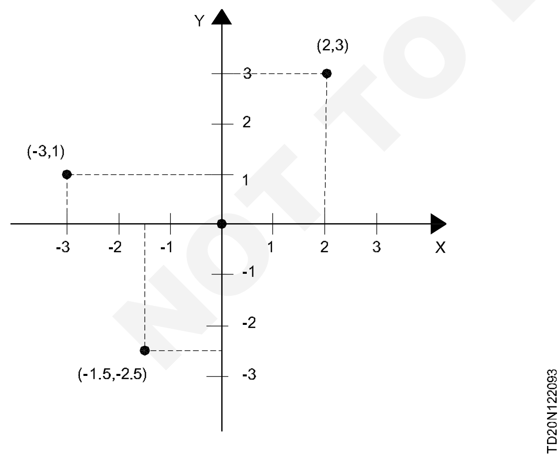


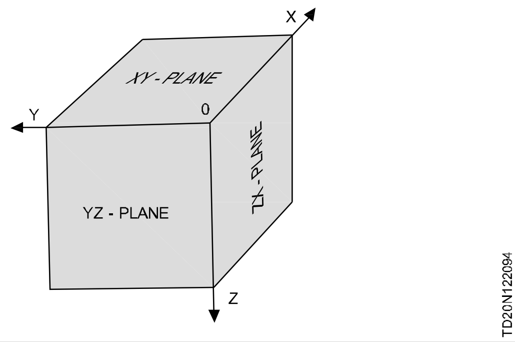
Fig 3



Rectangular coordinate system (Fig 4)

A rectangular coordinate system is a three-dimensional coordinate system that uses Cartesian coordinates to specify the position of points in space. It uses three perpendicular lines, called axes, to uniquely identify every point.

Fig 4

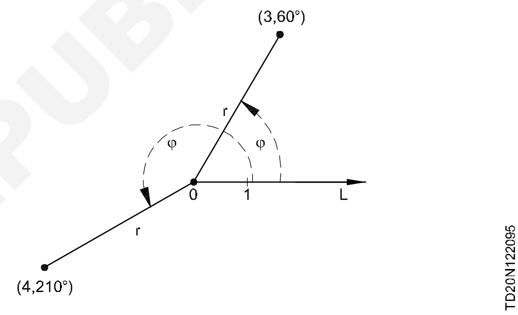


The x-axis is the horizontal line that goes through the point, the y-axis is the vertical line that goes through the point, and the z-axis is the line that goes perpendicular to the other two axes, and runs through the point. The point's position is specified by its distance from the origin along each axis. The origin is the point when the axes intersect.

Polar coordinate system

The polar coordinate system is a coordinate system that uses two angular coordinate. The point (r,) lies on the unit circle, which has its center at the origin and a radius of 1. The polar coordinate system is shown in figure 5.

Fig 5



Method of Marking

The following methods are often used for marking

1 Datum Line Method

2 Center line method

3 Marking by Template

4 Marking of Center on Round Rod End

1 Datum Line Method

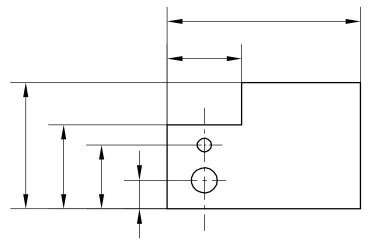
In this method, first, a baseline is applied which is called a datum line and all further lines are preceded by the preceding line.

This method is used where the jobs are finished at right angles (Fig 6)

2 Center line method

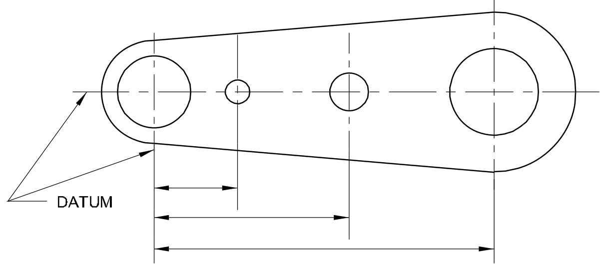
This method of marking is used on odd-shaped jobs firstly, a center line is drawn and other joint line of marking are attached with reference to this center line (Fig 7)

Fig 6



EDGE DATUM

Fig 7



CENTRE LINE DATUM

Point to be noted while marking

- Carefully study the drawing of the workpiece
- Determine the datum surfaces from which all the marks are to be marked
- Calculate all the dimensions and allowances of the workpiece
- Check the correct use of marking media on the surface on the surface of the workpiece
- First mark straight horizontal, vertical and slant line and then mark lines of circles
- After marking clean the tools used for marking

Bevel protractor

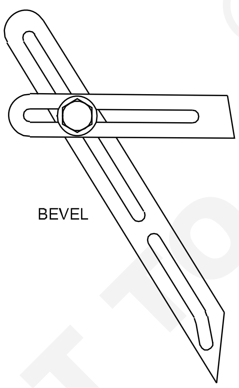
Objectives: At the end of this lesson you shall be able to

- **state the names of sem-precision angular measuring instruments**
- **differentiate between bevel and universal bevel gauges**
- **state the features of bevel protractors.**

The most common instruments used to check angles are the:

- bevel or bevel gauge (Fig 1)
- universal bevel gauge (Fig 2)
- bevel protractor (Fig 3)

Fig 1



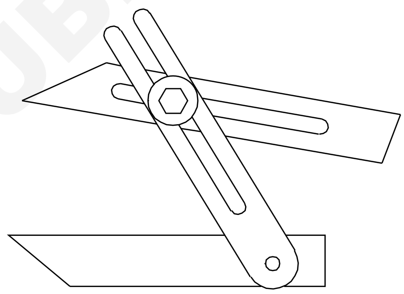
Bevel gauges

The bevel gauges cannot measure angles directly. They are, therefore, indirect angular measuring instruments. The angles can be set and measured with bevel protractors.

Universal bevel gauges

The universal bevel gauge has an additional blade. This helps in measuring angles which cannot be checked with an ordinary bevel gauge. (Fig 2)

Fig 2

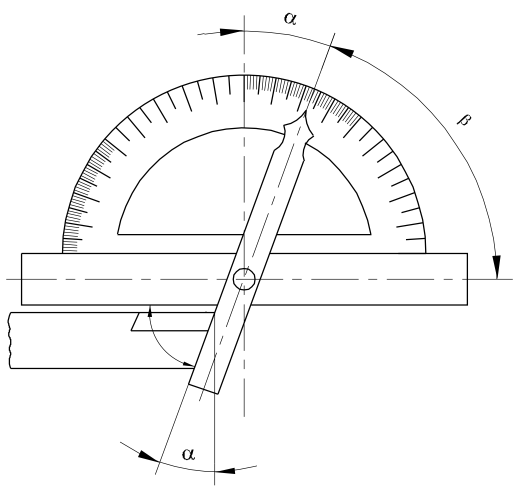


UNIVERSAL BEVEL

Bevel protractor

The bevel protractor is a direct angular measuring instrument, and has graduations marked from 0° to 180°. This instrument can measure angles within an accuracy of ±1°. (Fig 3)

Fig 3



BEVEL PROTRACTOR

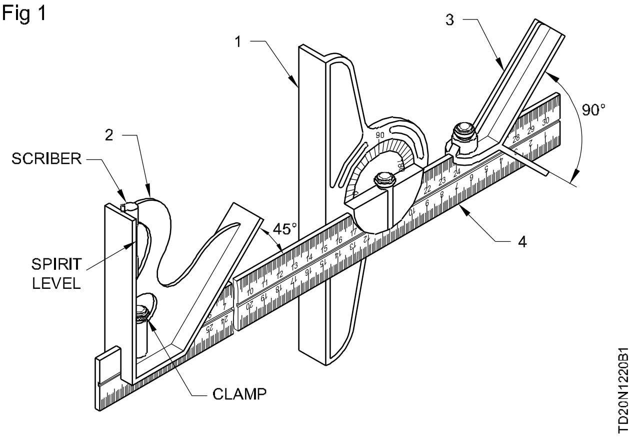
Combination set

Objectives: At the end of this lesson you shall be able to
- **identify the parts of a combination set**
- **state the uses of each attachment in a combination set.**

Combination sets can be used for different types of work, like layout work, measurement and checking of angles.

The combination set (Fig 1) has a

Fig 1



- protractor head (1)
- square head (2)
- centre head (3)
- rule (4)

Square head

The square head has one measuring face at 90° and another at 45° to the rule.

It is used to mark and check 90° and 45° angles. It can also be used to set workpieces on the machines and measure the depth of slots. (Figs 2,3 and 4)

Fig 2

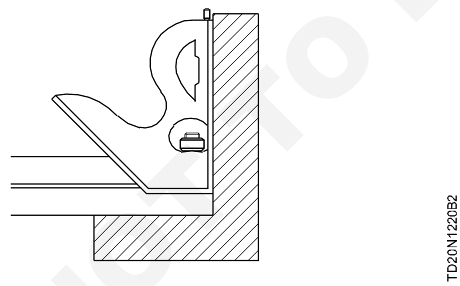


Fig 3

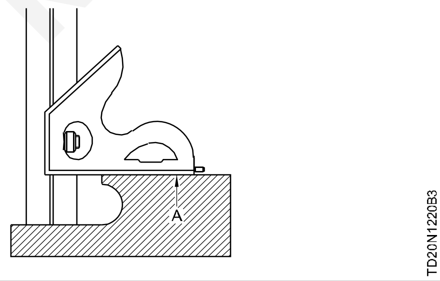
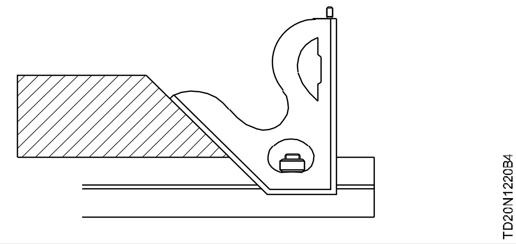


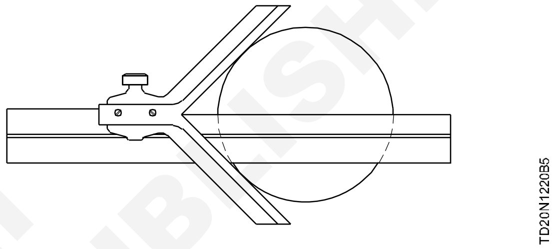
Fig 4



Centre head

This along with the rule is used for locating the centre of cylindrical jobs. (Fig 5)

Fig 5



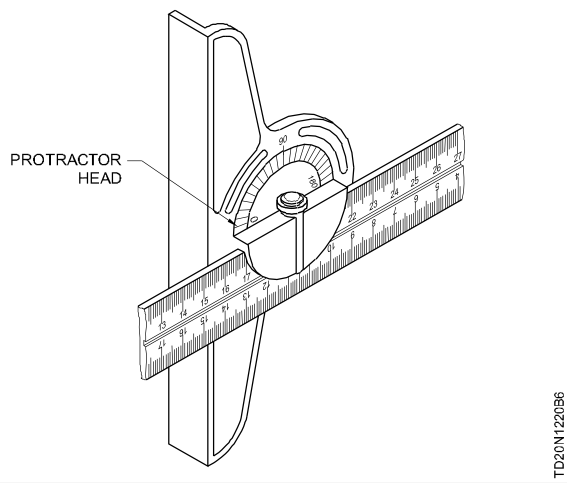
For ensuirng accurate results, the combination set should be cleaned well after use, and should not be mixed with cutting tools, either while using or storing.

Protractor head

The protractor head can be rotated and set to any required angle.

The protractor head is used for marking and measuring angles within an accuracy of ±1°. The spirit level attached to this, is useful for setting jobs in a horizontal plane. (Fig 6)

Fig 6



Pedestal grinders

Objectives: At the end of this lesson you shall be able to
- **state the purpose of off-hand grinding**
- **state the machines with which off-hand grinding is done**
- **state the features of bench and pedestal grinders.**

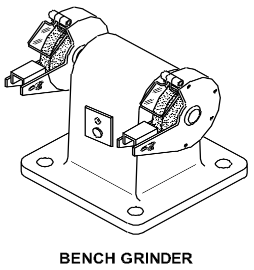
Off-hand grinding is the operation of removing material which does not require great accuracy in size or shape. This is carried out by pressing the workpiece by hand against a grinding wheel.

Off-hand grinding is performed for rough grinding of jobs and resharpening of

- scribers
- punches
- chisels
- twist drills
- single point cutting tools etc.

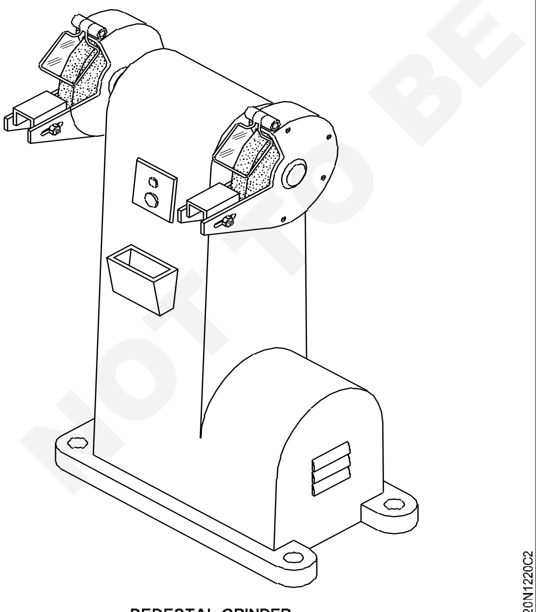
Off-hand grinding is performed with a bench or pedestal grinder. (Figs 1 and 2)

Fig 1



BENCH GRINDER

Fig 2



PEDESTAL GRINDER

Bench grinders

Bench grinders are fitted to a bench or table, and are useful for light duty work.

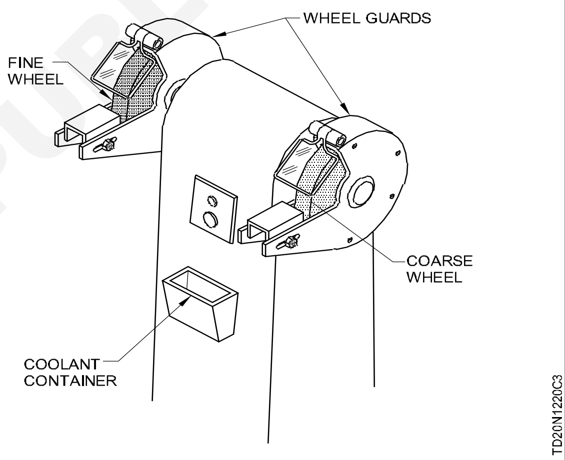
Pedestal grinders

Pedestal grinders are mounted on a base (pedestal), which is fastened to the floor. They are used for heavy duty work.

These grinders consist of an electric motor and two spindles for mounting grinding wheels. On one spindle a coarse-grained wheel is fitted, and on the other, a fine grained wheel. For safety, while working, wheel guards are provided. (Fig 3)

A coolant container is provided for frequent cooling of the work. (Fig 3)

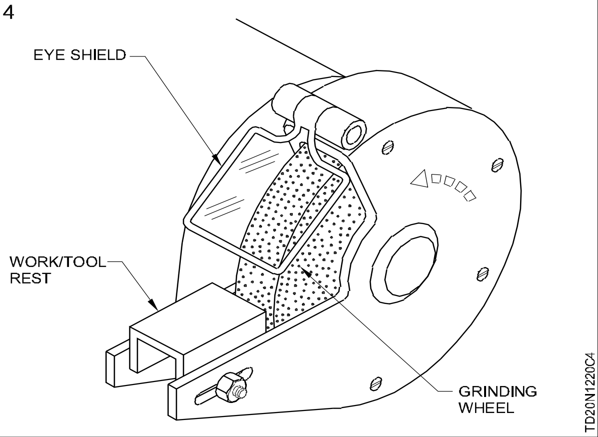
Fig 3



Adjustable work-rests are provided for both wheels to support the work while grinding. These work-rests must be set very close to the wheels. (Fig 4)

Extra eye shields are also provided for the protection of the eyes. (Fig 4)

Fig 4



Maintaining grinding wheels

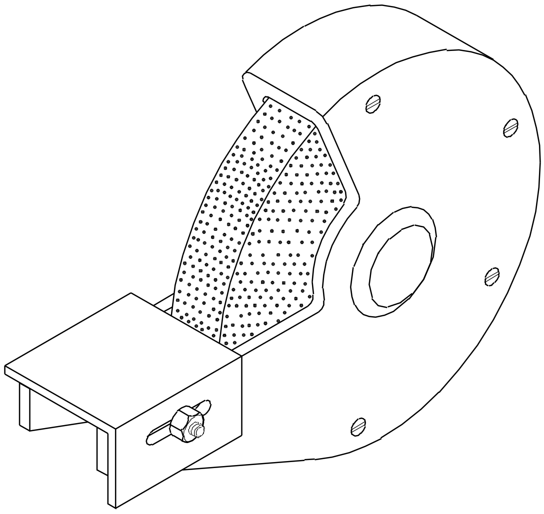
Objectives: At the end of this lesson you shall be able to
- **explain difference between loading and glazing**
- **state the effects of loading and glazing**
- **explain difference between dressing and truing.**

Grinding wheels become inefficient due to two main causes known as loading and glazing.

Loading

When soft materials such as aluminium, copper, lead etc. are ground, the metal particles get clogged in the pores of the wheel. This condition is called loading. (Fig 1)

Fig 1



TD20N1220D1

Glazing

When a surface of the wheel develops a smooth and shining appearance, it is said to be glazed. This indicates that the wheel is blunt, i.e. the abrasive grains are not sharp.

When such grinding wheels are used, there is a tendency to exert extra pressure in order to make the wheels cut. Excessive pressure on the grinding wheel will lead to the fracture of the wheel, excessive heating of the wheel, weakening of bonding of the wheel and bursting of the wheel.

Dressing

The purpose of dressing is to restore the correct cutting action of the wheel. Dressing removes the clogs on the surface of the wheel and the blunt grains of the abrasive, exposing the new sharp abrasive grains of the wheel which can be cut and brought to shape efficiently.

Truing

Truing refers to the shaping of the wheel to make it run concentric with the axis. When a new grinding wheel is mounted, it must be trued before use. The cutting surface of a new wheel may run out slightly due to the clearance between the bore and the machine spindle. Grinding wheels, which are in use, can also run out of true, due to uneven loading while grinding.

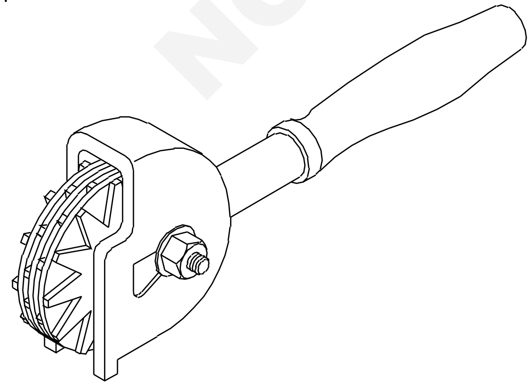
Dressing and truing are done at the same time.

Grinding wheel dressers

Objectives: At the end of this lesson you shall be able to
- **explain the common types of wheel dressers**
- **state the uses of each type of wheel dresser.**

The wheel dressers used for off-hand grinders are star wheel dressers (Fig 1) (Huntington type wheel dresser) and diamond dressers.

Fig 1



TD20N1220F1

The star wheel dresser consists of a number of hardened star-shaped wheels mounted on a spindle at one end and a handle at the other end.

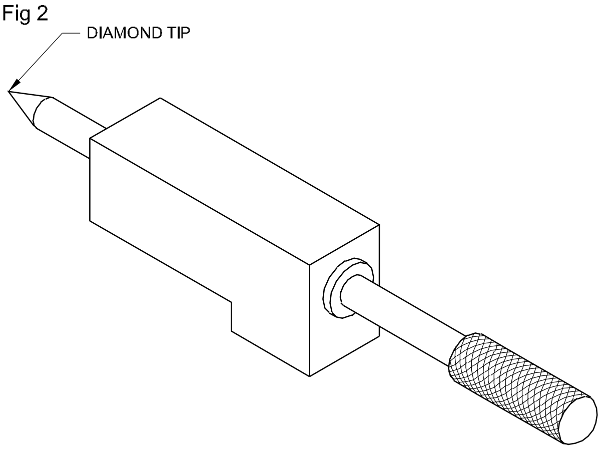
While dressing, the star wheel is pressed against the face of the revolving grinding wheel. The star wheel revolves and digs into the surface of the grinding wheel. This releases the wheel loading and dull grains, exposing sharp new abrasive grains.

Star wheels are useful for pedestal grinders in which a precision finish is not expected.

Star wheel dressers should be used only on wheels which are large enough to take the load.

Diamond dressers (Fig 2): Bench type off-hand grinders used for sharpening cutting tools are usually fitted with smaller and rather delicate wheels.

Fig 2



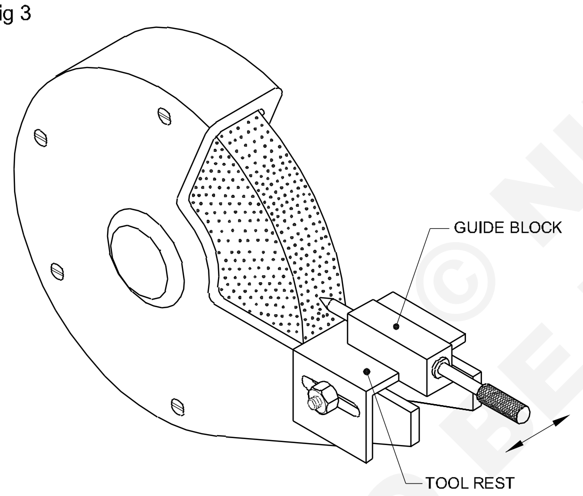
These wheels are dressed and trued with diamond dressers.

Diamond dressers consist of a small diamond mounted on a holder which can be held rigidly on the work-rest.

How to use a wheel dresser (Fig 3)

For dressing and truing, the dresser is slowly brought in contact with the wheel face and moved across.

Fig 3



The finish obtained depends on the rate at which the dresser is moved across the face.

For roughing, the dresser is moved faster.

For fine finish, the dresser is moved slowly.

Roughing will be efficient with a dresser that has a sharp point, while, for fine finishing, a blunt diamond dresser is more suitable.

Abrasive stick: When only a light dressing is required, abrasive sticks can also be used. There are abrasive materials made in the form of sticks for the convenience of handling.

> **Diamond dressers, if moved too slowly, can glaze the wheel.**

Safety precautions

Prior to using a bench or pedestal grinder, you must receive basic shop safety training and equipment specific training before using this tool. You must wear appropriate PPE and follow all shop rules. Refer to the manufacturer's operating manual for all operating procedures.

Before starting the bench or pedestal grinder

- Keep the tool rest as close to the grinding wheel as possible without touching it. The tool rest must be minimally within 3mm of the grinding wheel.
- If a magnetic chuck is being used on the surface grinder, make sure it is holding the work securely before starting to grind.
- Prior to starting the grinder, ensure the tang at the top of the wheel opening is located within 6mm of the wheel.
- Before starting the grinder, make absolutely sure that the grinding wheel clears the top of the work piece. Approach the work piece manually to ensure this. Do not feed the table in automatic grind mode.

While in operation

- Stand to one side of the wheel when turning on the power. Damaged wheels will sometimes fly apart, and this is most likely to happen upon startup.
- Be alert and cautious when a grinding operation requires locating fingers close to the wheel. Feed the stock into the wheel with light to medium pressure. Do not force the piece.
- Do not use the side of the grinding wheel to shape stock.
- Stand erect in front of the grinder with both legs straight and slightly apart. Avoid stooping or leaning in to the machine.
- Keep the grinding wheel dressed. Dressing a small amount frequently is better than having to dress a lot later and will allow the wheel to cut faster, cooler and with a better surface finish. Dressing is cleaning and smoothing the surface of the grinding wheel.
- Hold work securely while grinding, use the tool rest to support the work when off-hand grinding on bench or pedestal grinders.

Care and maintenance

- Dress the grinding wheels regularly
- Do frequent, light dressing rather than one hardly dressing
- Replace the worn wheels it you cannot dress it
- Ensure the grinding speed does not exceed the operating speed marked on the grinding wheels
- Visually inspect the grinding wheels for possible damaged before mounting
- Do not grind on the side of a regular wheel
- Do not use a wheel theet doesnot fit properly to the spindle.

Marking media

Objectives: At the end of this lesson you shall be able to
- **state the purpose of marking media**
- **explain the common types of marking media**
- **state the correct marking medium for different applications.**

Purpose of marking media

In marking off/Layout, the surface of the job/workpiece is coated with a medium to show the marked lines clear and visible. To get clear and thin lines, the best layout medium is to be selected.

Different marking media

The different marking media are Whitewash, Marking blue, Prussian Blue, Copper Sulphate and Cellulose Lacquer.

Whitewash

Whitewash is prepared in many ways.

Chalk powder mixed with water

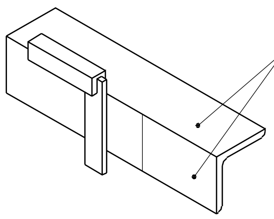
Chalk mixed with methylated spirit

White lead powder mixed with turpentine

Whitewash is applied to rough forgings andcastings with oxidised surface. (Fig 1)

Whitewash is not recommended for workpieces of high accuracy.

Fig 1



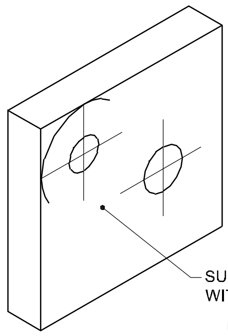
Marking blue

A Chemical dye, blue based colour mixed with methylated spirit used for marking on workpieces which are reasonably machined surface.

Prussian blue

This is used on filed or machine-finished surfaces. This will give very clear lines but takes more time for drying than the other marking media. (Fig 2)

Fig 2



Copper sulphated

The solution is prepared by mixing copper sulphate with water and a few drops of nitric acid. The copper sulphate is used on filed or machine-finished surfaces. Copper sulphate sticks to the finished surfaces well.

Copper sulphate needs to be handled carfully as it is poisonous. Copper sulphate coating should be dried before commencing marking, as otherwise, the solution may stick on the instruments used for marking.

Cellulose lacquer: This is a commercially available marking medium. It is made in different colours and dries very quickly.

The selection of marking medium for a particular job depends on the surface finish and the accuracy of the workpiece.

In present days, marking media used are readily available in aerosol container, which can be applied by spraying on to any surface, which needs marking.

Readymade solutions of marking dye/ink which are quick drying and thin layer to mark precise dimensions and clear visible lines. Also permanent marker pens are available in different. colours, which are quick drying and used for smaller workpieces of metal, wood and plastics.

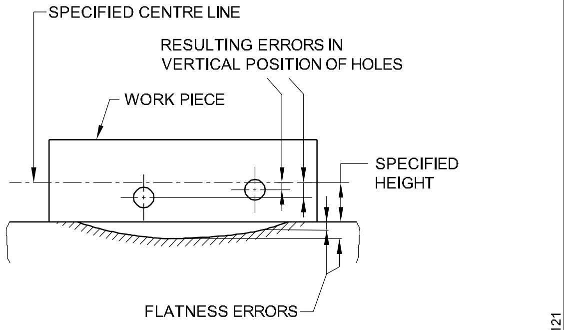
Surface plates

Objectives: At the end of this lesson you shall be able to
- **state the necessity of surface plate**
- **state the material of surface plate**
- **state the specification of surface plate.**

Surface plates - their necessity

When accurate dimensional features are to be marked, it is essential to have a datum plane with a perfectly flat surface. Marking using datum surfaces which are not perfectly flat will result in dimensional inaccuracies. (Fig.1) The most widely used datum surfaces in machine shop work are the surface plates and marking tables.

Fig 1



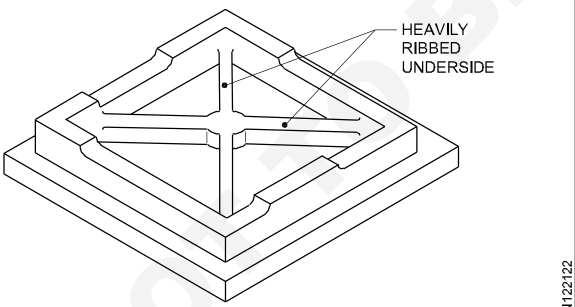
EFFECT OF MARKING-OUT FROM NON-FLAT SURFACE. NOTE THAT THE ERRORS IN THE HOLE POSITIONS ARE EQUAL TO THE FLATNESS ERRORS

Materials and construction

Surface plates are generally made of good quality cast iron which are stress-relieved to prevent distortion.

The work-surface is machined and scraped. The underside is heavily ribbed to provide rigidity. (Fig 2)

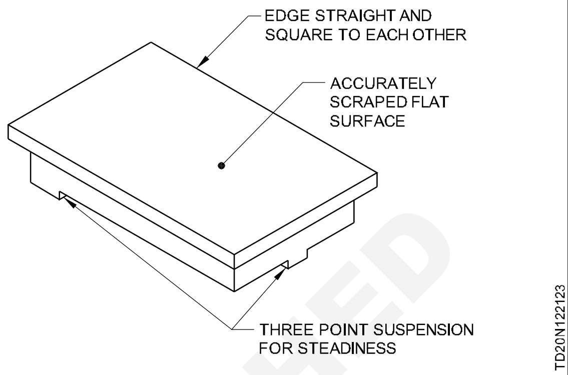
Fig 2



CAST IRON SURFACE PLATE

For the purpose of steadiness and convenience in levelling, a three point suspension is given. (Fig 3)

Fig 3



Smaller surface plates are placed on benches while the larger surface plates are placed on stands.

Other materials used

Granite is also used for manufacturing surface plates. Granite is a dense and stable material. Surface plates made of granite retain their accuracy, even if the surface is scratched. Burrs are not formed on these surfaces.

Classification and uses

Surface plates used for machine shop work are available in three grades - Grades 1, 2 and 3. The grade 1 surface plate is more acceptable than the other two grades.

Specifications

Cast iron surface plates are designated by their length, breadth, grade and the Indian Standard number.

Example

Cast iron surface plate 2000 x 1000 Gr1. I.S. 2285.

Care & maintenance

- Clean before and after use.
- Do not keep job on the surface plate.
- Don't keep any cutting tool on the table.

'V' Blocks

Objectives: At the end of this lesson you shall be able to
- **state the constructional features of 'V' Blocks**
- **explain the types of 'V' Blocks and state their uses**
- **state 'V' Blocks as per B.I.S recommended practice.**

Constructional features

'V' Blocks are devices used for marking and setting up work on machines. The features of the common type of 'V'Blocks are as given in Figs 1 and 2.

The included angle of the VEE is 90° in all cases. 'V' Blocks are finished to a high accuracy in respect of dimension, flatness and squareness.

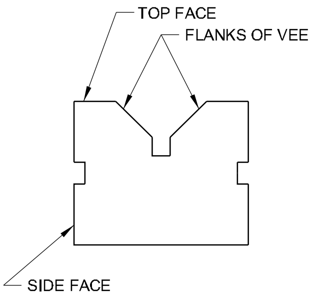
Types

Different types of 'V' blocks are available. As per BIS, there are four types, as listed below.

Single level single groove 'V' Block (Fig 1)

This type has only one 'V' groove, and has single groove (slots) on either side. These grooves are for accommodating the holding clamps.

Fig 1



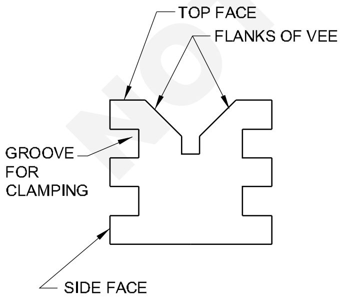
SINGLE LEVEL SINGLE GROOVE V-BLOCK

TD20N122211

Single level double groove 'V' Block (Fig 2)

This type will have one 'V' groove, and two grooves (slots) on either side for clamping in two positions.

Fig 2



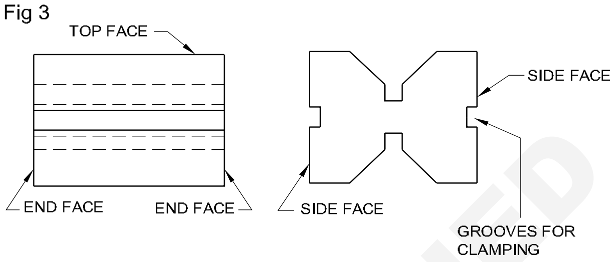
SINGLE LEVEL DOUBLE GROOVE V-BLOCK

TD20N122212

Double level single groove 'V' Block (Fig 3)

In this case, the 'V' Block will have two 'V' grooves on the top and bottom, and a single groove for clamping on either side.

Fig 3



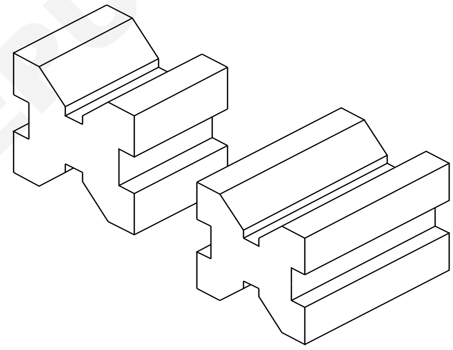
DOUBLE LEVEL SINGLE GROOVE V-BLOCK

TD20N122213

Matched pair 'V' Block (Figs 4 & 5)

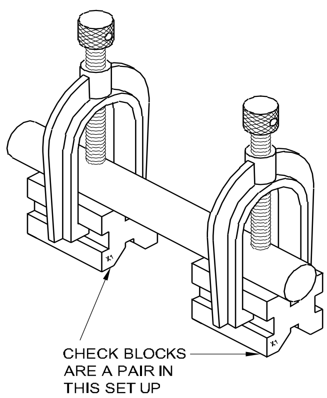
These blocks are available in pairs which have the same size and the same grade of accuracy. They are identified by the number or letter given by the manufacturer. These sets of blocks are used for supporting long shafts, parallel on machine tables or marking off tables.

Fig 4



TD20N122214

Fig 5



VEE BLOCKS SUPPORTING A SHAFT FOR MARKING OUT AND DRILLING

TD20N122215

Grades and materials

'V'Blocks are available in Grade A and Grade B.

Grade A 'V' Blocks

There are more accurate, and are available only up to 100 mm length. They are made of high quality steel.

Grade B 'V' Blocks

These blocks are not as accurate as the ones in Grade A. These blocks are used for general machine shop work. These blocks are available up to 300 mm length. These 'V' Blocks are made of closely grained cast iron.

Clamping devices for `V'-Blocks

'U' clamps are provided for holding cylindrical jobs firmly on 'V' Blocks. (Fig 6)

Designation

'V' Blocks are designated by the nominal size (length) and the minimum and maximum diameter of the workpiece capable of being clamped, and the grade and the number of the corresponding B.I.S. standard.

In the case of matched pairs, it should be indicated by the letter M.

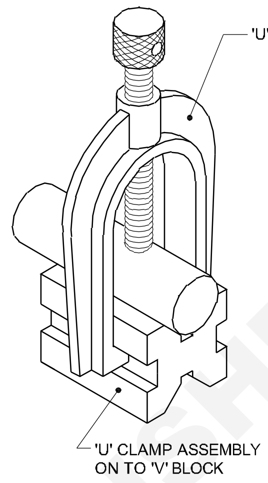
For 'V' Blocks with clamps it should be indicated as, 'WITH CLAMPS'.

Example

A 50 mm long (nominal size) 'V' Block capable of clamping workpieces between 5 to 40 mm in diameter and of Grade A will be designated as

- 'V' Block 50/5-40 A - I.S.2949.
- In the case of a matched pair, it will be designated as 'V' Block M 50/5-40 A I.S.2949.
- For 'V' Block supplied with clamps, the designation will be 'V' Block with clamp 50/5 40 A I.S. 2949.

Fig 6



Angle plates

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

Constructional features

Angle plates have two plane surfaces, machined perfectly flat and at right angles. 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)

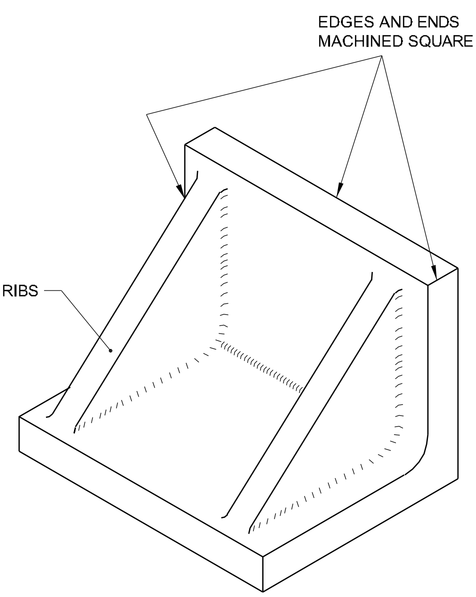
Among the three types of angle plates normally used, the plain solid angle plate is the most common. It has the two plane surfaces perfectly machined at 90° to each other. Such angle plates are suitable for supporting work-pieces during layout work. They are comparatively smaller in size.

Slotted type angle plate (Fig 2)

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

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

Fig 1



ANGLE PLATE

Fig 2

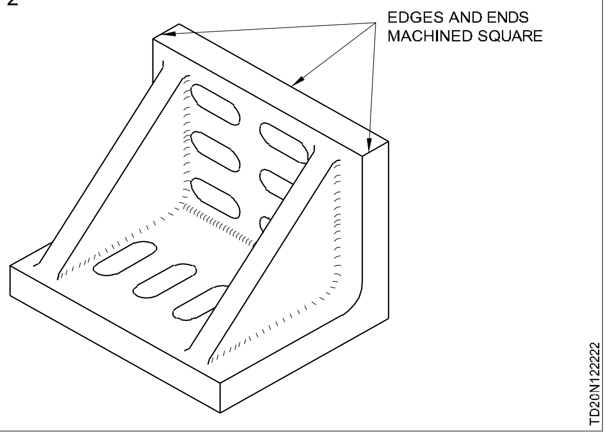


Fig 3

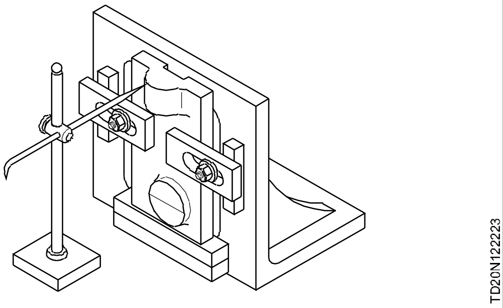
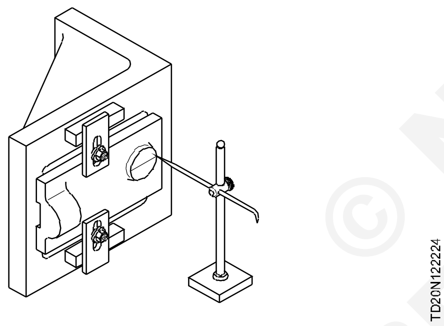


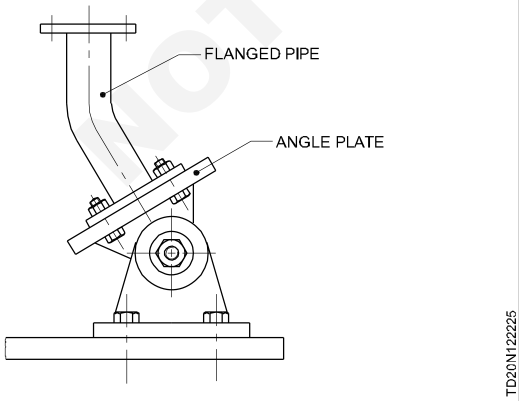
Fig 4



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.

Fig 5

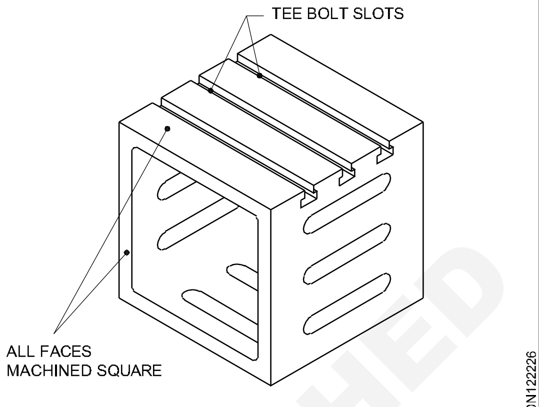


When both zeros coincide, the two plane surfaces are at 90° to each other. A bolt and nut are provided for locking in position.

Box 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 marking out or machining. This is a significant advantage. This has all the faces machined square to each other.

Fig 6



BOX ANGLE PLATE

Grades

Angle plates are 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 inspection 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

a Size 6 Grade 1

Box plate will be designated as - box angle plate 6 Gr 1 IS 623.

b Size 2 - Grade 2 angle plate will be designated as

Angle plate 2 Gr 2 I.S 623.

Table 1

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
Grade 2 only			

Care & Maintenance

- Clean before and after use.
- Apply oil after the use.

Parallel blocks

Objectives: At the end of this lesson you shall be able to
- **explain the types of parallels**
- **state the constructional features of parallel blocks**
- **state parallel blocks as per BIS recommended**
- **state the uses of parallel blocks.**

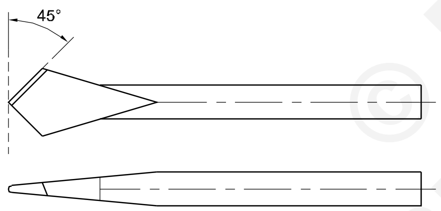
Parallel blocks of different types are used for setting workpieces for machining. The commonly used are of two types.

- Solid Parallels
- Adjustable Parallels

Solid parallels (Solid parallel blocks) (Fig 1)

This is the type of parallel which is very much used in machine shop work. They are made of steel pieces of rectangular cross section, and are available in different lengths and cross sectional sizes.

Fig 1



They are hardened and ground, and, sometimes, finished by lapping.

Parallels are machined to close limits, and are perfectly flat, square, and parallel throughout the length. These are made in pairs of identical dimensions.

Grades

Parallels are made in two grades - Grade A and Grade B. Grade A is meant for fine toolroom type of work, and Grade B for general machine shop work.

Adjustable parallels (Fig 2)

These consist of two tapered blocks sliding one over the other in a tongue and groove assembly. These types of parallels can be adjusted and set to different heights.

Uses

Solid and adjustable parallels are used for parallel setting of workpieces while machining. They are also useful for raising the workpieces held in vices or machine tables to provide better observation of the machining process. (Fig 3)

Fig 2

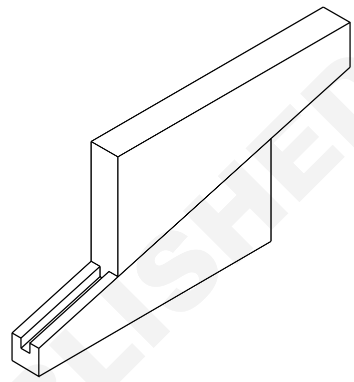
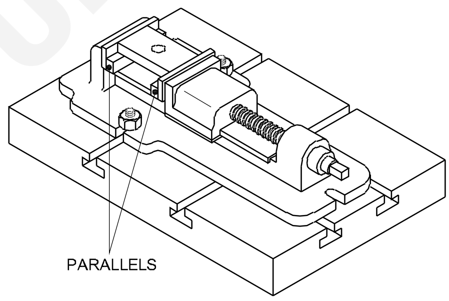


Fig 3



Parallels are made in pairs and should be used in matching pairs to ensure accuracy in set-up.

Care and maintenance

- Clean before and after the use.
- Apply oil after use
- Do not use as a hammer.

Sizes of parallels

These are given in TABLE 1, TABLE 2 and Fig 5.

Designation of parallels

Parallels are designated by the type, grade (for solid parallels only) size, and the number of the standard. (Fig 4)

Examples

Solid parallel A5 x 10 x 100 IS: 4241

Adjustable parallel 10 x 13 IS:4241

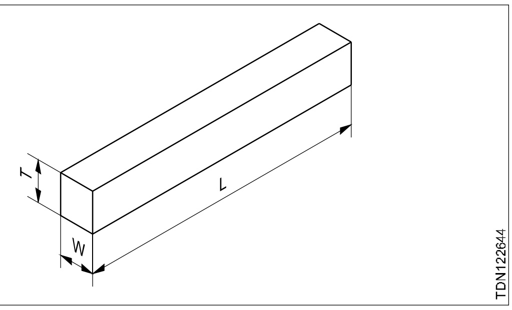


Fig 4

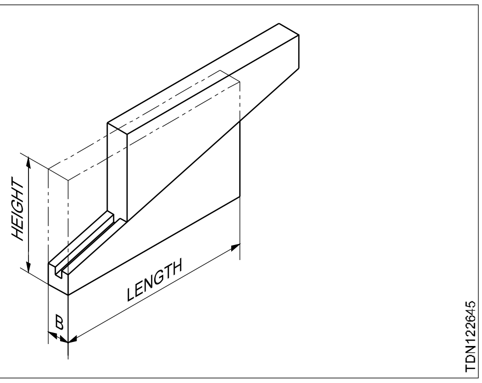


Fig 5

Table 1

Sizes of solid parallels

Grade	Size T.W.L.
A & B	5 x 10 x 100
A & B	10 x 20 x 150
A & B	15 x 25 x 150
A & B	20 x 35 x 200
A & B	25 x 45 x 250
A & B	30 x 60 x 250
A & B	35 x 70 x 300
B	40 x 80 x 350
B	50 x 100 x 400

Table 2

Range and size of Adjustable Parallels

Height Range	Length
10 - 13	40
13 - 16	50
16 - 20	60
20 - 25	65
25 - 30	70
30 - 40	85
40 - 50	100

Drill & Tap

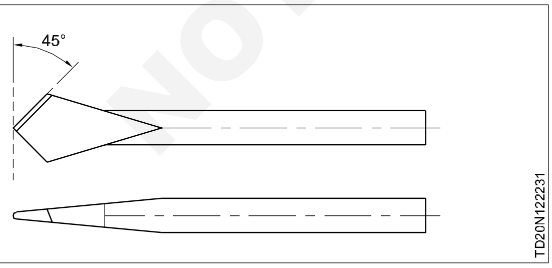
Objectives: At the end of this lesson you shall be able to
- **state the different types of drills**
- **explain the parts of a drill**
- **state the functions of each part of a drill.**

Drilling is a process of making holes on workpieces. The tool used for drilling is a drill and it is rotated with a downward pressure causing the tool to penetrate into the material.

Flat or spade drill (Fig 1)

This type of drill is used where the required size of twist drill is not available. It is made from a round tool steel piece which is forged to shape and ground to size, and then hardend and tempered. This type of drill is used for hand drilling (without power) or with a ratchet brace.

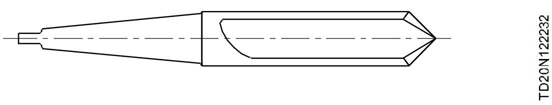
Fig 1



Straight fluted drill (Fig 2)

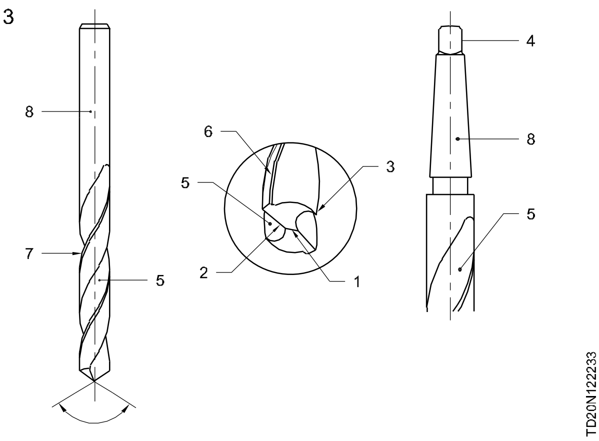
It has grooves or flutes running parallel to the drill axis. It is mainly used in drilling brass, copper or soft materials. This type of drill is inconvenient in standard practice as the chips do not come out from the hole automatically. It is used to drill sheet metals and cores in the castings.

Fig 2



Twist drills (Fig 3)

Fig 3



In this type, two spiral flutes or grooves run lengthwise around the body of the drill. It is the most common type of drill used for all purposes, and especially for faster drilling of accurate holes and for harder materials - in comparison with the other drills.

Parts of a twist drill (Fig 3)

Point

The cone shaped end which does the cutting is called the point. It consists of a dead centre (1), lips or cutting edge (2) and a heel (3).

Tang (4)

This is provided only on taper shank drills, for driving (giving torque to) the drill, which when overloaded, becomes twisted or gets sheared off.

Flutes (5)

Flutes are the spiral grooves which run to the length of the drill. The flutes help.

to form the cutting edge

to curl the chips and allow these to come out

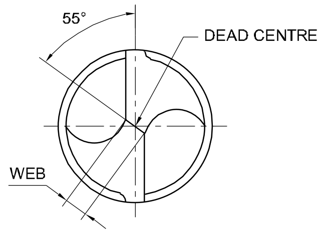
the coolant to flow to the cutting edge.

Shank (8)

This is the driving end of the drill which is fitted on to the machine. Shanks are of two types.

Taper shank, with Morse taper provided, is used for larger diameter drills, and the straight shank is used for smaller diameter drills.

Fig 4



Land/margin (6)

The land/margin is the narrow strip which extends to the entire length of the flutes. The diameter of the drill is measured across the land/margin.

Body clearance (7)

Body clearance is that part of the body which is reduced in diameter to cut down the friction between the drill and the hole being drilled.

Web (Fig 4)

Web is the metal column which seperates the flutes. it gradually increases in thickness towards the shank.

Material for twist drills: Twist drills, used in a machine shop, are usually made out of high speed steel. For drilling hard materials at higher cutting speeds, there are drills with carbide tips, brazed at the lips of the drill.

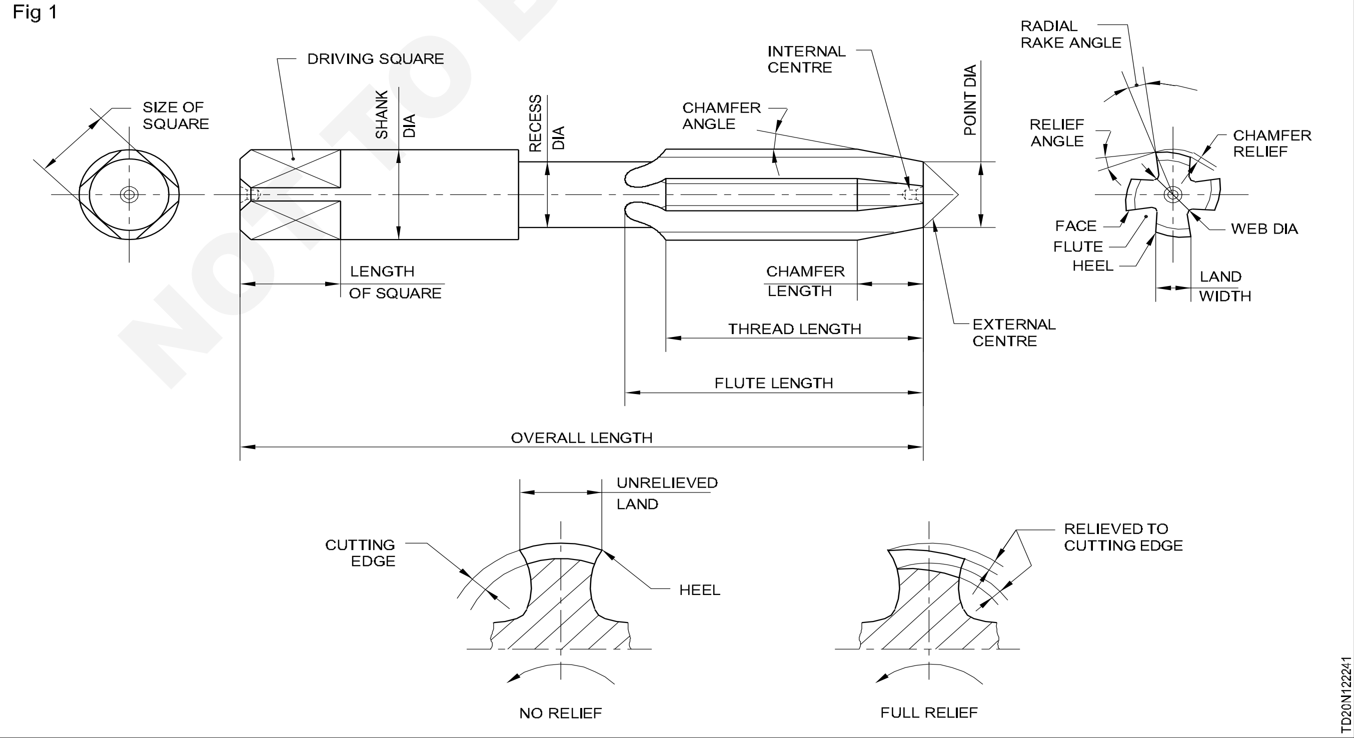
Hand taps and wrenches

Objectives: At the end of this lesson you shall be able to
- **state the uses of hand taps**
- **state the features of hand taps**
- **explain the different types of tap wrenches**
- **state the uses of the different types of wrenches.**

Use of hand taps: Hand taps are used for internal threading of components.

Features (Fig 1): They are made from high carbon steel or high speed steel, hardened, tempered and ground.

Fig 1



The threads are cut on the periphery and are accurately finished.

To form the cutting edges, flutes are cut across the thread.

The end of the shank of the tap is made of square shape for the purpose of holding and turning the taps.

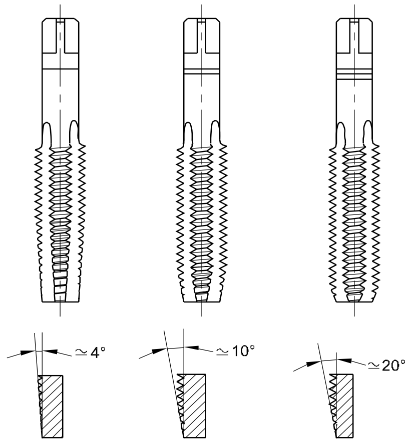
The end of the taps are chamfered (taper lead) for assisting, aligning and starting of the thread.

The size of the taps, the thread standard, the pitch of the thread, the diameter of the tapping hole are usually marked on the shank.

Marking on the shank are also made to indicate the type of tap i.e. first, second and plug.

Types of taps in a set: Hand taps for a particular thread are available as a set consisting of three pieces. (Fig 2)

Fig 2



These are:

– first tap or taper tap

– second tap or intermediate tap

– plug or bottoming tap

These taps are identical in all features except in the taper lead. The taper tap is to start the thread. It is possible to form full threads by the taper tap in through holes which are not deep.

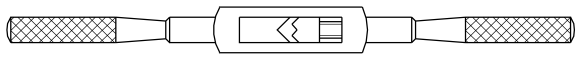
The bottoming tap (plug) is used to finish the threads of a blind hole to the correct depth.For identifying the type of taps quickly - the taps are either numbered 1,2 and 3 or rings are marked on the shank.

The taper tap has one ring, the intermediate tap has two and the bottoming tap has three rings. (Fig 2)

Tap wrenches: Tap wrenches are used to align and drive the hand taps correctly into the hole to be threaded.

Tap wrenches are of different types, such as double-ended adjustable wrench, T-handle tap wrench, solid type tap wrench etc.

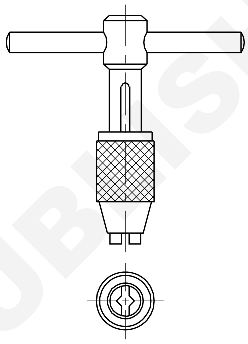
Fig 3



Double-ended adjustable tap wrench or bar type tap wrench (Fig 3)**:** This is the most commonly used type of tap wrench. It is available in various sizes - 175, 250, 350 mm long. These tap wrenches are more suitable for large diameter taps, and can be used in open places where there is no obstruction to turn the tap.

It is important to select the correct size of wrench.

Fig 4



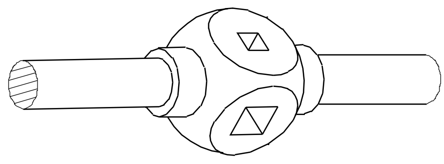
T-handle tap wrench (Fig 4)**:** These are small, adjustable chucks with two jaws and a handle to turn the wrench.

This tap wrench is useful to work in restricted places, and is turned with one hand only. Most suitable for smaller sizes of taps.

Solid type tap wrench (Fig 5)**:** These wrenches are not adjustable.

They can take only certain sizes of taps. This eliminates the use of wrong length of the tap wrenches, and thus prevents damage to the taps.

Fig 5



Tap drill size

Objectives: At the end of this lesson you shall be able to
- **state tap drill size**
- **state the tap drill sizes for metric and BSW thread tables**
- **state the tap drill sizes for ISO metric ISO inch.**

What is tap drill size?

Before a tap is used for cutting internal threads, a hole is to be drilled. The diameter of the hole should be such that it should have sufficient material in the hole for the tap to cut the thread.

Tap drill sizes for different threads

ISO metric thread

Tap drill size for M 10 x 1,5 thread

Minor diameter = Major diameter - (2 x depth)

Depth of thread = 0.6134 x pitch of a screw

2 depth of thread = 0.6134 x 2 x pitch

= 1.226 x 1.5 mm

= 1.839 mm

Minor dia. = 10 mm - 1.839 mm

= 8. 161 mm or 8.2 mm.

This tap drill will produce 100% thread because this is equal to the minor diameter of the tap. For most fastening purposes a 100% formed thread is not required.

A standard nut with 60% thread is strong enough to be tightened until the bolt breaks without stripping the thread. Further it also requirees a greater force for turning the tap if a higher percentage formation of thread is required.

Considering this aspect, a more practical approach for determining the tap drill sizes is

Tap drill size = major diameter minus pitch

= 10mm - 1.5 mm

= 8.5 mm.

Compare this with the table of tap drill sizes for ISO metric threads.

BSW inch (unified) threads formula

Tap drill size =

$$\text{Major diameter} - \frac{1\ \text{inch}}{\text{No.of threads per inch}}$$

For calculating the tap drill size for 5/8” UNC thread

Tap drill size = 5/8” - 1/11”

= 0.625” - 0.091”

= 0.534”

The next drill size is 17/32” (0.531 inches).

Compare this with the table of drill sizes for unified inch threads.

What will be the size for the following threads?

a M20

b BSW 3/8

Refer to the chart for determining the pitches of the thread.

Table for tap drill dize

Nominal diameter M.M	ISO Metric (60°)		B.S.W. (55°)		Tap drill sizes
	Pitch	Tap drill sizes	Nominal diameter (inch)	Threads per inch (mm)	
3	0.5	2.05	1/8	40	2.5
4	0.7	3.30	5/32	32	3.2
5	0.8	4.20	3/16	24	4.0
6	1.0	5.00	1/4	20	5.0
8	1.25	6.80	5/16	18	6.0
10	1.50	8.0	3/8	16	8.0
12	1.75	10.20	1/2	12	10
14	2.00	12.00	9/16	12	12.5
16	2.00	14.00	5/8	11	14.00
18	2.50	15.50	3/4	10	16.00
20	2.50	17.50	13/16	10	18.00
22	2.50	19.50	7/8	9	19.5
24	3.00	21.00	1	8	22.2

Die and die stock

Objectives: At the end of this lesson you shall be able to
- **explain the different types of dies**
- **state the features of each type of die**
- **state the use of each type of die**
- **name the type of die stock for each type of die.**

Uses of dies

Threading dies are used to cut external threads on cylindrical workpieces. (Fig 1)

Types of Dies

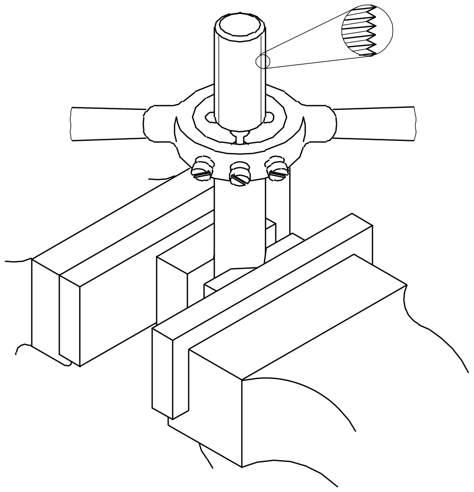
The following are the different types of dies.

– Circular split die (Button die)

– Half die

– Adjustable screw plate die

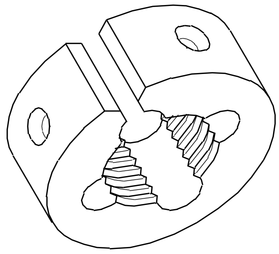
Fig 1



Circular split die/button die (Fig 2)

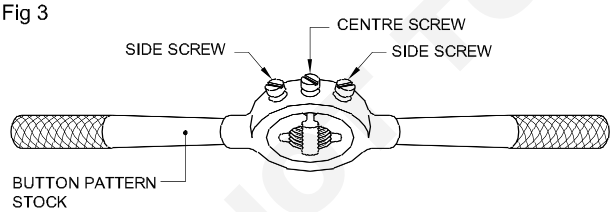
This has a slot cut to permit slight variation in size.

Fig 2



When held in the die stock, variation in the size can be made by using the adjusting screws. This permits increasing or decreasing of the depth of cut. When the side screws are tightened the die will close slightly.(Fig 3) For adjusting the depth of the cut, the centre screw is advanced and locked in the groove. This type of die stock is called button pattern stock.

Fig 3



Half die (Fig 4)

Half dies are stronger in construction.

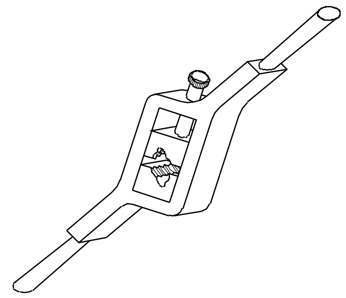
Adjustments can be made easily to increase or decrease the depth of cut.

These dies are available in matching pairs and should be used together.

By adjusting the screw of the die stock, the die pieces can be brought closer together or can be moved apart.

They need a special die holder.

Fig 4



Adjustable screw plate die (Fig 5)

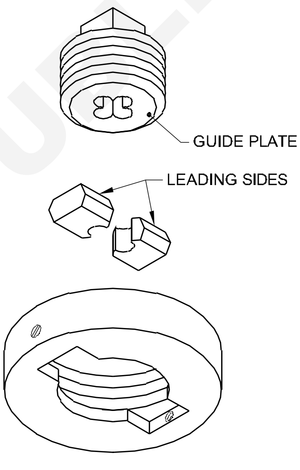
This is another type of a two-piece die similar to the half die.

This provides greater adjustment than the split die.

The two die halves are held securely in a collar by means of a threaded plate (guide plate) which also acts as a guide while threading.

When the guide plate is tightened after placing the die pieces in the collar, the die pieces are correctly located and rigidly held.

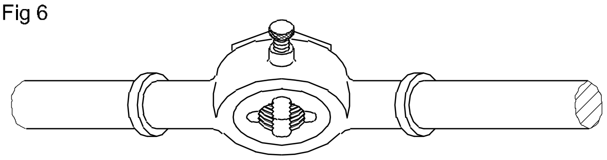
Fig 5



The die pieces can be adjusted, using the adjusting screws on the collar.

This type of die stock is called quick cut die stock.(Fig 6)

Fig 6



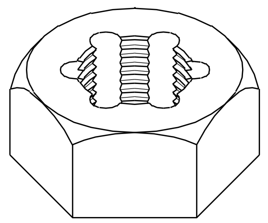
The bottom of the die halves is tapered to provide the lead for starting the thread. On one side of each die head, the serial number is stamped.

Both pieces should have the same serial numbers.

Die nut (Solid die) (Fig 7)

The die nut is used for chasing or reconditioning the damaged threads.

Fig 7



Die nuts are not to be used for cutting new threads.

The die nuts are available for different standards and sizes of threads.

The die nut is turned with a spanner.

Blank size for external threading

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

- **state the diameter of blank size for external thread cutting.**

Why should the blank size be less?

It has been observed from practice that the threaded diameters of steel blanks show a slight increase in diameter. Such increase in the diameter will make the assembly of external and internal threaded components very difficult. To overcome this, the diameter of the blank is slightly reduced before commencing the threading.

What should be the blank size?

The diameter of the blank should be less by 1/10th of the pitch of the thread.

Example

For cutting the thread of M12 with 1.75 mm pitch the diameter of the blank is 11.80.

Formula, D = d – p/10

= 12 mm – 0.175 mm

= 11.825 or 11.8 mm.

d = diameter of bolt

D = the blank diameter

p = pitch of thread

Calculate the blank size for preparing a bolt of M16 x 1.5.

Answer

____

____

____

Screw thread and elements

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

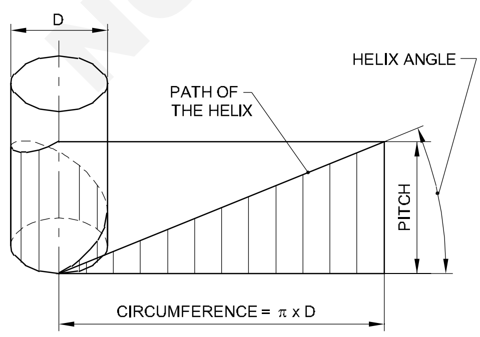
- **state the uses of screw threads**
- **explain the difference between external and internal threads**
- **state the elements of screw threads.**

What is a screw thread?

A screw thread is a ridge of uniform section formed helically on the surface of a cylindrical body. (Fig 1)

An external screw thread is formed on the outer surface of a cylindrical part. Examples: bolts, screws, studs, threaded spindles, etc. (Fig 1)

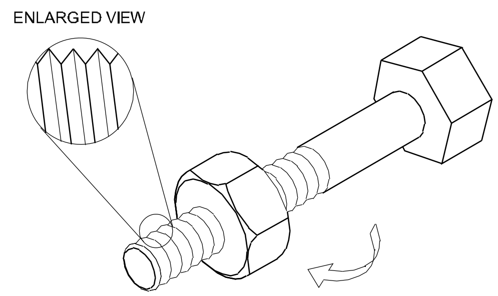
Fig 1



An internal screw thread is formed on the inner surface of a hollow cylindrical part. Examples: nuts, threaded lids etc.

External threads and internal threads are assembled together for different engineering uses. (Fig 2)

Fig 2

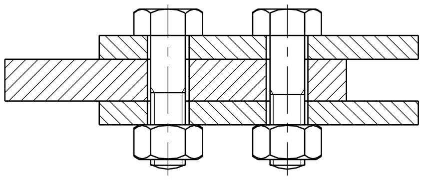


Uses of screw threads

Screw threads are used

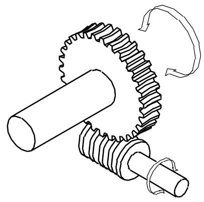
- as fasteners to hold together and dismantle components when needed (Fig 3)

Fig 3



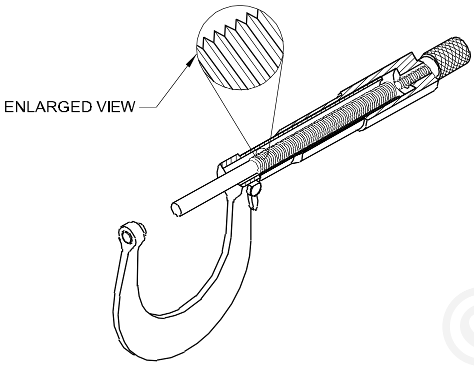
- to transmit motion on machines from one unit to another (Fig 4)

Fig 4



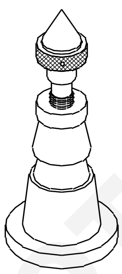
- to make accurate measurements (Fig 5)

Fig 5



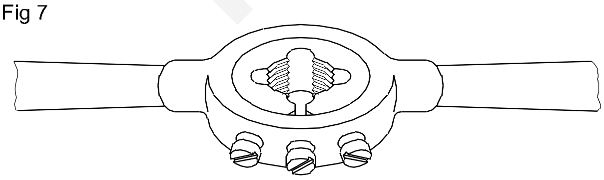
- to apply pressure (Fig 6)

Fig 6



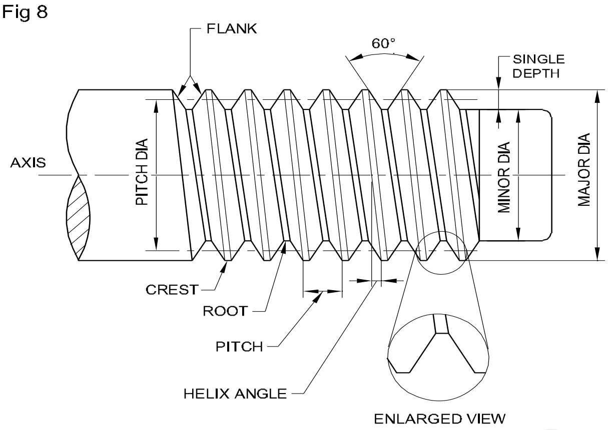
- to make adjustments. (Fig 7)

Fig 7



- Parts of a screw thread (Fig 8)

Fig 8



Crest

The top surface joining the two sides of a thread.

Root

The bottom surface joining the two sides of adjacent threads.

Flank

The surface joining the crest and the root.

Thread angle

The included angle between the flanks of adjacent threads.

Depth

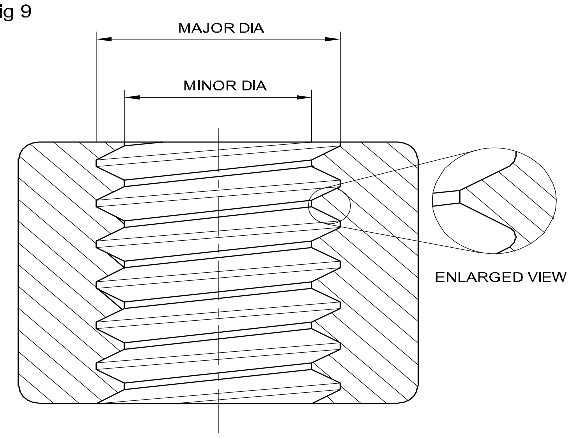
The perpendicular distance between the roots and crest of the thread.

Major diameter

In the case of external threads it is the diameter of the blank on which the threads are cut and in the case of internal threads it is the largest diameter after the threads are cut that are known as the major diameter. (Fig 9)

This is the diameter by which the sizes of screws are stated.

Fig 9



Minor diameter

For external threads, the minor diameter is the smallest diameter after cutting the full thread. In the case of internal threads, it is the diameter of the hole drilled for forming the thread which is the minor diameter.

Pitch diameter (effective diameter)

The diameter of the thread at which the thread thickness is equal to one half of the pitch.

Pitch (Fig 8)

It is the distance from a point on one thread to a corresponding point on the adjacent thread measured parallel to the axis.

Lead

Lead is the distance a threaded component moves along the matching component during one complete revolution. For a single start thread the lead is equal to the pitch.

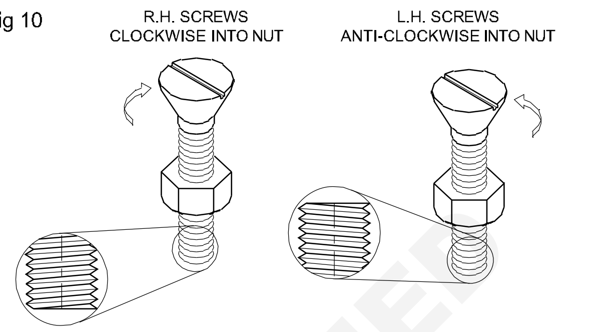
Helix angle

The angle of inclination of the thread to the imaginary perpendicular line.

Hand

The direction in which the thread is turned to advance. A right hand thread is turned clockwise to advance, while a left hand thread is turned anticlockwise. (Fig 10)

Fig 10



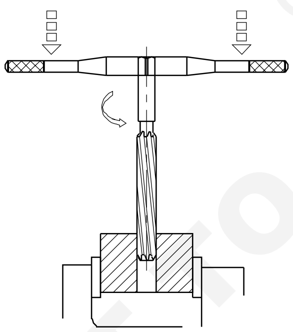
Reamers

Objectives: At the end of this lesson you shall be able to
- **state the use of reamers**
- **state the advantages of reaming**
- **explain difference between hand and machine reaming**
- **explain the elements of a reamer and state their functions.**

What is a reamer?

A reamer is a multi-point cutting tool used for enlarging by finishing previously drilled holes to accurate sizes.(Fig 1)

Fig 1



Advantages of 'reaming'

Reaming produces

- high quality surface finish
- dimensional accuracy to close limits.

Also small holes which cannot be finished by other processes can be finished.

Classification of reamers

Reamers are classified as hand reamers and machine reamers. (Figs 2 and 3)

Reaming by using hand reamers is done manually for which great skill is needed.

Fig 2

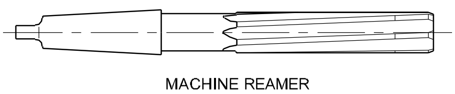
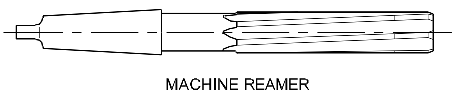


Fig 2



Machine reamers are fitted on spindles of machine tools and rotated for reaming.

Machine reamers are provided with morse taper shanks for holding on machine spindles.

Hand reamers have straight shanks with 'square' at the end, for holding with tap wrenches. (Fig 3)

Parts of a hand reamer

The parts of a hand reamer are listed hereunder. Refer to Fig 4.

Axis

The longitudinal centre line of the reamer.

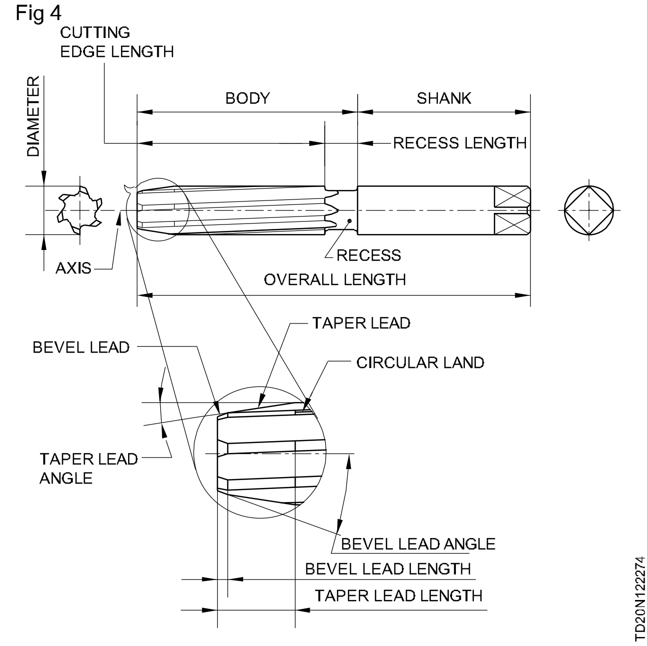
Body

The portion of the reamer extending from the entering end of the reamer to the commencement.

Recess

The portion of the body which is reduced in diameter below the cutting edges, pilot or guide diameters.

Fig 4



Shank

The portion of the reamer which is held and driven. It can be parallel or tapered.

Circular land

The cylindrically ground surface adjacent to the cutting edge on the leading edge of the land.

Bevel lead

The bevel lead cutting portion at the entering end of the reamer cutting its way into the hold. It is not provided with a circular land.

Taper lead

The tapered cutting portion at the entering end to facilitate cutting and finishing of the hole. It is not provided with a circular land.

Bevel lead angle

The angle formed by the cutting edges of the bevel lead and the reamer axis.

Taper lead angle

The angle formed by the cutting edges of the taper and the reamer axis.

Terms relating to cutting geometry flutes

The grooves in the body of the reamer to provide cutting edges, to permit the removal of chips, and to allow the cutting fluid to reach the cutting edges. (Fig 5)

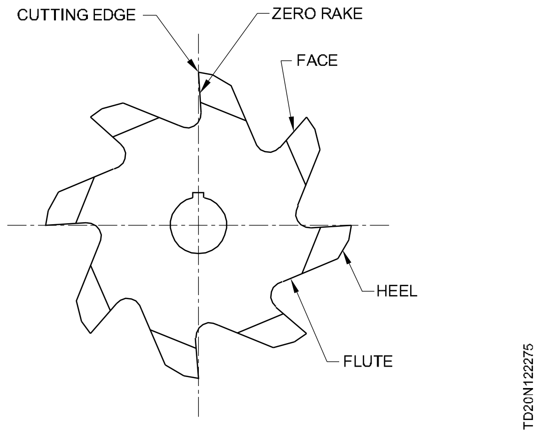
Heel

The edge formed by the intersection of the surface left by the provision of a secondary clearance and the flute. Fig 5)

Cutting edge

The edge formed by the intersection of the face and the circular land or the surface left by the provision of primary clearance. (Fig 5)

Fig 5



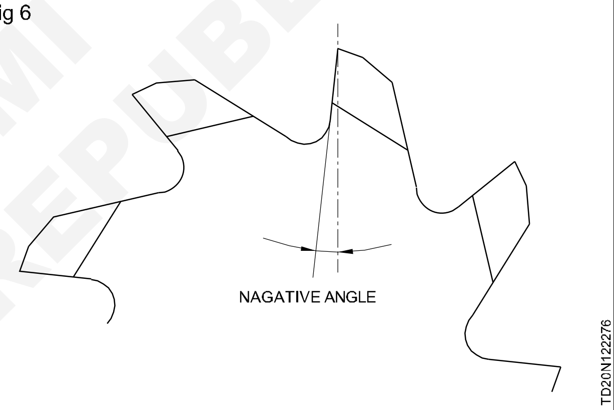
Face

The portion of the flute surface adjacent to the cutting edge on which the chip impinges as it is cut from the work. (Fig 5)

Rake angles

The angles in a diametral plane formed by the face and a radial line from the cutting edge. (Fig 6)

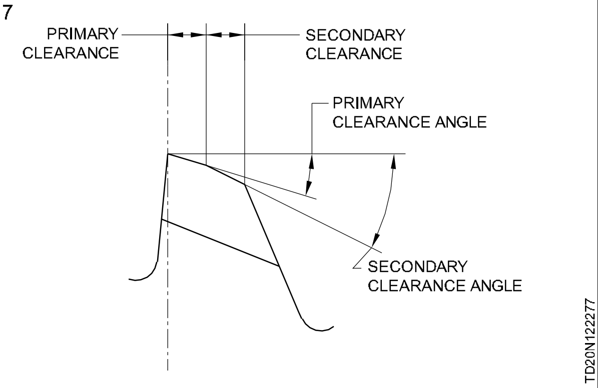
Fig 6



Clearance angle

The angles formed by the primary or secondary clearances and the tangent to the periphery of the reamer at the cutting edge. They are called primary clearance angle and secondary clearance angle respectively. (Fig 7)

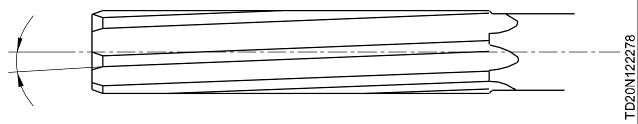
Fig 7



Helix angle

The angle between the edge and the reamer axis. (Fig 8)

Fig 8



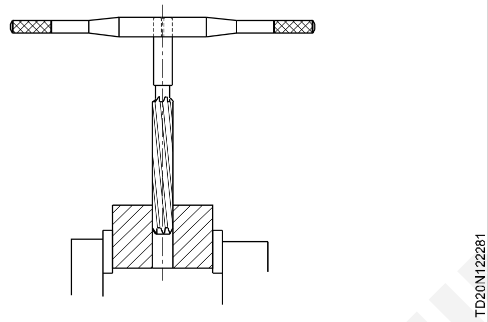
Hand reamers

Objectives: At the end of this lesson you shall be able to
- **state the general features of hand reamers**
- **explain the types of hand reamers**
- **explain difference between the uses of straight fluted and helical fluted reamers**
- **state the materials from which reamers are made and specify reamers.**

General features of hand reamers (Fig 1)

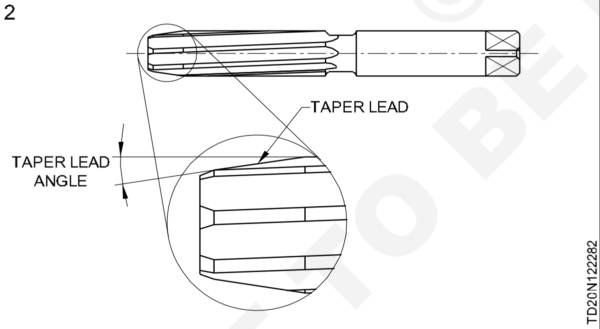
Hand reamers are used to ream holes manually using tap wrenches.

Fig 1



These reamers have a long taper lead. (Fig 2) This allows to start the reamer straight and in alignment with the hole being reamed.

Fig 2

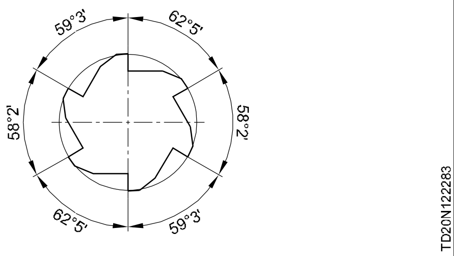


Most hand reamers are for right hand cutting.

Helical fluted hand reamers have left hand helix. The left hand helix will produce smooth cutting action and finish.

Most reamers, machine or hand, have uneven spacing of teeth. This feature of reamers helps to reduce chattering while reaming. (Fig 3)

Fig 3



Types, Features and Functions

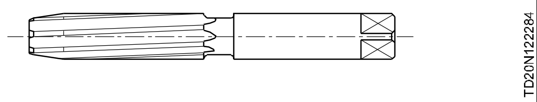
Hand reamers with different features are available for meeting different reaming conditions. The commonly used types are listed here under.

Parallel hand reamer with parallel shank (Fig 4)

A reamer which has virtually parallel cutting edges with taper and bevel lead. The body of the reamer is integral with a shank. The shank has the nominal diameter of the cutting edges. One end of the shank is square shaped for tuning it with a tap wrench. Parallel reamers are available with straight and helical flutes. This is the commonly used hand reamer for reaming holes with parallel sides.

Reamers commonly used in workshop produce H8 holes.

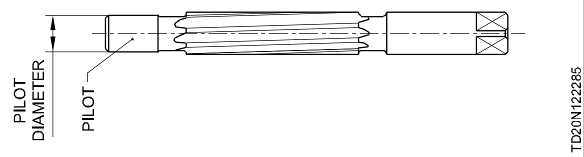
Fig 4



Hand reamer with pilot (Fig 5)

For this type of reamer, a portion of the body is cylindrically ground to form a pilot at the entering end. The pilot keeps the reamer concentric with the hole being reamed.

Fig 5



Socket reamer with parallel shank (Figs 6 & 7)

This reamer has tapered cutting edges to suit metric morse tapers. The shank is integral with the body, and is square shaped for driving. The flutes are either straight or helical. The socket reamer is used for reaming internal morse tapered holes.

Fig 6

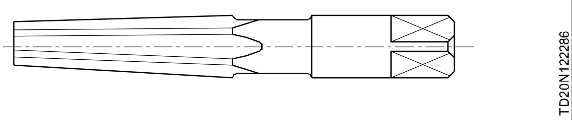
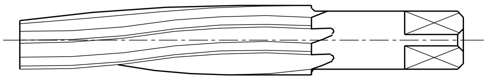


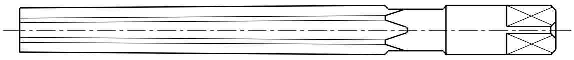
Fig 7



Taper pin hand reamer (Fig 8)

This reamer has tapered cutting edges for reaming taper holes to suit taper pins. A taper pin reamer is made with a taper pin of 1 in 50. These reamers are available with straight or helical flutes.

Fig 8



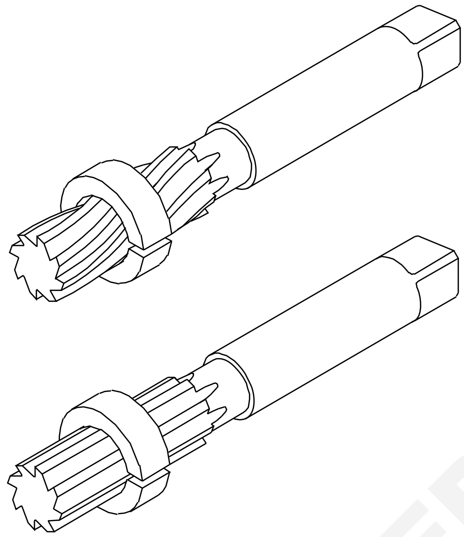
Use of straight and helical fluted reamers (Fig 9)

Straight fluted reamers are useful for general reaming work. Helical fluted reamers are particularly suitable for reaming holes with keyway grooves or special lines cut into them. The helical flutes will bridge the gap and reduce binding and chattering.

Material of hand reamers

When the reamers are made as a one-piece construction, high speed steel is used. When they are made as two-piece construction then the cutting portion is made of high speed steel while the shank portion is made of carbon steel. They are butt-welded together before manufacturing.

Fig 9



Specifications of a reamer

To specify a reamer the following data is to be given.

- Type
- Flute
- Shank end
- Size

Example

Hand reamer, Straight flute, Parallel shank of Ø 20 mm.

Machine reamers

Objectives: At the end of this lesson you shall be able to
- **state the different types of commonly used machine reamers**
- **state the features and uses of machine jig reamers**
- **state the features and advantages of shell reamers**
- **explain the different types of machine reamers**
- **state the advantages of adjustable reamers**
- **state the advantages of floating reamer-holders**
- **explain the different taper reamers**
- **state the advantages of step drilling while taper reaming.**

Reamers are used to finish previously drilled holes accurately to smooth finish. This can be done either by hand or on machine. Reamers used on machines are called machine reamers. The basic difference between hand and machine reamers is the lead angle at the cutting end. (Fig 1) Hand reamers will have long taper lead while machine reamers will have a short bevel lead.

The shank end of the hand reamers will be square to facilitate reaming using tap wrenches. Machine reamers of small diameters will have parallel shank and the larger reamers are provided with taper shanks.

Types of machine reamers

Solid fluted machine reamer (jobber reamer)

This is idenitical to a hand reamer. These reamers are either straight fluted or with left hand helix to prevent the tendency of 'cork screwing' when rotated clockwise for reaming. (Fig 2)

Chucking reamers (Fig 3)

These reamers are similar to jobber's reamers but have shorter and deeper flutes, and are available as straight or helical fluted. This is a side cutting reamer and cuts along the full length of the land and produces smooth and accurately sized holes. The ends of these reamers are slightly chamfered to initiate the cutting action.

Fig 1

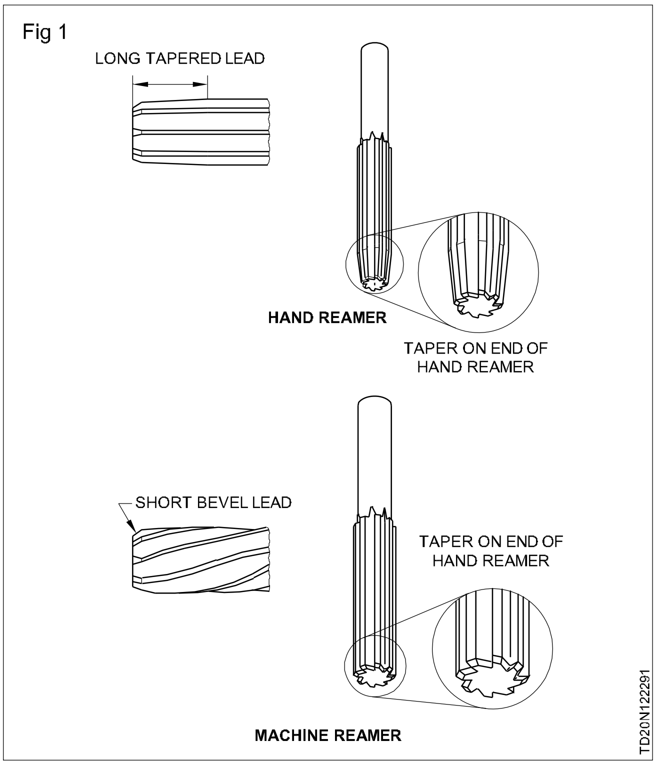


Fig 2

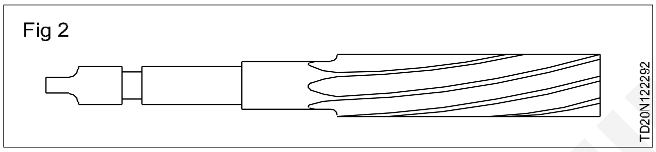
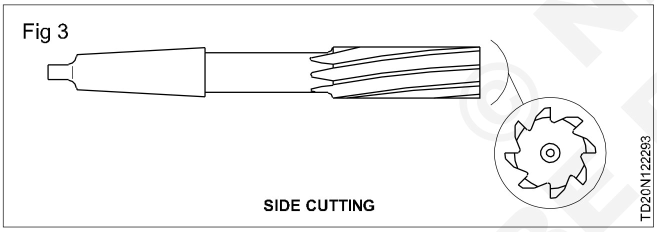


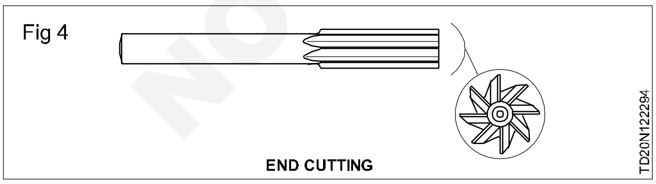
Fig 3



Rose reamer/rose chucking reamer (Fig 4)

This reamer is designed to cut on its end. The flutes help in chip clearance and act as guides while cutting. This is used when a considerable amount of metal is to be removed, and the finish is not very critical. This is sometimes used as a roughing tool. Final finishing is done with other finishing reamers.

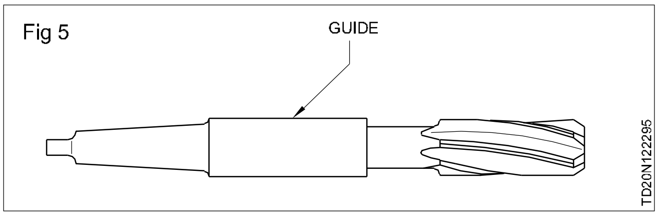
Fig 4



Machine jig reamers (Fig 5)

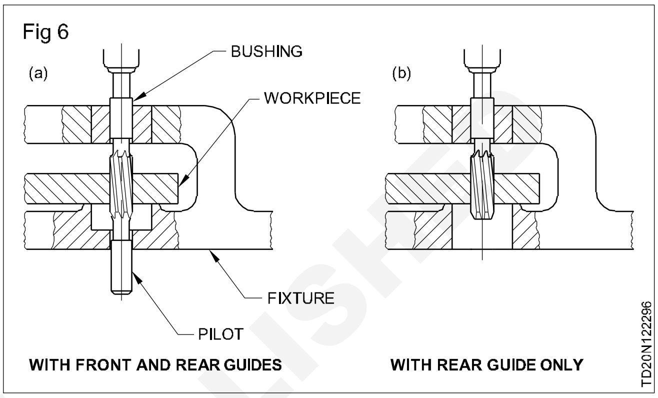
When reaming is done using jigs, the bushings of the jigs can be used to guide the reamer. Machine jig reamers are provided with special, long guide surfaces of standard diameters according to the diameter of the reamer.

Fig 5



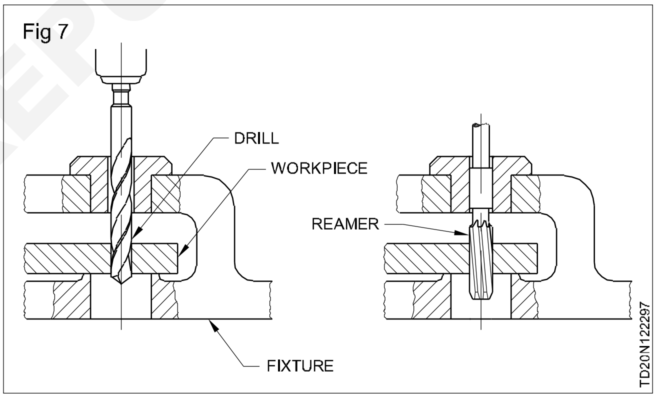
Reamers of this type are available with rear guides only or with front and rear guides. (Fig 6) They can produce very accurate holes in the spindle and bushes are aligned accurately. While reaming deep holes it is better to select the jig reamers with guides on both ends.

Fig 6



When reaming is carried out immediately after drilling in the same setting, renewable bushes are used on the jig. (Fig 7)

Fig 7



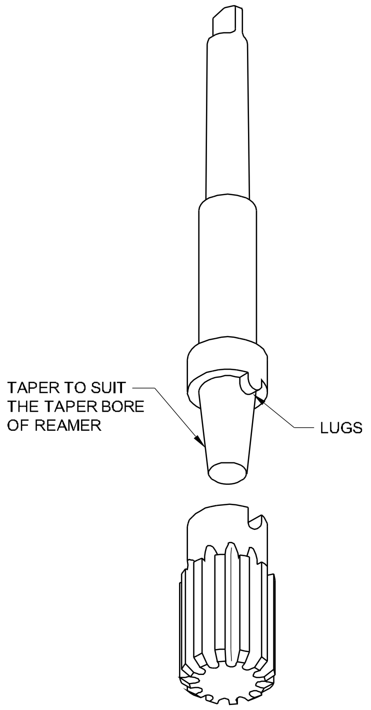
Shell reamer (Fig 8)

The shell reamer is an independent reaming unit which has a slightly tapered hole through the centre that permits the reamer to be held on a separate shank or arbor that has driving lugs. Several sizes of reamers may be used with one shank. Shell reamers are made with either fluted teeth having clearance, or the rose chucking type which cuts on the end only.

Adjustable machine reamer

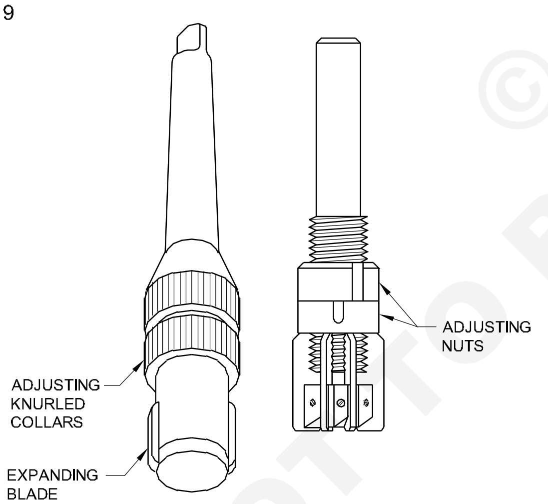
These reamers are easy to adjust when worn out. They can be re- sharpened and adjusted back to the correct size. As such these reamers have longer working life than ordinary reamers.

Fig 8



Adjustable remers have adjustable insert blades. (Fig 9) When worn out or damaged, the blade can be easily replaced. These reamers are not meant to produce holes of different sizes. However, they can be used to increase the hold slinghtly.

Fig 9



Note: Similarly, adjustable reamers are also available for hand reaming. (Fig 10)

The size of the reamer can be adjusted by moving the blades in the tapered slots using the nuts provided on either end of the blade.

Reamers with floating holders: While machine reaming, the taper lead at the reamer end guides into the hole being reamed. In the event of any misalignment, the hole being reamed can enlarge at the starting end. (Fig 11) This can be avoided by the use of reamers with floating holders. Floating holders compensate minor discrepancies in the axis alignment. Floating holders are available with angular floats and parallel floats. (Fig 12)

Fig 10

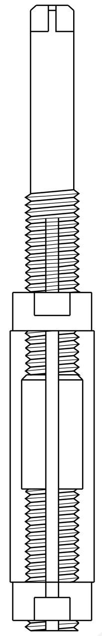


Fig 11

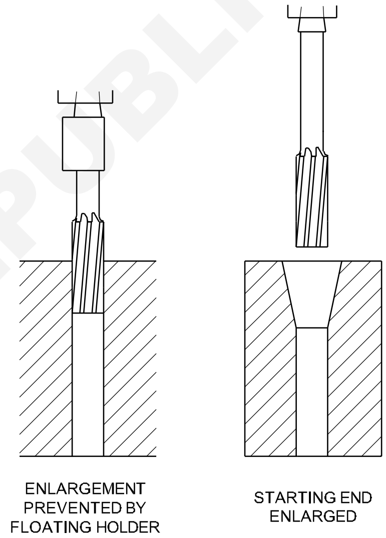
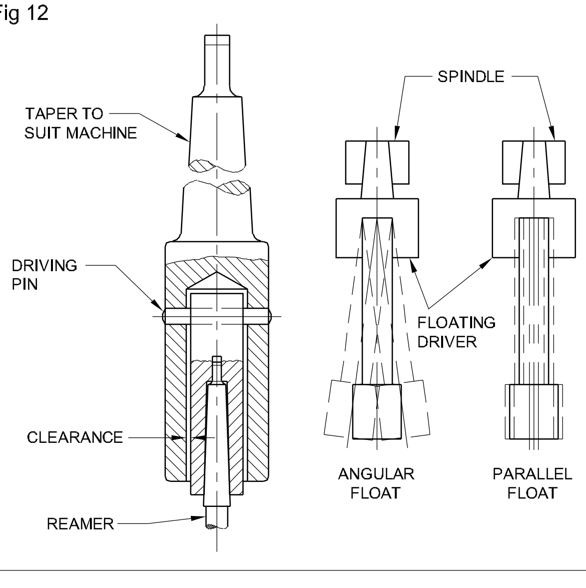


Fig 12



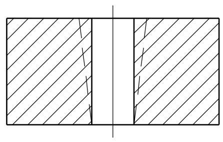
Taper reamers

Taper machine reamers are manufactured in all standard tapers and with tapered shank. They can be mounted directly in the spindle of the machine.

For taper reaming the diameter on the hole drilled is slightly smaller than the finished diameter of the small end. of the taper.

While reaming, the taper reamer will have to remove more material at the big end and less material at the small end. (Fig 13) While cutting, the entire length of the reamer will be in contact will the workpiece. This can cause chatter marks and poor finish.

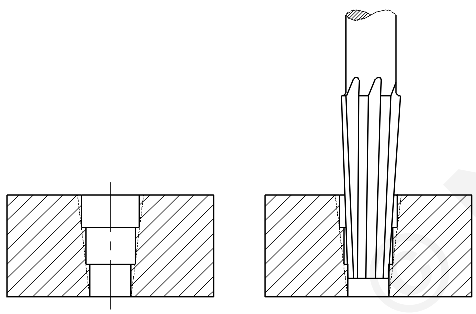
Fig 13



TD20N12229D

For better results use a roughing reamer first and then finish with a finishing reamer. Step drilling the hole will help to reduce the strain on taper reamers. (Fig 14)

Fig 14

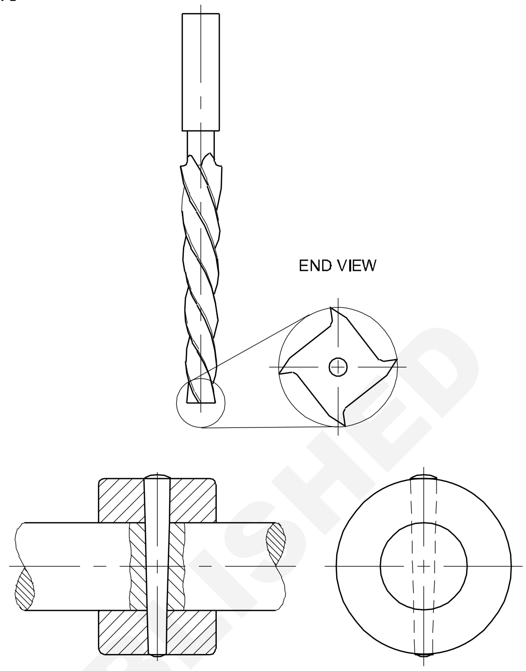


TD20N12229E

Taper pin machine reamers (Fig 15)

These reamers are used for reaming taper holes needed for fitting taper pins.

Fig 15



TD20N12229F

Drill size for reaming

Objective: At the end of this lesson you shall be able to
- **state the hole size for reaming.**

For reaming with a hand or a machine reamer, the hole drilled should be smaller than the reamer size.

The drilled hole should have sufficient metal for finishing with the reamer. Excessive metal will impose a strain on the cutting edge of the reamer and damage it.

Calculating drill size for reamer

A method generally practised in workshop is by applying the following formula.

Drill size = Reamed size – (Undersize + Oversize)

Finished size

Finished size is the diameter of the reamer.

Undersize

Undersize is the recommended reduction in size for different ranges of drill diameter. (Table 1)

Table 1

Undersizes for reaming

Diameter of ready reamed hole (mm)	Undersize of rough bored hole (mm)
under 5	0.1.....0.2
5......20	0.2.....0.3
21....50	0.3.....0.5
over 50	0.5.....1

Drilling machines - Types & Application

Objectives: At the end of this lesson you shall be able to
- **state types of drilling machines**
- **explain application of drilling machines**

The drilling machine is one of the most important machine tools in a workshop and second to lathe. It was primarily designed to originate a hole.

Types of drilling machine

Drilling machines are made in many different types and sizes, each designed to handle a class of work or specific job to the best advantage.

The different types of drilling machines are

1 Portable drilling machine
2 Sensitive drilling machine
   a Bench mounting
   b Floor mounting
3 Upright drilling machine
4 Radial drilling machine
5 Gang drilling machine
6 Multiple spindle drilling machine
7 Automatic drilling machine
8 Deep hole drilling machine

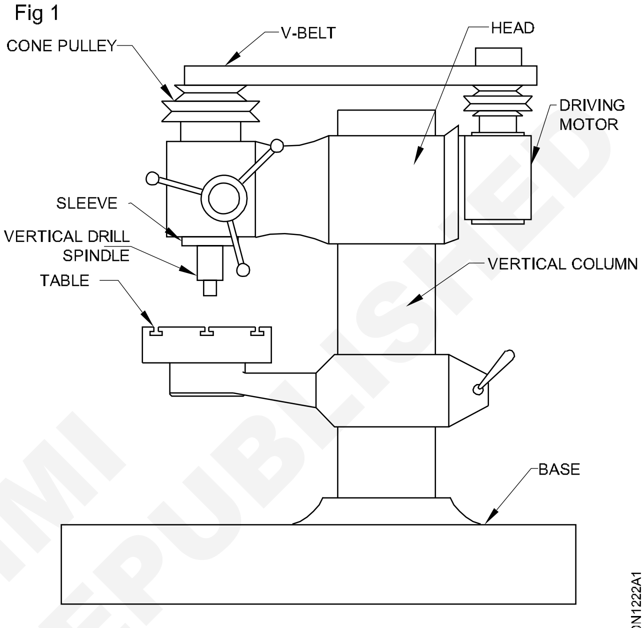
Portable Drilling machine

As the name implies this type of drilling machine can be operated with ease anywhere in the workshop and is used for drilling holes in work pieces in any position which cannot be drilled in a standard in a standard drilling machine. Some of the portable machines are operated by hand power, but most of the machines are driven by individual motor. The entire drilling mechanism including the motor is compact and small in size. The motor is usually of universal type which may be driven by both A.C. and D.C. The maximum size of the drill that it can be accommodated is not more than 12 to 18mm. the machine is operated at high speed as smaller size drills are only used. Some of the portable machines are driven by pneumatic power.

Sensitive Drilling Machine (Fig 1&2)

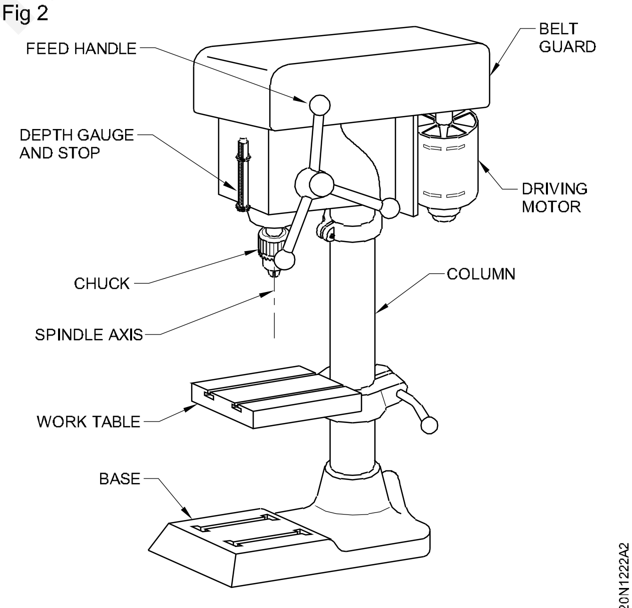
The sensitive drilling machine is a small machine designed for drilling a small hole at high speed in light jobs, the base of the machine may be mounted on a bench or on the floor. It consists of a vertical column, a horizontal table, a head supporting the motor and driving mechanism, and a vertical spindle for driving and rotating the drill. There is no arrangement for any automatic feed of the drill spindle. The drill is fed into the work by purely hand control. High speed and hand feed are necessary for drilling small holes. High speeds are necessary to attain required cutting speed by small diameter drill. Hand feed permits the operator to feel or sense the progress of the drill into the work, so that the drill becomes worn out or jams on any account, the pressure on the drill may be released immediately to prevent it from breaking.

Fig 1



SENSITIVE DRILLING MACHINE

Fig 2

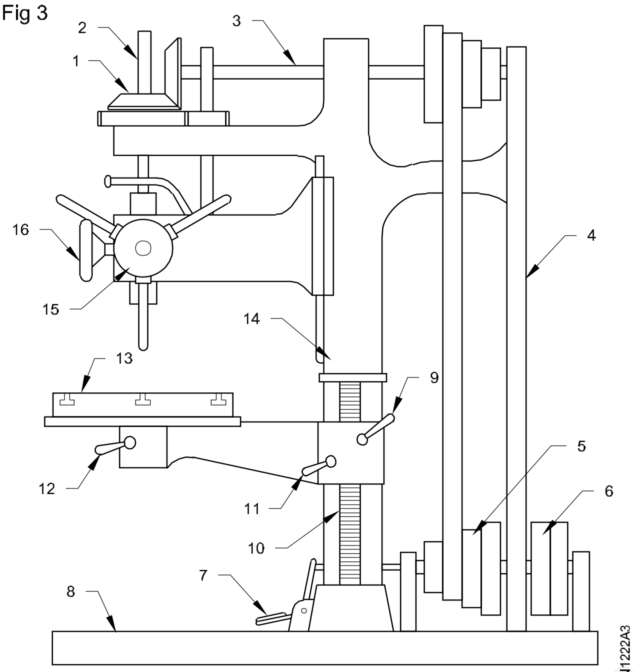


BENCH DRILLING MACHINE

Box column section upright drilling machine (Fig 3)

The upright drilling machine with box column section has square table fitted on the slides at the front face of the machine column. Heavy box column gives the machine strength and rigidity. The table is raised or lowered by an elevating screw that gives additional support to the table. These special features permit the machine to work with heavier workpieces, and holes more than 50mm in diameter can be drilled by it.

Fig 3



UPRIGHT PILLAR DRILLING MACHINE

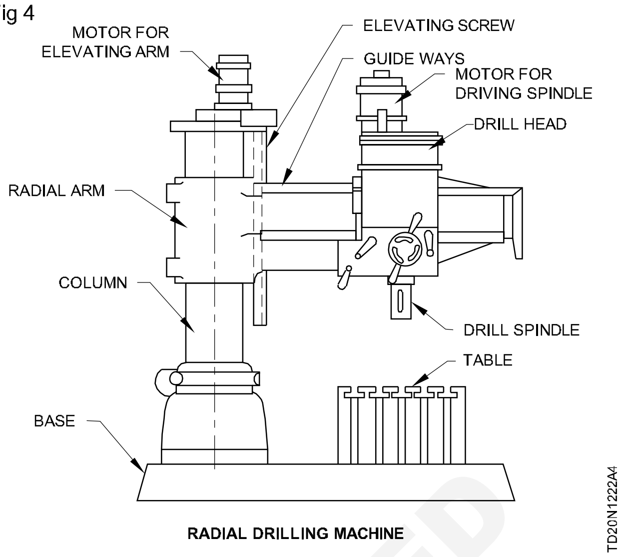
1. Bevel gear drive to spindle, 2 Spindle, 3 Overhead shaft, 4 Back stay, 5 Counter shaft cone pulley, 6. Fast and loose pulley, 7. Table elevating handle, 8. Foot pedal, 9.Base, 10. Rack 11. Table elevating clamp handle, 12. Table clamp, 13. Table, 14. Column, 15. Handwheel for quick hand feed, 16. Handwheel for sensitive hand feed.

Radial Drilling machine (Fig 4)

The radial drilling machine is intended for drilling medium to large and heavy workpieces. The machine consists of a heavy, round, vertical column mounted a radial arm which can be raised and lowered to accommodate work pieces of different heights. The arm may be swung around to any position over the work bed. The drill head containing mechanism for rotating and feeding the drill is mounted on a radial arm and can be moved horizontally on the guide - ways and clamped at any desired position. These three movements in a radial drilling machine when combined when combined together permit the drill to be located at any desired point on a large work piece, the position of the arm and the drill head is altered so that the drill spindle may be moved from one position to the other after drilling the hole without altering the setting of the work. This versality of the machine allows it to work on large work pieces. The work may be mounted on the table or when the work is very large it may be placed on the floor or in a pit.

1. Base, 2. column, 3. Radial arm , 4. Motor for elevating the arm, 5. Elevating screw, 6. Guide ways, 7. Motor for driving the drill spindle, 8. Drill head, 9. Drill spindle, 10. Table

Fig 4

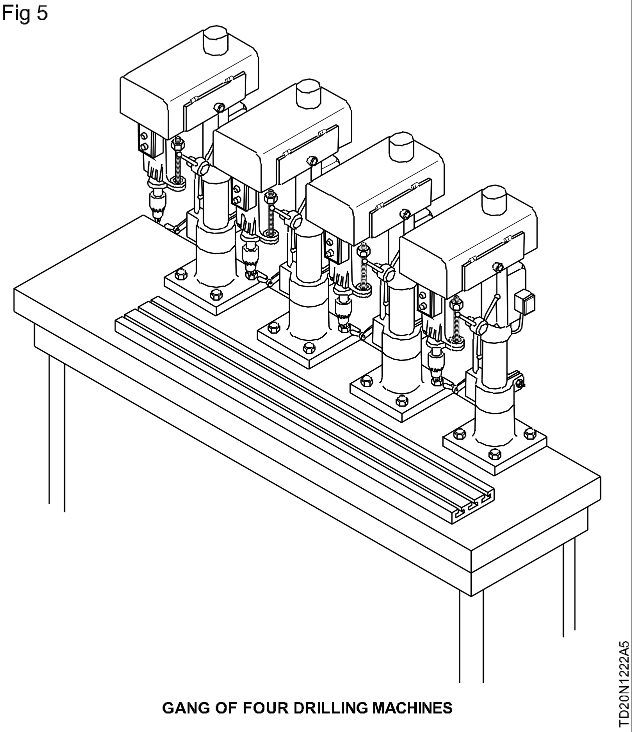


RADIAL DRILLING MACHINE

Gang Drilling Machine (Fig 5)

When a number of single spindle drilling machine columns are placed side by side on a common base and have a common work table, the machine is known as the gang drilling machine. In a gang drilling machine four to six spindles may be mounted side by side. In some machines the drill spindles are permanently spaced on the work table, and in others the position of the columns may be adjusted so that the space between the spindles may be varied. The speed and feed of the spindles are controlled independently. This type of machine is specially adapted for production work. A series of operations may be performed on the work by simply shifting the work from one position to the other on the work table. Each spindle may be set up properly with different tools for different operations.

Fig 5

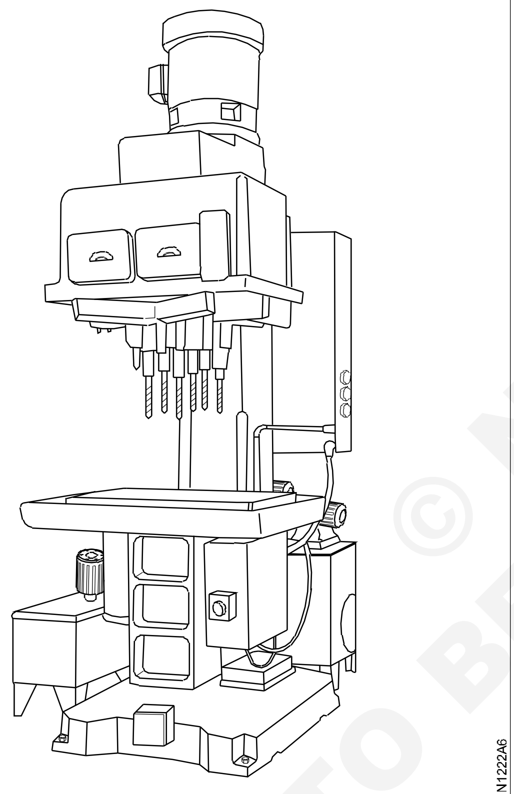


GANG OF FOUR DRILLING MACHINES

Multiple Spindle Machine (Fig 6)

The function of a multiple spindle drilling machine is to drill a number of holes in a piece of work simultaneously and to reproduce the same pattern of holes in a number of identical pieces in a mass production work. Such machine shave several spindles driven by a single motor and all the spindles holding drills are fed into the work simultaneously. Feeding motion is usually obtained by raising the work table. But the feeding motion may also be secured by lowering the drill heads. The spindles are so constructed that their centre distance may be adjusted in any position as required by various jobs within the capacity of the drill head. For this purpose, the drill spindles are connected to the main drive by universal joints. Drill jigs may be used for guiding the drills in mass production work.

Fig 6

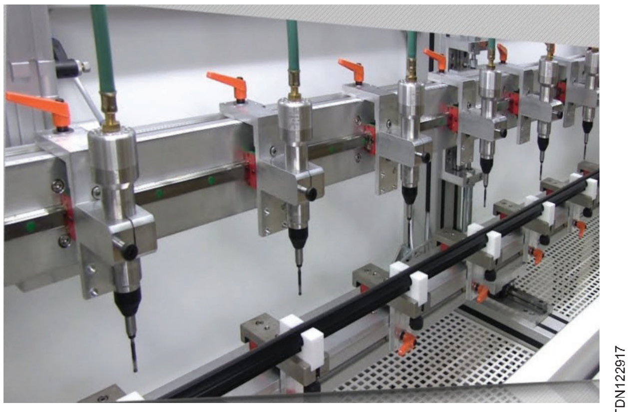


MULTI SPINDLE DRILLING MACHINE

Automatic Drilling machine (Fig 7)

Automatic drilling machine can perform a series of machining operations at successive units and transfer the work from one unit to the other automatically. Once the work is loaded at the first machine, the work will move from one machine to the other where different operations can be performed and the finished work comes out from the last unit without any manual handling. This type of machine is intended purely for production purposes and may be used for milling, honing and similar operations in addition to drilling and tapping.

Fig 7

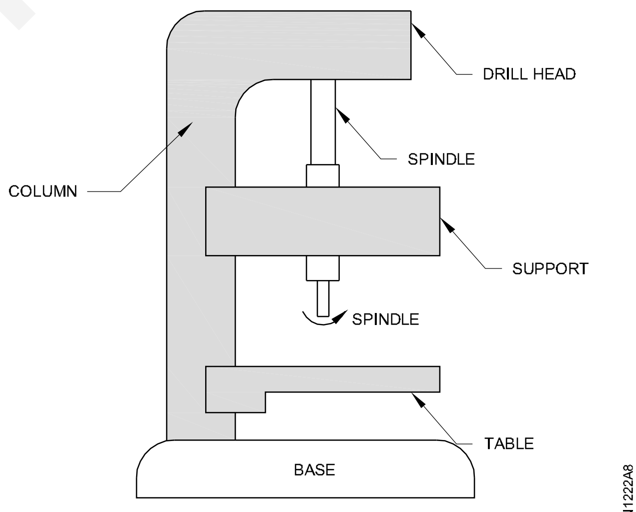


Deep Hole Drilling Machine (Fig 8)

Special machines and drills are required for drilling deep holes in rifle barrels, crank shafts, etc. The machine is operated at high speed and low feed. Sufficient quantity of lubricant is pumped to the cutting points for removal of chips and cooling the cutting edges of the drill. A long job is usually supported at several points to prevent any deflection. Usually the work piece and drill rotates in opposite direction while the drill is fed into the work.This helps in feeding the drill in a straight path.The machine may be horizontal or vertical type In some machines step feed is applied. The drill is withdrawn automatically each time when it penetrates into the work on a depth equal to its diameter. This process permits the chip to clear out from the work.

The work piece and drill rotates in opposite direction

Fig 8



DEEP HOLE DRILLING MACHINE

Construction of pillar type drilling machine

Objectives : At the end of this lesson you shall be able to
- **state the features of a pillar drilling machine**
- **state the parts and uses of pillar drilling machine**
- **explain the features and function of a pillar drilling machine.**

Upright Drilling Machine

The upright drilling machine is designed for handling medium sized workpieces. In Construction the machine is very similar to a sensitive drilling machine for having a vertical column mounted upon the base. But this is larger and heavier than a sensitive drilling machine and is supplied with power feed arrangement. In an upright drilling machine a large number of spindle speeds and feeds may be available for drilling different types of work. The table of the machine also have different types of adjustments. There are two general classes of upright drilling machine.

- Round column section or pillar driling machine.
- Box column section

Construction of Pillar drilling machine: The round column section upright drilling machine or pillar drilling machine consists of a round column that rises from the base which rests on the floor, an arm and a round table assembly, and a drll head assembly.

The arm and the table have three adjustments for locating workpieces under the spindle. The arm and the table may be moved up and down on the column for accommodating workpieces of different heights. The table and the arm may be moved in an arc upto 180º around the column and may be clamped at any position. This permits setting of the work below the spindle. Moreover, heavy and odd-size work may be supported directly on the base of the machine and drilled after the arm is swung out the way. The table may be rotated 360º about its own centre independent of the position of the arm for locating workpieces under the spindle.

The construction of the machine being not very rigid and the table being supported on a horizontal arm, this is particularly intended for lighter work. The maximum size of holes that the machine can drill is not more than 50mm

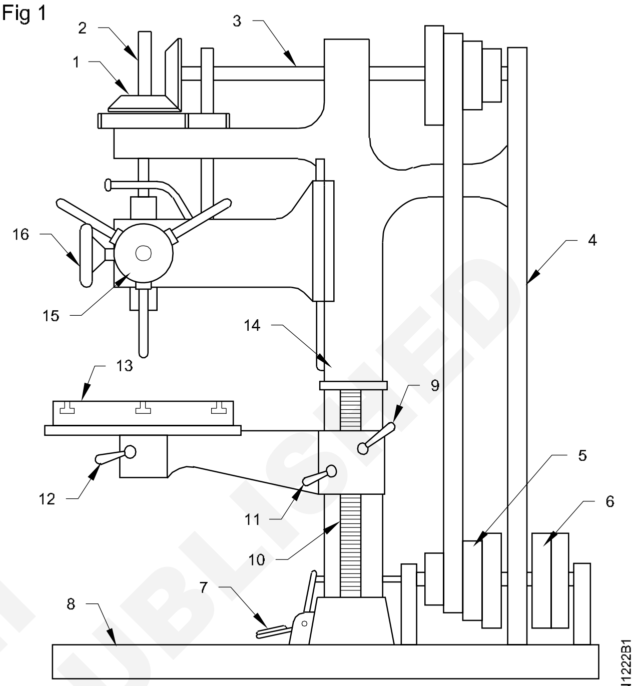
Parts of pillar drilling machine (Fig 1)

1 Bevel gear drive to spindle, 2. Spindle . 3. Overhead shaft, 4. Nack stay counter shaft cone pulley, 6. Fast and loose pulley, 7. Table elevating handle foot pedal, 9. Base, 10. Rack on column, 11. Table elevating clamp hadle Table clamp, 13. Table, 14. Column, 15. Handwheel for quick hand feed and Handwheel for sensitive hand feed.

The compund table for a pillar type drilling machine (Fig 2)

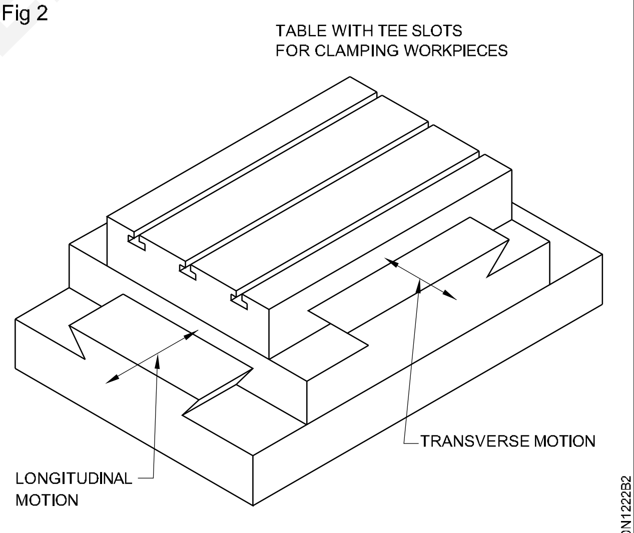
This is a development of the box-column type pillar drilling machine with a table mounted on the two slideways to give horizontal motions at 90° to each other, and controlled by the operating screws.

Fig 1



UPRIGHT PILLAR DRILLING MACHINE

Fig 2

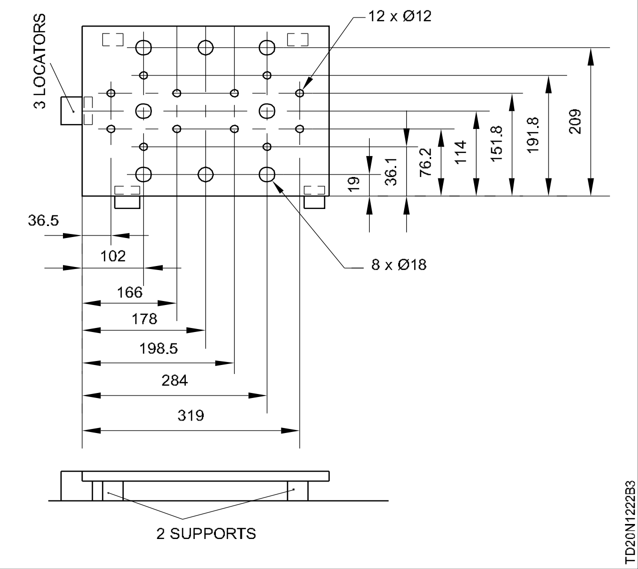


BASE REPLACES NORMAL MACHINE TABLE

A typical component for drilling on a compound table machine is shown in Fig 3.

The work is clamped to the compound table which can then be accurately adjusted to bring each hole under the spindle axis, in turn for drilling. If the work is located accurately on the table by locators in the 'T' slots, once the position to the first hole is fixed the remaining holes can be positioned by means of table movements without recourse to marking out.

Fig 3



Radial drilling machines

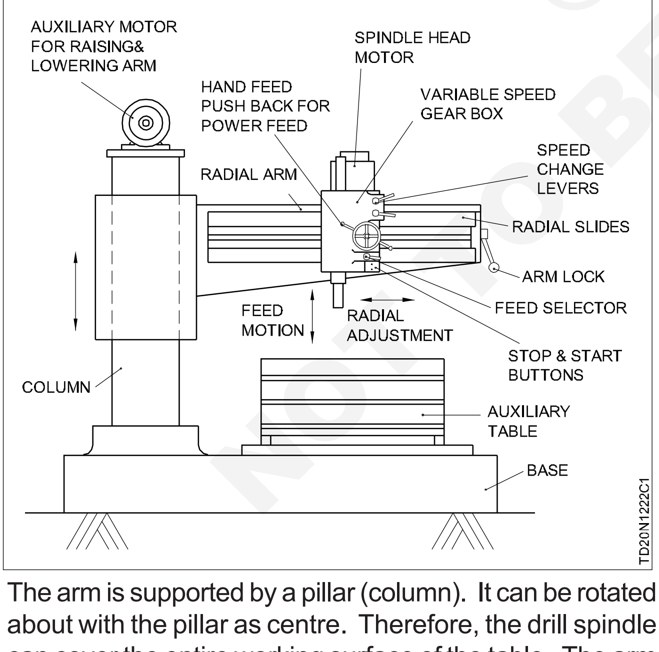
Objectives: At the end of this lesson you shall be able to
- **state the features of a radial drilling machine**
- **state the uses of a radial drilling machine.**

Features (Fig 1)

The radial drilling machine has a radial arm on which the spindle head is mounted.

The spindle head can be moved along the radial arm and can be locked in any position.

Fig 1

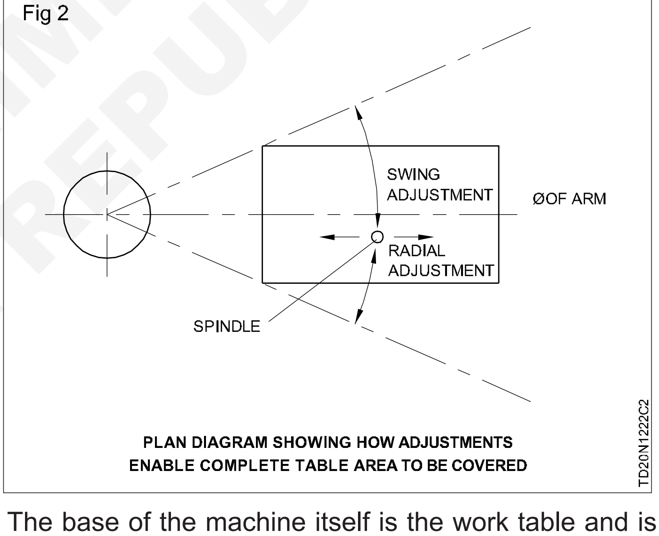


The arm is supported by a pillar (column). It can be rotated about with the pillar as centre. Therefore, the drill spindle can cover the entire working surface of the table. The arm can be lifted or lowered. (Fig 2)

The motor mounted on the spindle head rotates the spindle. The variable speed gearbox provides a large range of r.p.m.

The spindle can be rotated in both clockwise and anti-clockwise directions.

Fig 2



PLAN DIAGRAM SHOWING HOW ADJUSTMENTS ENABLE COMPLETE TABLE AREA TO BE COVERED

The base of the machine itself is the work table and is provided with 'T' slots for clamping large workpieces. An auxiliary table is usually provided to which smaller workpieces can be clamped, and in some cases, two such tables are used. One is placed on the machine while drilling is in progress and the other is on one side, with the previously finished work removed and new work positioned. When the work is completed the tables are interchanged by a hoist, the radial arm being swung clear for the purpose.

Radial drilling machines are used to drill

- large diameter holes
- multiple holes in one setting of the work
- heavy and large workpieces.
- angular holes on machines having tilting tables.

Drilling operation - Counter sinking

Objectives: At the end of this lesson you shall be able to
- **explain counter sinking**
- **list the purposes of counter sinking**
- **state the angles of counter sinking for different applications**
- **explain the different types of counter sinks and their applications.**

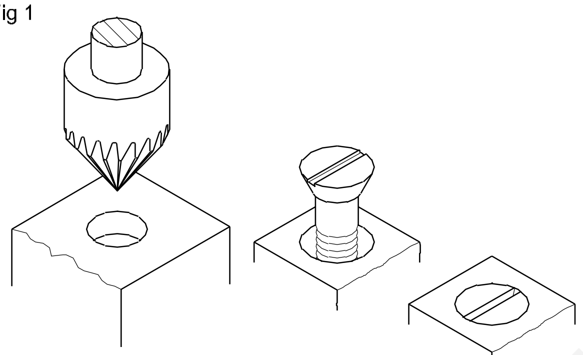
Counter sinking

Countersinking is an operation of beveling the end of a drilled hole. The tool used is called a countersink.

Counter sinking is carried out for the following purposes.

- To provide a recess for the head of a counter sink screw, so that it is flush with the surface after fixing. (Fig 1)

Fig 1



TD20N1222D1

- To deburr a hole after drilling
- To accommodate counter sink rivet heads.
- To chamfer the ends of holes for thread cutting and other machining processes.

Angles for counter sinking

Counter sinks are available in different angles and for differeent uses.

75° Counter sink riveting.

80° Counter sink self-tapping screws.

90° Counter sink head screws and for deburring.

120° Chamfering ends of holes to be threaded or other machining processes.

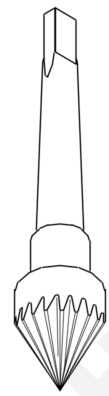
Counter sinks

Counter sinks of different types are available.

The commonly used counter sinks have multiple cutting edges and are available in taper shank (Fig 2) and straight shank.

For counter sinking small diameter holes special counter sinks with one or two flutes are available. This will reduce the vibration while cutting.

Fig 2

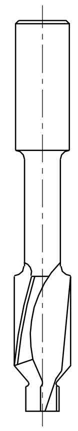


TD20N1222D2

Counter sinks with pilot (Fig 3)

For precision counter sinking, needed for machine tool assembling and after machining process, counter sinks with pilots are used.

Fig 3



TD20N1222D3

They are particularly useful for heavy duty work.

The pilot is provided at the end for guiding the counter sink to the hole.

Counter sinks with pilots are available with interchange able and solid pilots.

Counter sink hole sizes

The counter sink holes according to Indian Standards IS 3406 (Part 1) 1986 are of four types.

Type A

Type B

Type C

Type E

These types are used for different purposes.

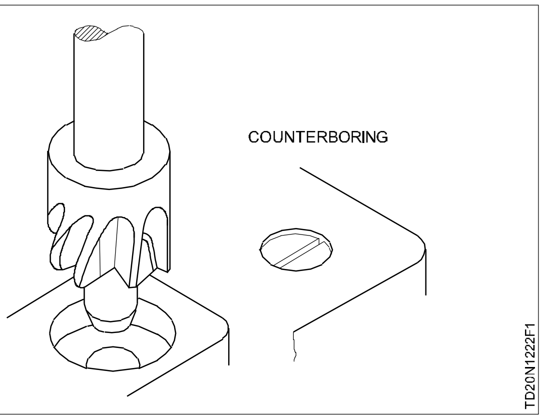
Counter boring and spot facing

Objectives: At the end of this lesson you shall be able to
- **explain the difference between counter boring and spot facing**
- **state the various types of counter bores and their uses**
- **state the correct counter bore sizes for different holes.**

Counter boring

Counter boring is an operation of enlarging a hole to a given depth, to house heads of socket heads or cap screws with the help of a counter bore tool. (Fig 1)

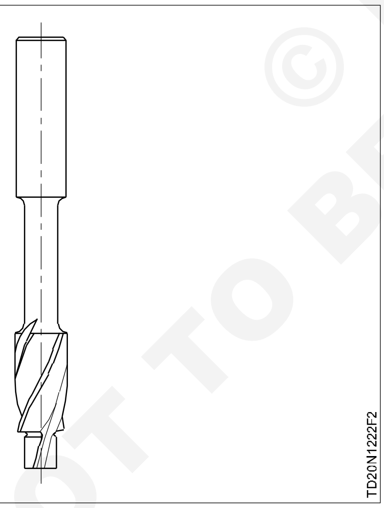
Fig 1



Counter bore (tool)

The tool used for counter boring is called a counter bore. (Fig 2) Counter bores will have two or more cutting edges. At the cutting end, a pilot is provided to guide the tool concentric to the previously drilled hole.

Fig 2



The pilot also helps to avoid chattering while counter boring. (Fig 3)

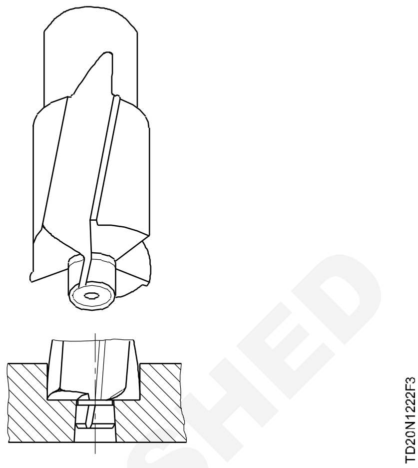
Counter bores are available with solid pilots or with interchangeable pilots. The inter-changeable pilots provide flexibility of counter boring on different diameters of holes.

Counter bores: sizes and specification

Counter bore sizes are standardized for each diameter of screws as per BIS.

There are two main types of counterbores. Type H and Type K. These types are used for different purposes.

Fig 3



Spot facing

Spot facing is a machining operation for producing a flat seat for a bolt head, washer or nut at the opening of the drilled hole. The tool is called a spot facer or a spot facing tool. Spot facing is similar to counterboring, except that the hole is shallower. Tools that are used for counterboring can be used for spot facing as well. (Fig 4) Spot facing is also done by fly cutters by end cutting action. The cutter blade is inserted in the slot of the holder, which can be mounted on to the spindle. (Fig 5)

Fig 4

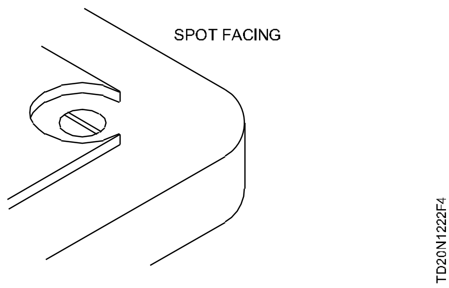
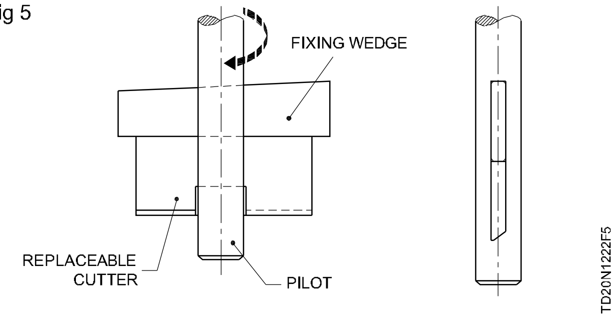


Fig 5



Cutting speed and r.p.m.

Objectives: At the end of this lesson you shall be able to
- **define cutting speed**
- **state the factors for determining the cutting speed**
- **explain difference between cutting speed and r.p.m.**
- **state the r.p.m./spindle speed**
- **explain the r.p.m. for drill sizes from tables.**

For a drill to give a satisfactory performance, it must operate at the correct cutting speed and feed.

Cutting speed is the speed at which the cutting edge passes over the material while cutting, and is expressed in metres per minute.

Cutting speed is also sometimes stated as surface speed or peripheral speed.

The selection of the recommended cutting speed for drilling depends on the materials to be drilled, and the tool material.

Tool manufacturers usually provide a table of cutting speeds required for different materials.

The recommended cutting speeds for different materials are given in the table. Based on the cutting speed recommended, the r.p.m. at which a drill has to be driven, is determined.

Materials being drilled for HSS	Cutting speed(m/min)
Aluminium	70 - 100
Brass	35 - 50
Bronze(phosphor)	20 - 35
Cast iron (grey)	25 - 40
Copper	35 - 45
Steel (medium carbon/mild steel)	20 - 30
Steel (alloy,high tensile)	5 - 8
Thermosetting plastic (low speed due to abrasive properties)	20 - 30

Calculating r.p.m.

$$v = \frac{n \times d \times \pi}{1000} \text{ m/min}$$

$$n = \frac{v \times 1000}{d \times \pi}$$

n - r.p.m.

v - cutting speed in m/min.

d - diameter of the drill in mm

p = 3.14.

Examples

Calculate the r.p.m. for a high speed steel drill Ð24 to cut mild steel.

The cutting speed for MS is taken as 30 m/min. from the table.

$$n = \frac{1000 \times 30}{3.14 \times 24} = 398 \text{ r.p.m.}$$

It is always preferable to set the spindle speed to the nearest available lower range. The selected spindle speed is 300 r.p.m.

The r.p.m. will differ according to the diameter of the drills. The cutting speed being the same, larger diameter drills will have lesser r.p.m. and smaller diameter drills will have higher r.p.m.

The recommended cutting speeds are achieved only by actual experiments.

Feed in drilling

Objectives: At the end of this lesson you shall be able to
- **state what is meant by feed**
- **state the factors that contribute to an efficient feed rate.**

Feed is the distance (X) a drill advances into the work in one complete rotation. (Fig 1)

Feed is expressed in hundredths of a millimetre.

Example - 0.040 mm

The rate of feed is dependent upon a number of factors.

Finish required

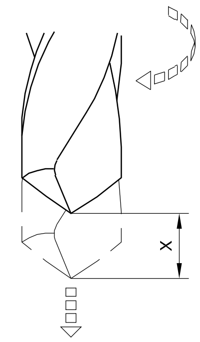
Type of drill (drill material)

Material to be drilled

the feed rate. If these are not to the required standard, the feed rate will have to be decreased.

It is not possible to suggest a particular feed rateif all the factors are not taken into account.

Fig 1



The table for the feed rate given here is based on the average feed values suggested by the different manufacturers of drills. (Table 1)

Table 1

Drill diameter (mm) H.S.S.	Rate of feed (mm/rev)
1.0 - 2.5	0.040 - 0.060
2.6 - 4.5	0.050 - 0.100
4.6 - 6.0	0.075 - 0.150
6.1 - 9.0	0.100 - 0.200
9.1 -12.0	0.150 - 0.250
12.1 - 15.1	0.200 - 0.300
15.1 - 18.0	0.230 - 0.330
18.1 - 21.0	0.260 - 0.360
21.1 - 25.0	0.280 - 0.380

Too coarse a feed may result in damage to the cutting edges or breakage of the drill.

Too slow a rate of feed will not bring improvement in surface finish but may cause excessive wear of the tool point, and lead to chattering of the drill.

For optimum results in the feed rate while drilling, it is necessary to ensure the drill cutting edges are sharp. Use the correct type of cutting fluid.

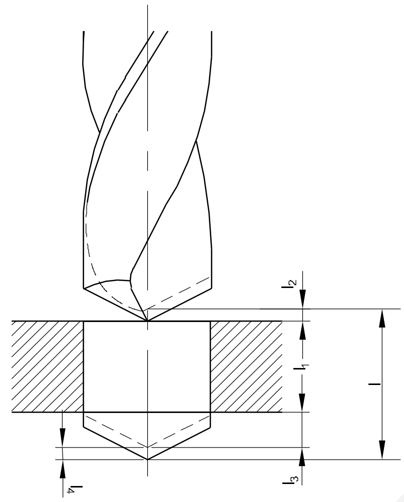
Cutting speed, feed, drilling time calculation

Machining Time in drilling (Fig 2)

Machining time in drilling is determined by the formula:

$$T = \frac{L}{n \times s_r} \text{ min.}$$

Fig 2



Where, n = r.p.m. of the drill

Sr = Feed per revolution of the drill in mm

L = Length of travel of the drill in mm

and T = Machining time in min.

$L = l_1 + l_2 + l_3 + l_4$

Where, l_1 = length of the workpiece

l_2 = approach of the drill,

l_3 = length of the drill point (0.29d)

l_4 = overtravel

Example:

Calculate the drilling time to drill 12mm dia hole in a plate of thickness 62mm, cutting speed 30m/min and federate is 0.05mm/ rev

Formulae for drilling time $= T = \frac{L}{n \times sr}$

$L = l_1 + l_2 + l_3 + l_4$

= 62 + 5 + 4 + 2

= 73mm

$$n = \frac{1000 \times 30}{3.143 \times 12} = 795$$

sr - 0.05 mm

$$T = \frac{73}{0.5 \times 795} = 1.84 \text{ minutes}$$

= 1 minute 50 secs

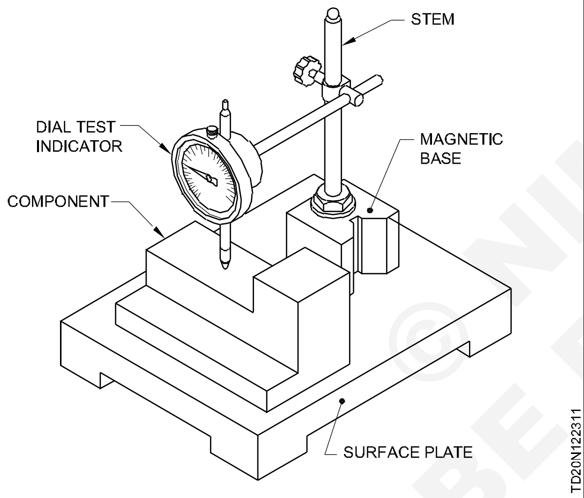
Factors like rigidity of the machine, holding of the workpiece and the drill, will also have to be considered while determining

Dial test indicators

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

Dial test indicators are instruments of high prescision, used for comparing and determining the variation in the sizes of a component. (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 pointer on a graduated dial. This direct reading of the deviations gives an accurate picture of the conditions of the parts being tested.

Fig 1



Principle of working

The magnification of the small movement of the plunger or stylus is converted into a rotary motion of the pointer on a circular scale. (Fig 2)

Types

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

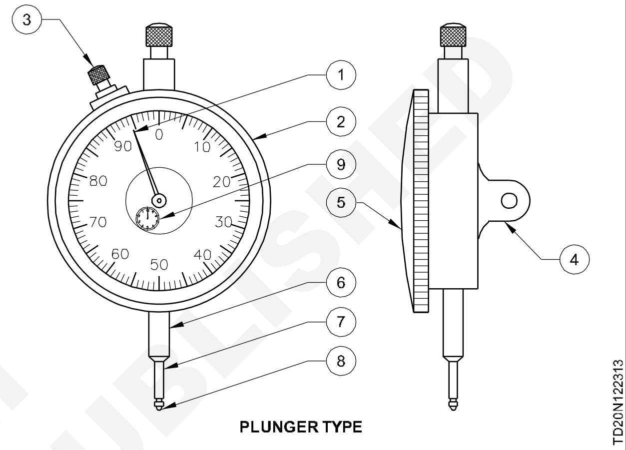
- **plunger type (Fig 3)**
- **lever type. (Fig 4)**

The plunger type dial test indicator

The external parts and features of a dial test indicator areas shown in figure 3.

1 Pointer	2 Rotatable bezel
3 Tezel clamp	4 Back lug
5 Transparent dial cover	6 Stem
7 Plunger	8 Anvil

Fig 3

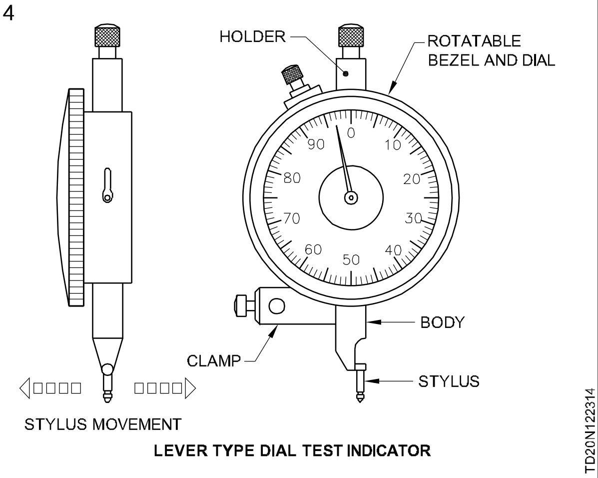


9 Revolution counter

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

The lever type dial test indicator (Fig 4)

Fig 4



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

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 2

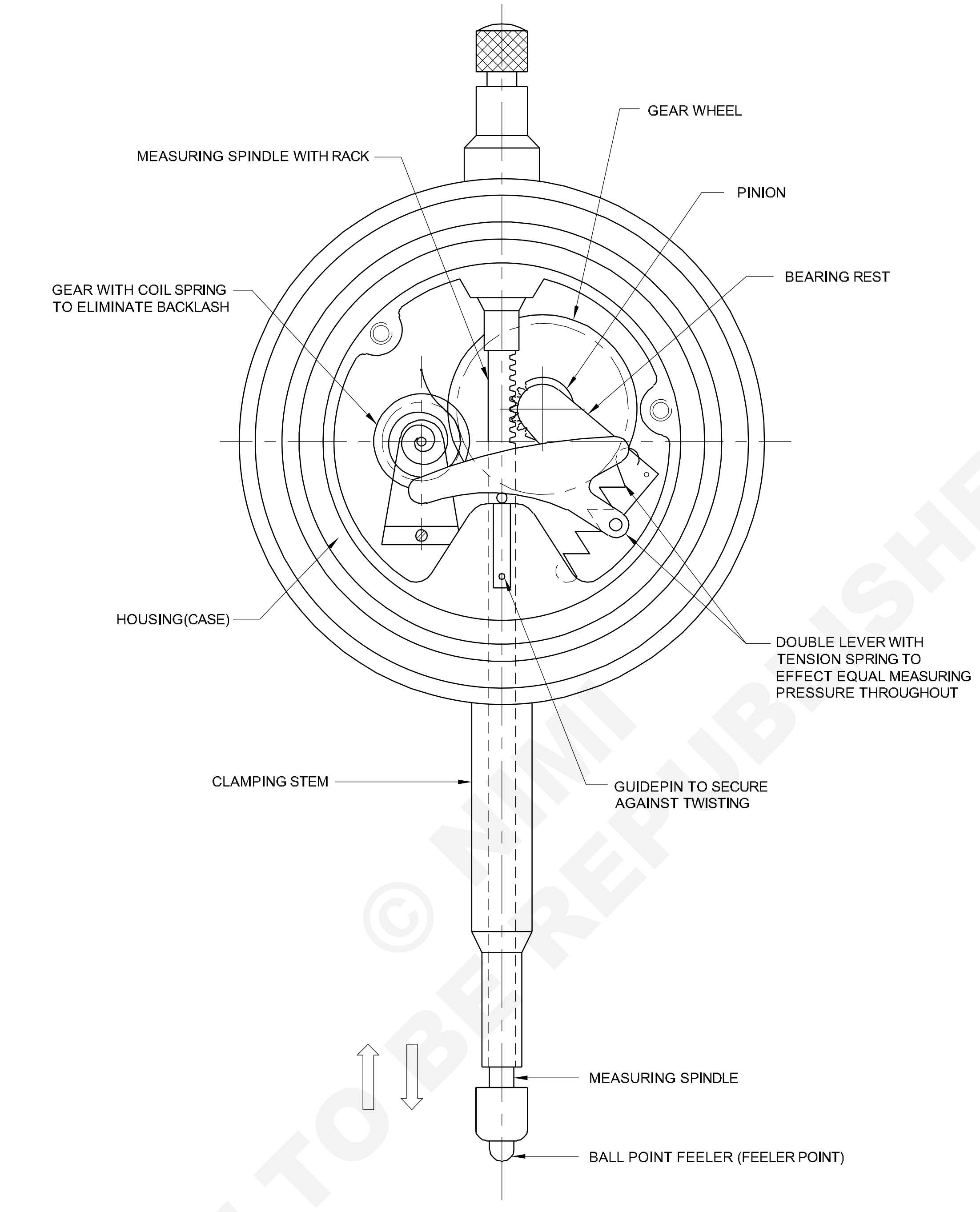
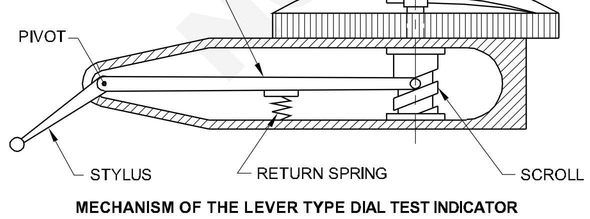
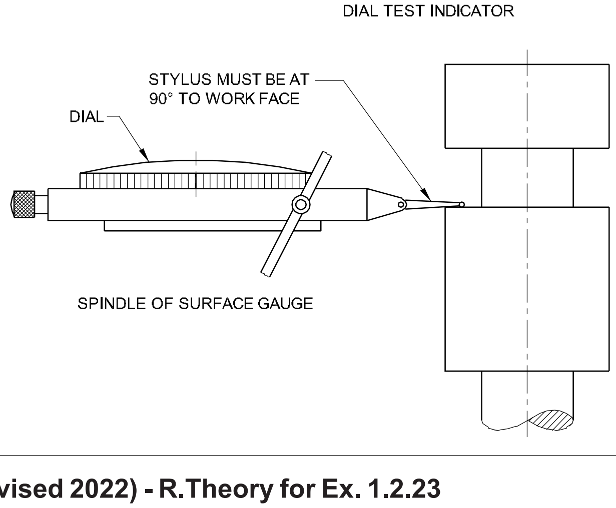


Fig 5



MECHANISM OF THE LEVER TYPE DIAL TEST INDICATOR

Fig 6



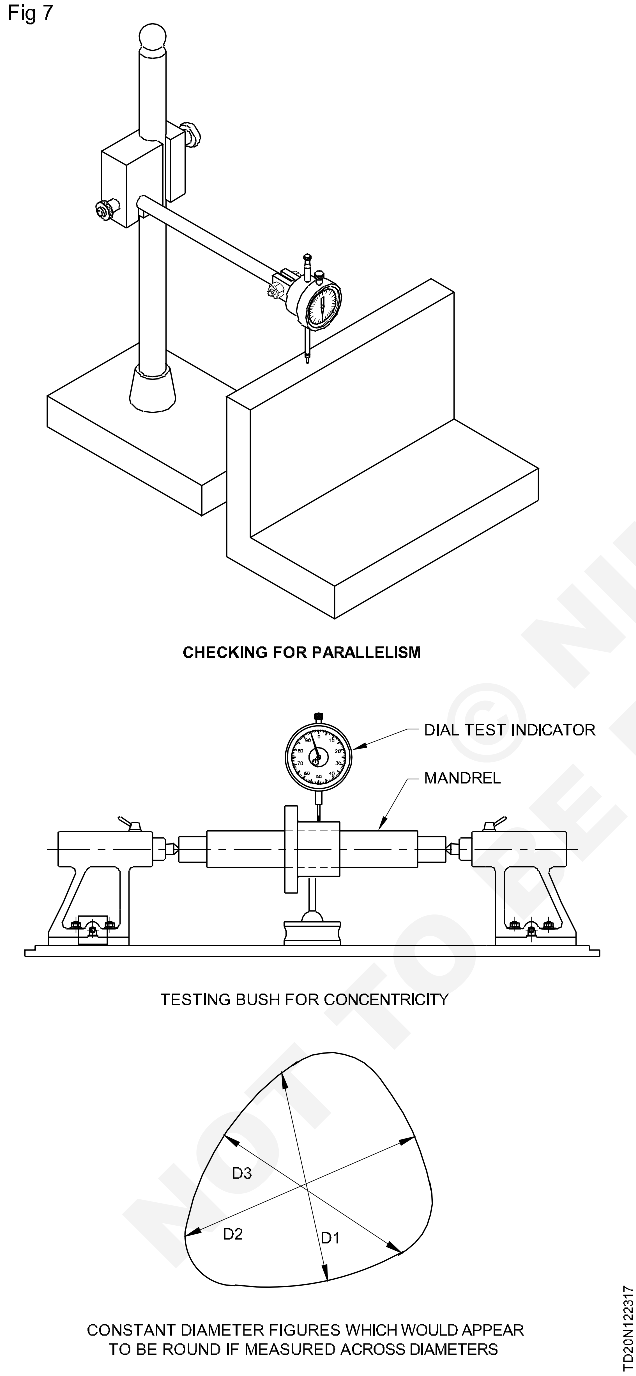
Important features of dial test indicators

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

Many dial test indicators read plus in a clockwise direction from zero, and minus in the anticlockwise directions so as to give plus and minus indications.

Uses (Figure 7 shows a few applications.)

Fig 7



- To compare the dimensions of a workpiece against a known standard, eg.slip gauges.
- To check plane surfaces for parallelism and flatness.
- To check parallelism of shafts and bars.
- To check concentricity of holes and shafts.

Indicator stands

Dial test indicators are used in conjunction with stands for holding them so that the stand itself may be placed on a datum surface or machine tools. (Fig 8)

The different types of stands are (Fig 9)

- magnetic stand with universal clamp
- magnetic stand with flexible post
- general purpose holder with cast iron base.

The arrows indicate the provisions in the clamps for insertion of the dial test indicator.

Fig 8

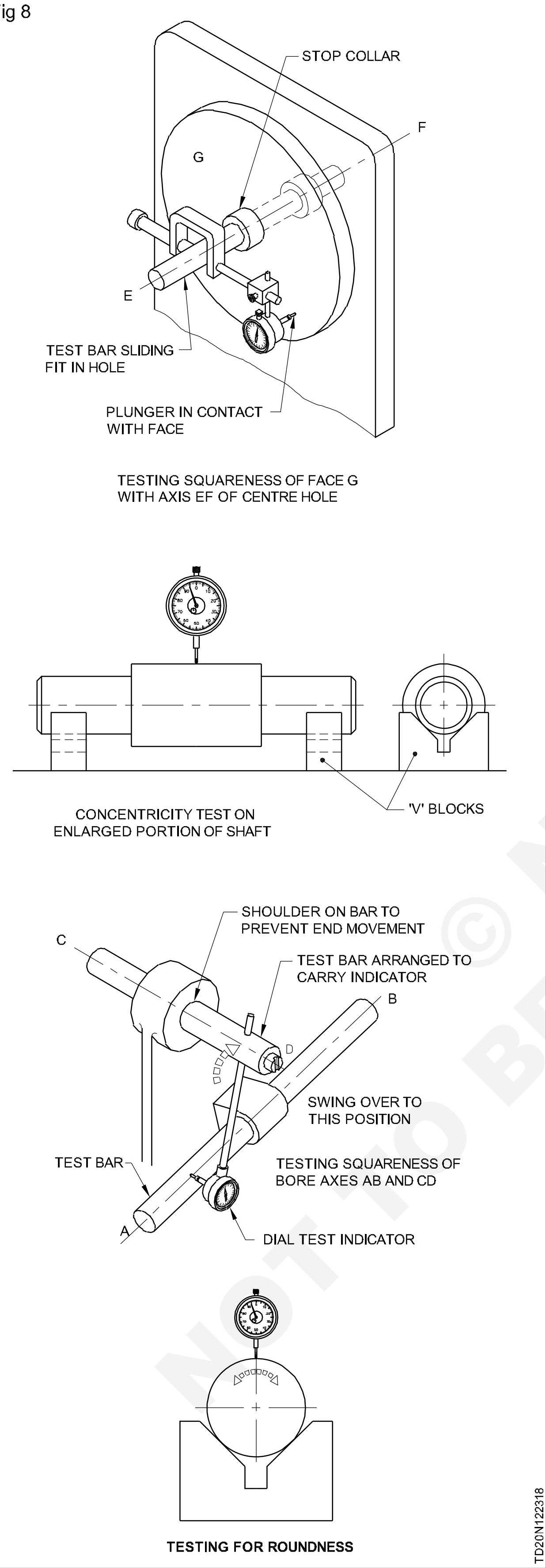
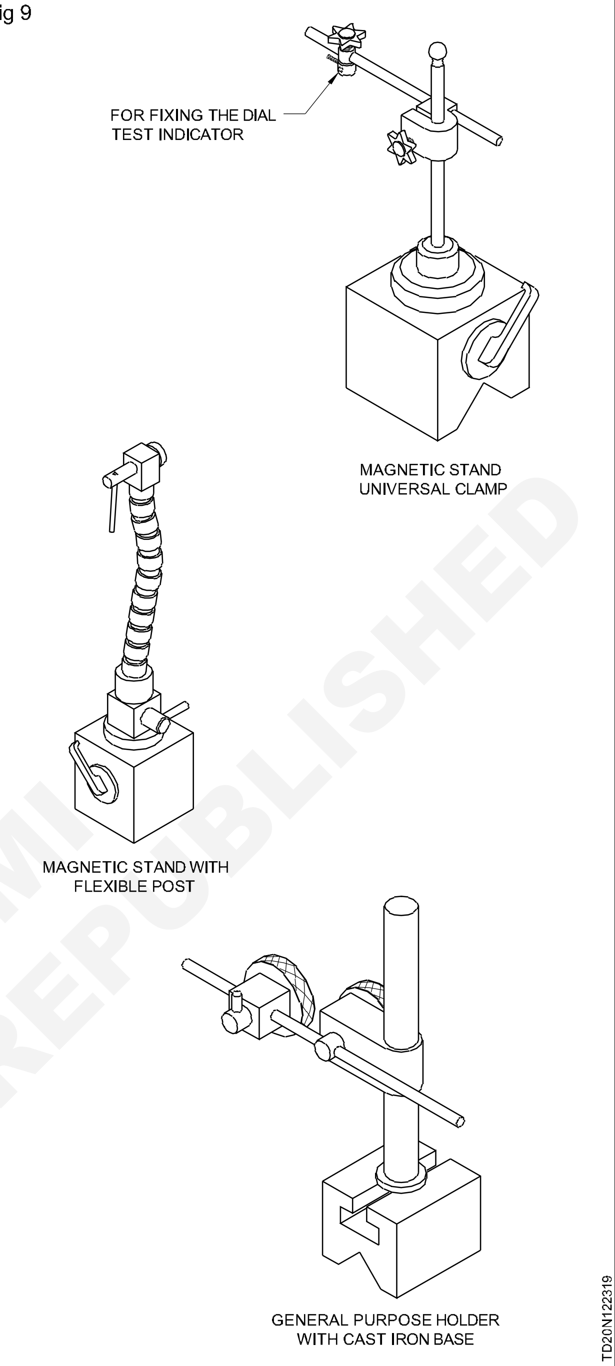

STOP COLLAR
F
G
E
TEST BAR SLIDING FIT IN HOLE
PLUNGER IN CONTACT WITH FACE
TESTING SQUARENESS OF FACE G WITH AXIS EF OF CENTRE HOLE
'V' BLOCKS
CONCENTRICITY TEST ON ENLARGED PORTION OF SHAFT
SHOULDER ON BAR TO PREVENT END MOVEMENT
C
TEST BAR ARRANGED TO CARRY INDICATOR
B
D
SWING OVER TO THIS POSITION
TEST BAR
TESTING SQUARENESS OF BORE AXES AB AND CD
A
DIAL TEST INDICATOR
TESTING FOR ROUNDNESS
TD20N122318

Fig 9


FOR FIXING THE DIAL TEST INDICATOR
MAGNETIC STAND UNIVERSAL CLAMP
MAGNETIC STAND WITH FLEXIBLE POST
GENERAL PURPOSE HOLDER WITH CAST IRON BASE
TD20N122319

Mass production and interchangeable manufacture

Objectives: At the end of this lesson you shall be able to
- **state the advantages and disadvantages of mass production**
- **explain the meaning of the term, 'interchangeability'**
- **state the necessity for the limit system**
- **state the different standards of system of limits and fits.**

Mass production

Mass production means production of a unit, component or part in large numbers.

Advantages of mass production

Time for the manufacture of components is reduced.

The cost of work 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.

Manufacturing and measuring time is saved.

Disadvantages of mass production

Special purpose machines are necessary.

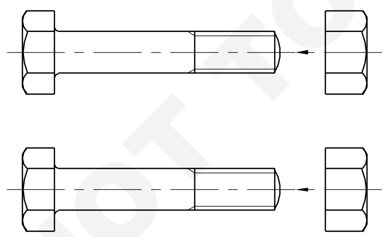
Jigs and fixtures are needed.

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.

Fig 1



SELECTIVE ASSEMBLY

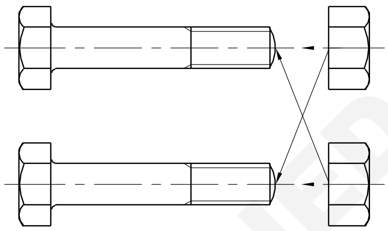
TD20N122411

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)

Non-selective assembly provides interchangeability between the components.

Fig 2



NON-SELECTIVE ASSEMBLY INTERCHANGABILITY

TD20N122412

In modern engineering production, i.e. mass production, 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 production 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.

Various standard systems of limits and fits are followed by different countries based on the ISO (International Standards Organisation) specifications.

The system of limits and fits followed in our country is that which is stipulated by the BIS. (Bureau of Indian Standards)

Other systems of limits and fits

- British Standard System (BSS)
- German Standard (DIN)

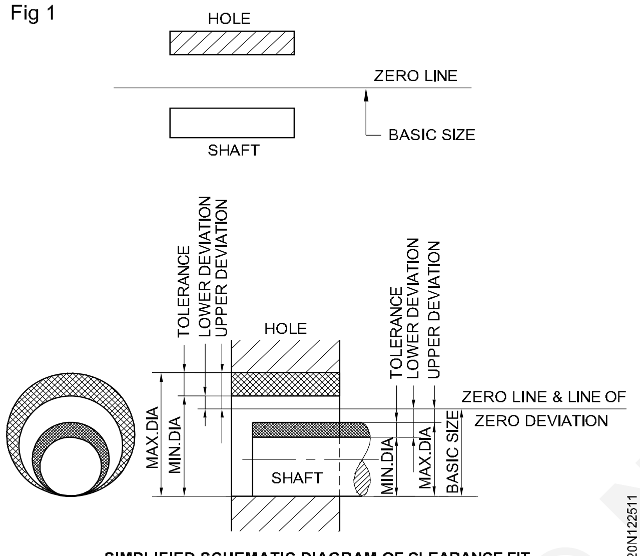
The Indian standard system of limits and fits - terminology

Objectives: At the end of this lesson you shall be able to
- **state the terms used under the BIS system of limits and fits**
- **define each term under the BIS system of limits and fits.**

Size: It is a number expressed in a particular unit in the measurement of length.

Basic size: It is the size based on which the dimensional deviations are given. (Fig 1)

Fig 1



SIMPLIFIED SCHEMATIC DIAGRAM OF CLEARANCE FIT

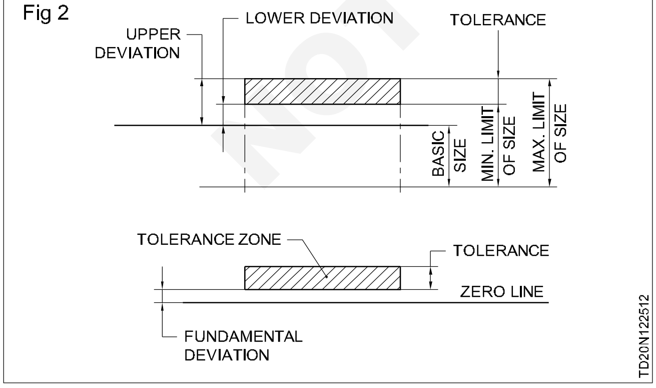
Actual size

It is the size of the component by actual measurement after it is manufactured. It should lie between the two limits of size if the component is to be accepted.

Limits of size

These are the extreme permissible sizes within which the operator is expected to make the component. (Maximum and minimum limits) (Fig 2)

Fig 2

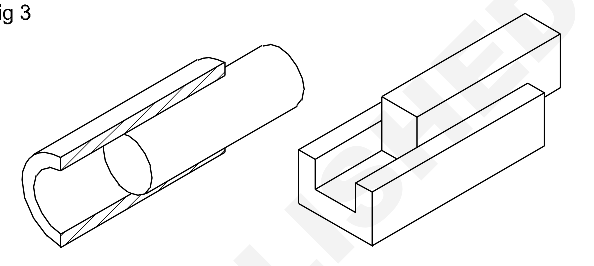


Maximum limit of size: It is the greater of the two limits of sizes. (Fig 2) (Table 1)

Minimum limit of size: It is the smaller of the two limits of size. (Fig 2) (Table 1)

Hole: In the BIS system of limits and fits, all internal features of a component including those which are not cylindrical are designated as hole. (Fig 3)

Fig 3



Shaft: In the BIS system of limits and fits, all external features of a component including those which are not cylindrical are designated as shaft. (Fig 3)

Deviation: It is the algebraic difference between a size and its corresponding basic size. It may be positive, negative or zero. (Fig 2)

Upper deviation: It is the algebraic difference between the maximum limit of size and its corresponding basic size. (Fig 2) (Table 1)

Lower deviation

It is the algebraic difference between the minimum limit of size and its corresponding basic size. (Fig 2) (Table 1)

Upper deviation is the deviation which gives the maximum limit of size. Lower deviation is the deviation which gives the minimum limit of size.

Actual deviation: It is the algebraic difference between the actual size and its corresponding basic size. (Fig 2)

Tolerance: It is the difference between the maximum limit of size and the minimum limit of size. It is always positive and is expressed only as a number without a sign. (Fig 2)

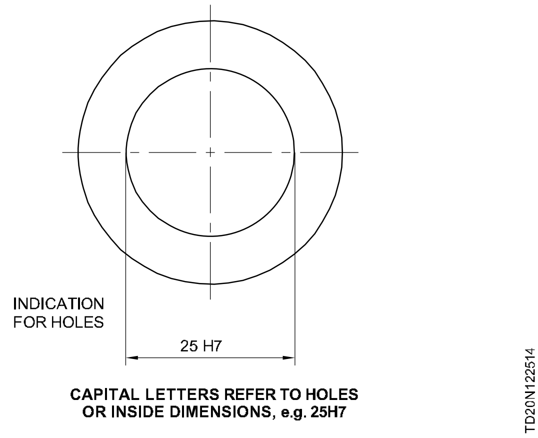
Zero line

In the graphical representation of the above terms, the zero line represents the basic size. This line is also called the line of zero deviation. (Figs 1 and 2)

Fundamental deviation

There are 25 fundamental deviations in the BIS system represented by letter symbols (capital letters for holes and small letters for shafts), i.e. for holes - ABCD .... Z excluding I,L,O,Q and W. (Fig 4)

Fig 4

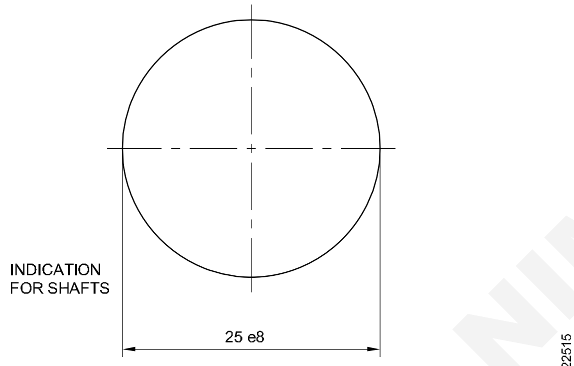


CAPITAL LETTERS REFER TO HOLES OR INSIDE DIMENSIONS, e.g. 25H7

In addition to the above, four sets of letters JS,ZA,ZB and ZC are included.

For shafts, the same 25 letter symbols but in small letters are used. (Fig 5)

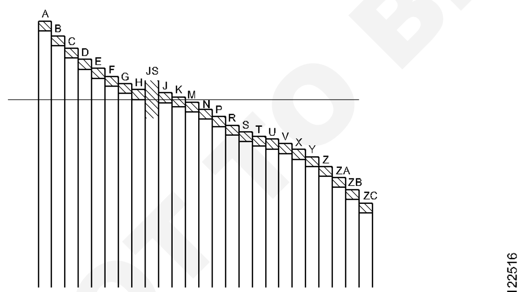
Fig 5



SMALL LETTERS REFER TO SHAFTS OR OUTSIDE DIMENSIONS, e.g. 25 e8

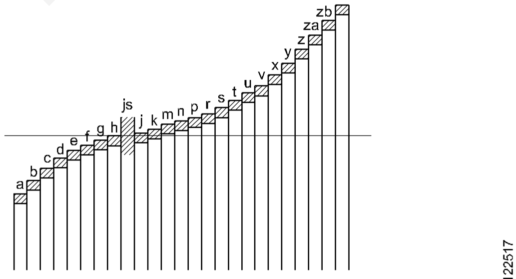
The position of tolerance zone with respect to the zero line is shown in Figs 6 and 7.

Fig 6



CAPITAL LETTERS INDICATE FUNDAMENTAL DEVIATION FOR HOLES

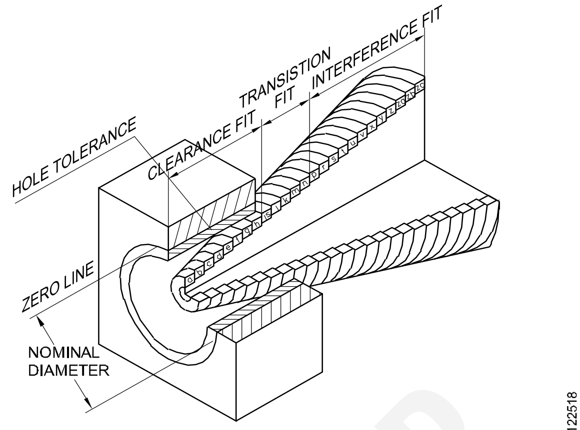
Fig 7



SMALL LETTERS INDICATE FUNDAMENTAL DEVIATION FOR SHAFTS

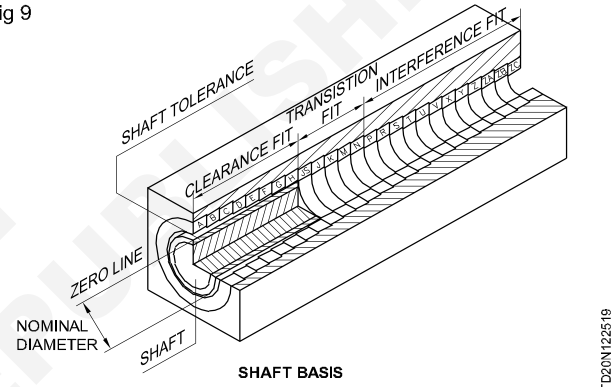
The fundamental deviations are for achieving the different classes of fits. (Figs 8 and 9)

Fig 8



HOLE BASIS

Fig 9

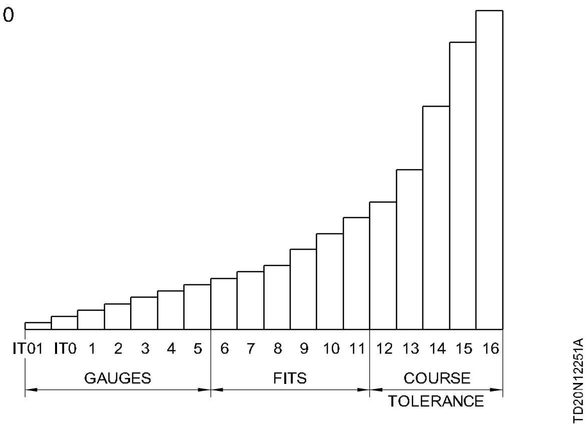


SHAFT BASIS

Fundamental tolerance

This is also called 'grade of tolerance'. In the B.I.S. system, there are 18 grades of tolerances represented by number symbols both for hole and shaft, denoted as IT01, IT0, IT1, IT2 .... IT16 (Fig 10)

Fig 10



A higher number gives a larger tolerance.

Grade of tolerance refers to the accuracy of manufacture.

In a standard chart, the upper and lower deviations for each combination of fundamental deviation and fundamental tolerance are indicated for sizes ranging up to 500 mm. (Refer to IS 919.)

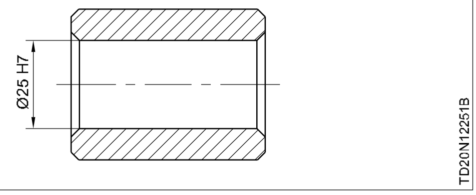
Tolerance size

This includes the basic size, the fundamental deviation and the grade of tolerance.

Examples

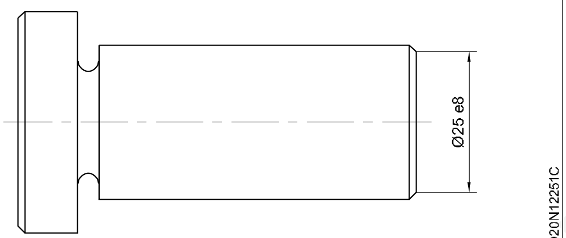
25 H7 - is the tolerance size of a hole whose basic size is 25. The fundamental deviation is represented by the letter symbol H and the grade of tolerance is represented by the number symbol 7. (Fig 11)

Fig 11



25 e8 - is the tolerance size of a shaft whose basic size is 25. The fundamental deviation is represented by the letter symbol and the grade of tolerance is represented by the number 8. (Fig 12)

Fig 12



A very wide range of selection can be made by the combination of the 25 fundamental deviations and 18 grades of tolerances.

Example

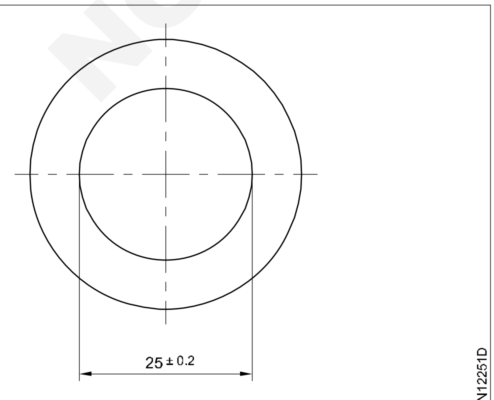
In figure 13, a hole is shown as 25 ± 0.2 which means that 25 mm is the basic dimension and ± 0.2 is the deviation.

As pointed out earlier, the permissible variation from the basic dimension is called ‘DEVIATION’.

The deviation is mostly given on the drawing with dimensions.

In the example, 25 ± 0.2, ± 0.2 is the deviation of the hole of 25 mm diameter. (Fig 13) This means that the hole is of acceptable size if its dimension is between

Fig 13



TOLERANCED SIZES OF HOLES

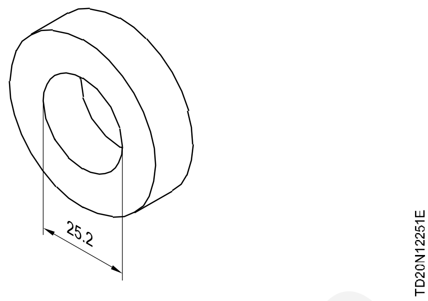
25 + 0.2 = 25.2 mm

or

25 - 0.2 = 24.8 mm.

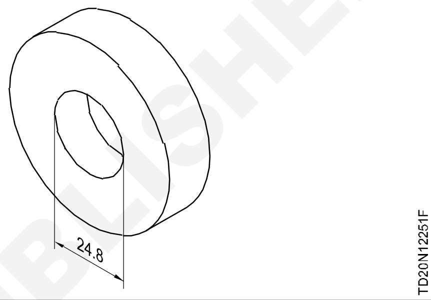
25.2 mm is the maximum limit. (Fig 14)

Fig 14



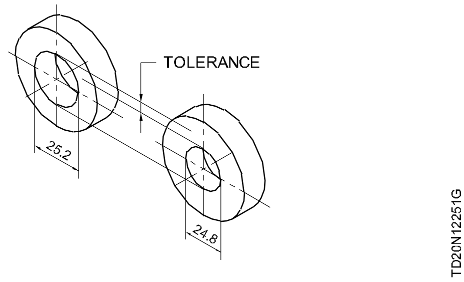
24.8 mm is the minimum limit. (Fig 15)

Fig 15



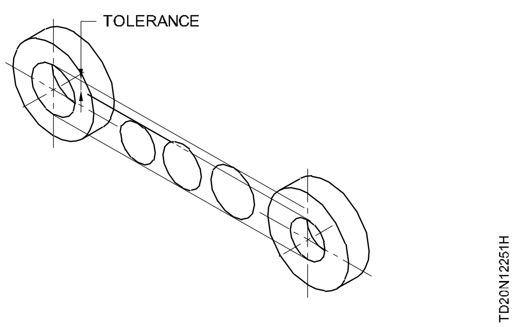
The difference between the maximum and minimum limits is the TOLERANCE. Tolerance here is 0.4 mm. (Fig 16)

Fig 16



All dimensions of the hole within the tolerance zone are of an acceptable size as shown in Fig 17.

Fig 17



As per IS 696, while dimensioning the components as a drawing convention, the deviations are expressed as tolerances.

Different Standard System of fits & limits (As per the Indian standard)

Objectives: At the end of this lesson you shall be able to
- **define 'fit' as per the Indian Standard**
- **list out the terms used in limits and fits as per the Indian Standard**
- **state examples for each class of fit**
- **state the graphical representation of different classes of fits.**

Fit

It is the relationship that exists between two mating parts, a hole and a shaft, with respect to their dimensional differences before assembly.

Expression of a fit

A fit is expressed by writing the basic size of the fit first, (the basic size which is common to both the hole and the shaft) followed by the symbol for the hole, and the symbol for the shaft.

Example

30 H7/g6 or 30 H7 – g6 or 30

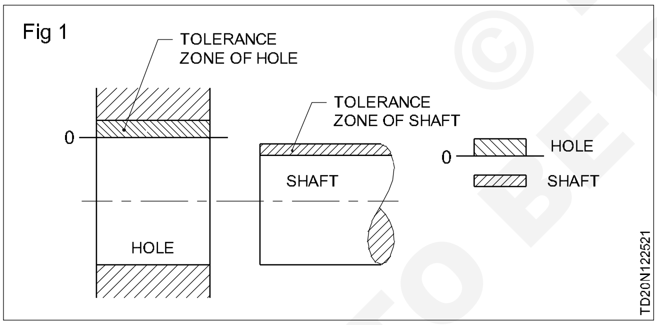
Clearance

In a fit the clearance is the difference between the size of the hole and the size of the shaft, when the hole is bigger than the shaft.

Clearance fit

It is a fit which always provides clearance. Here the tolerance zone of the hole will be above the tolerance zone of the shaft. (Fig 1)

Fig 1



Example

20 H7/g6

With the fit given, we can find the deviations from the chart.

For a hole 20 H7 we find in Table 1, +21.

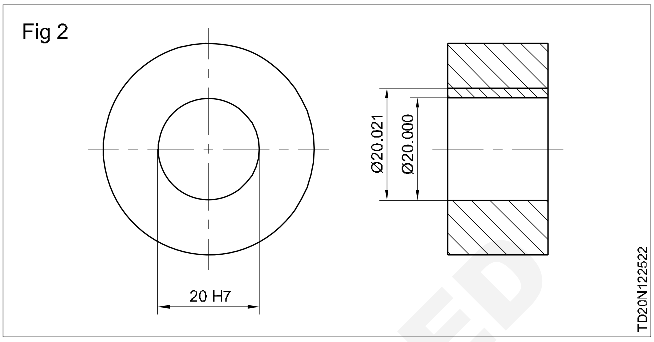
These numbers indicate the deviations in microns. (1 micron = 0.001 mm)

The limits of the hole are 20 + 0.021 = 20.021 mm and 20 + 0 = 20.000 mm. (Fig 2)

For a shaft 20 g6 we find in the Table – 7

– 20.

Fig 2

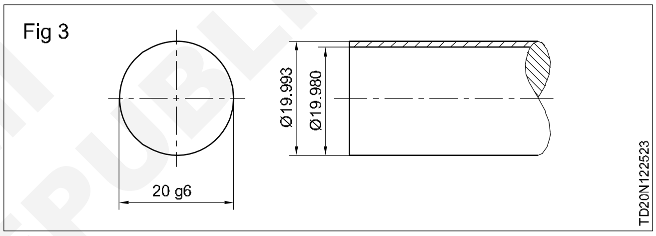


So the limits of the shaft are

20 – 0.007 = 19.993 mm

and 20 – 0.020 = 19.980 mm. (Fig 3)

Fig 3



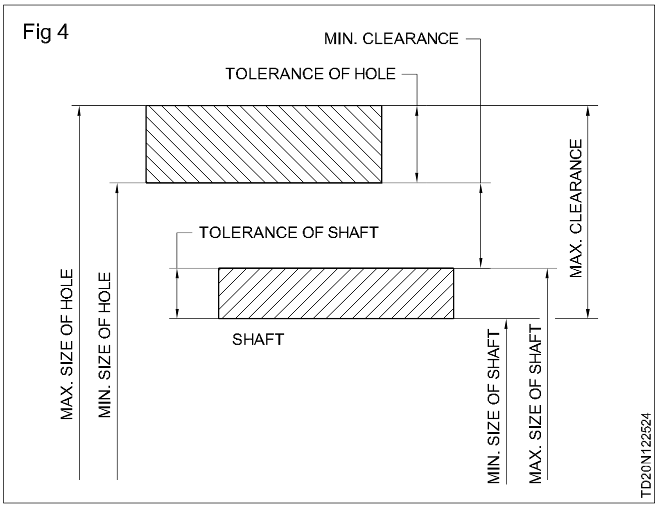
Maximum clearance

In a clearance fit or transition fit, the maximum clearance is the difference between the maximum size hole and the minimum size shaft. (Fig 4)

Minimum clearance

In a clearance fit, the minimum clearance is the difference between the minimum hole and the maximum shaft. (Fig 4)

Fig 4



The minimum clearnace is 20.000 – 19.993 = 0.007 mm. (Fig 4)

The maximum clearnace is 20.021 – 19.980 = 0.041 mm. (Fig 4)

There is always a clearance between the hole and the shaft. This is the clearance fit.

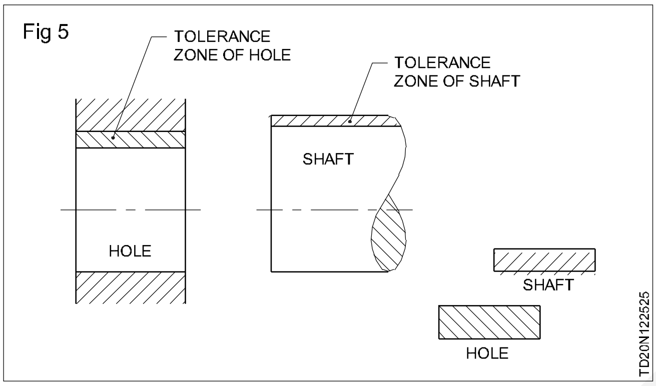
Interference

It is the difference between the size of the hole and the shaft before assembly, and this is negative. In this case, the shaft is always larger than the hole size.

Interference fit

It is a fit which always provides interference. Here the tolerance zone of the hole will be below the tolerance zone of the shaft. (Fig 5)

Fig 5

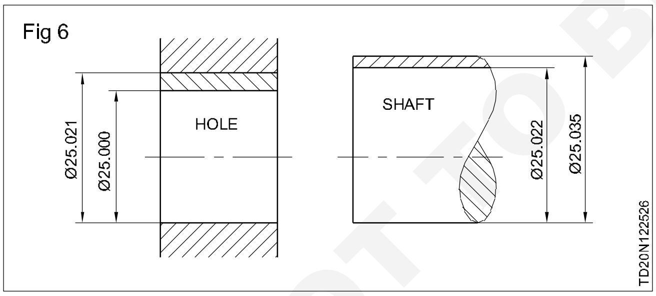


Example

Fit 25 H7/p6 (Fig 6)

The limits of the hole are 25.000 and 25.035 mm. and the limits of the shaft are 25.022 and 25.035. The shaft is always bigger than the size of the hole. This is an interference fit.

Fig 6



Maximum interference

In an interference fit, it is the algebraic difference between the minimum hole size and the maximum shaft size . (Fig 7)

Minimum interference

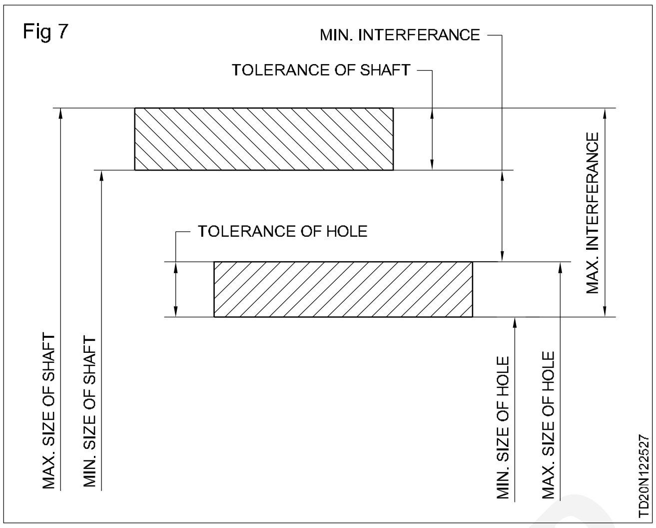
In an interference fit, it is the algebraic difference between the maximum hole size and minimum shaft size. (Figs 7 & 8)

In the example shown in figure 6,

the maximum interference is = 25.035 – 25.000

= 0.035.

Fig 7



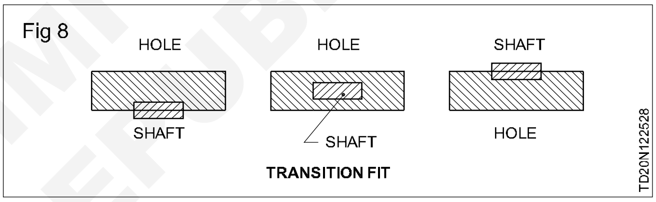
the minimum inteference is = 25.022 – 25.021

= 0.001.

Transition fit

It is a fit which may sometimes provide clearance, and sometimes interference. When this class of fit is represented graphically, the tolerance zones of the hole and shaft will overlap each other. (Fig 8)

Fig 8



Example

Fit 75 H8/j7 (Fig 9)

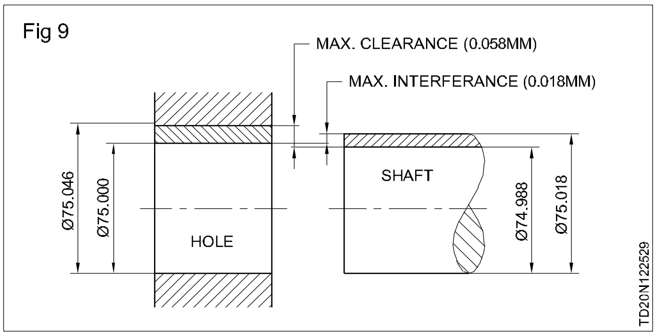
The limits of the hole are 75.000 and 75.046 mm and those of the shaft are 75.018 and 74.988 mm.

Maximum clearnace = 75.046 – 74.988

= 0.058 mm.

If the hole is 75.000 and the shaft 75.018 mm, the shaft is 0.018 mm bigger than the hole. This results in interference. This is a transition fit because it can result in a clearance fit or an interference fit.

Fig 9

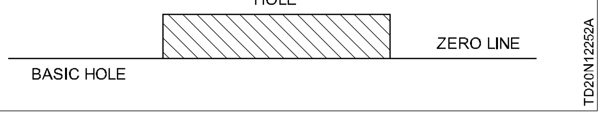


Hole basis system

In a standard system of limits and fits, where the size of the hole is kept constant and the size of the shaft is varied to get the different classes of fits, it is known as the hole basis system.

The fundamental deviation symbol 'H' is chosen for the holes, when the hole basis system is followed. This is because the lower deviation of the 'H' hole is zero. It is known as the 'basic hole'. (Fig 10)

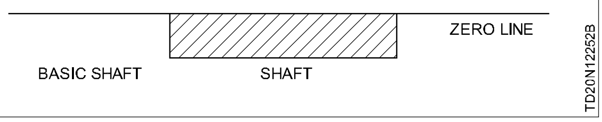
Fig 10



Shaft basis system

In a standard system of limits and fits, where the size of the shaft is kept constant and the variations are given to the hole for obtaining different classes of fits, then it is known as shaft basis system. The fundamental deviation symbol 'h' is chosen for the shaft when the shaft basis is followed. This is because the upper deviation of the 'h' shaft is zero. It is known as the 'basic shaft'. (Fig 11)

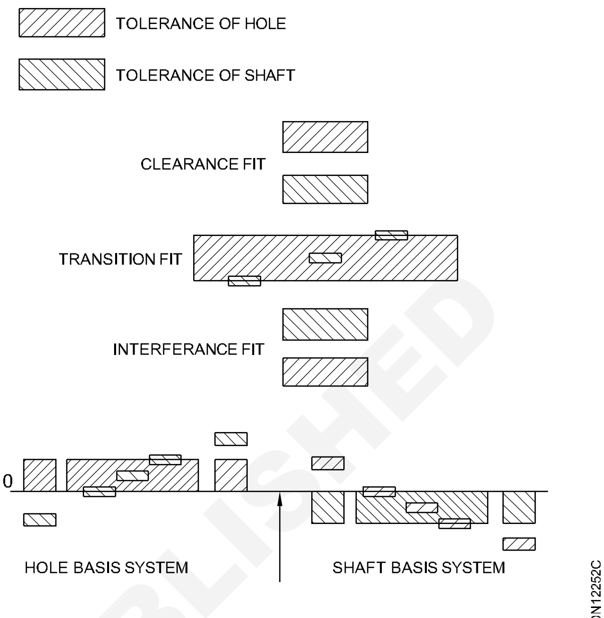
Fig 11



The hole basis system is followed mostly. This is because, depending upon the class of fit, it will be always easier to alter the size of the shaft as it is external, but it is difficult to do minor alterations to a hole. Moreover the hole can be produced by using standard toolings.

The three classes of fits, both under the hole basis and the shaft basis, are illustrated in figure 12.

Fig 12



EXAMPLES ILLUSTRATING THE HOLE BASIS AND SHAFT BASIS SYSTEMS

The B.I.S. system of limits and fits - reading the standard chart

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

- **explain to the standard limit system chart and state the limits of sizes.**

The standard chart covers sizes up to 500 mm (I.S.919 of 1963) for both holes and shafts. It specifies the upper and lower deviations for a certain range of sizes for all combinations of the 25 fundamental deviations, and 18 fundamental tolerances.

The upper deviation of the hole is denoted as ES and the lower deviation of the hole is denoted as EI. The upper deviation of the shaft is denoted as 'es' and the lower deviation of shaft is denoted as 'ei'.

Note

es is expanded as ECART SUPERIOR and **ei** as ECART INFERIOR.

Determining the limits from the chart

Note whether it is an internal measurement or an external measurement.

Note the basic size.

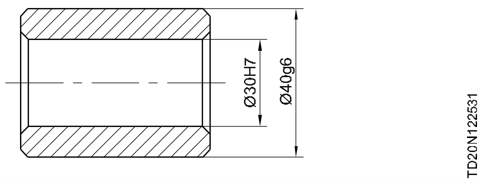
Note the combination of the fundamental deviation and the grade of tolerance.

Then refer to the chart and note the upper and lower deviations which are given in microns, with the sign. Accordingly add or subtract from the basic size and determine the limits of size of the components.

Example

30H7 (Fig 1)

Fig 1



It is an internal measurement. So we must refer to the chart for 'holes'.

The basic size is 30 mm. So see the range 30 to 40.

Look for es, and ei values in microns for H7 combination for 30 mm basic size.

It is given as +0 to +25

Therefore, the maximum limit of the hole is 30 + 0.025 = 30.025 mm.

The minimum limit of the hole is 30 + 0.000 = 30.000 mm.

Refer to the chart and note the values of 40 g6.

Note

The table for tolerance zones and limits as per IS2709 is attached. (Table 1)

Table for Tolerance zones & Limits (Dimensions in µm)

Nominal size range		s6	r6	p6	n6	k6	js6	h6	h7	h9	h11	g6	f7	e8	d9	c11	b11	a11	S7	R7	P7	N7	K7	JS7	H7	H8	H9	H11	G7	F8	E9	D10	C11	B11	A11
From up to	1 3	+20 +14	+16 +10	+12 +6	+10 +4	+6 0	+3 -3	0 -6	0 -10	0 -25	0 -60	-2 -8	-6 -16	-14 -28	-20 -45	-60 -120	-140 -200	-270 -330	-14 -24	-10 -20	-6 -16	-4 -14	0 -10	+5 -5	+10 0	+14 0	+25 0	+60 0	+12 +2	+20 +6	+39 +14	+60 +20	+120 +60	+200 +140	+330 +270
Over up to	3 6	+27 +19	+23 +15	+20 +12	+16 +8	+9 +1	+4 -4	0 -8	0 -12	0 -30	0 -75	-4 -12	-10 -22	-20 -28	-30 -60	-70 -145	-140 -215	-270 -345	-15 -27	-11 -23	-8 -20	-4 -16	+3 -9	+6 -6	+12 0	+18 0	+30 0	+75 0	+16 +4	+28 +10	+50 +20	+78 +30	+145 +70	+215 +140	+345 +270
Over up to	6 10	+32 +23	+28 +19	+24 +15	+19 +10	+10 +1	+4.5 -4.5	0 -9	0 -15	0 -36	0 -90	-5 -14	-13 -28	-25 -47	-40 -76	-80 -170	-150 -240	-280 -370	-17 -32	-13 -28	-9 -24	-4 -19	+5 -10	+7.5 -7.5	+15 0	+22 0	+36 0	+90 0	+20 +5	+35 +13	+61 +25	+98 +40	+170 +80	+240 +150	+370 +280
Over up to	10 14	+39	+34	+29	+23	+12	+5.5	0	0	0	0	-6	-16	-32	-50	-95	-150	-290	-21	-16	-11	-5	+6	+9	+18	+27	+43	+110	+24	+43	+75	+120	+205	+260	+400
Over up to	14 18	+28	+23	+18	+12	+1	-5.5	-11	-18	-43	-110	-17	-34	-59	-93	-205	-260	-400	-39	-34	-29	-23	-12	-9	0	0	0	0	+7	+16	+32	+50	+95	+150	+290
Over up to	18 24	+48	+41	+35	+28	+15	+6.5	0	0	0	0	-7	-20	-40	-65	-110	-160	-300	-27	-20	-14	-7	+6	+10.5	+21	+33	+52	+130	+28	+53	+92	+149	+240	+290	+430
Over up to	24 30	+35	+28	+22	+15	+2	-6.5	-13	-21	-52	-130	-20	-41	-73	-117	-240	-290	-430	-48	-41	-35	-28	-15	-10.5	0	0	0	0	+7	+20	+40	+65	+110	+160	+300
Over up to	30 40	+59	+50	+42	+33	+18	+8	0	0	0	0	-9	-25	-50	-80	-120 -280	-170 -330	-310 -470	-34	-25	-17	-8	+7	+12.5	+25	+39	+62	+160	+34	+64	+112	+180	+280 +120	+330 +170	+470 +310
Over up to	40 50	+43	+34	+26	+17	+2	-8	-16	-25	-62	-160	-25	-50	-89	-142	-130 -290	-180 -340	-320 -480	-59	-50	-42	-33	-18	-12.5	0	0	0	0	+9	+25	+50	+80	+290 +130	+340 +180	+480 +320
Over up to	50 65	+72 +53	+60 +41	+51	+39	+21	+9.5	0	0	0	0	-10	-30	-60	-100	-140 -330	-190 -380	-340 -530	-42 -72	-30 -60	-21	-9	+9	+15	+30	+46	+74	+190	+40	+76	+134	+220	+330 +140	+380 +190	+530 +340
Over up to	65 80	+78 +59	+62 +43	+32	+20	+2	-9.5	-19	-30	-74	-190	-29	-60	-106	-174	-150 -340	-200 -390	-360 -550	-48 -78	-32 -62	-51	-39	-21	-15	0	0	0	0	+10	+30	+60	+100	+340 +150	+390 +200	+550 +360
Over up to	80 100	+93 +71	+73 +51	+59	+45	+25	+11	0	0	0	0	-12	-36	-72	-120	-170 -390	-220 -440	-380 -600	-58 -93	-38 -73	-24	-10	+10	+17.5	+35	+54	+87	+220	+47	+90	+159	+260	+390 +170	+440 +220	+600 +380
Over up to	100 120	+101 +79	+76 +54	+37	+23	+3	-11	-22	-35	-87	-220	-34	-71	-126	-207	-180 -400	-240 -460	-410 -630	-66 -101	-41 -76	-59	-45	-25	-17.5	0	0	0	0	+12	+36	+72	+120	+400 +180	+460 +240	+630 +410
Over up to	120 140	+117 +92	+88 +63													-200 -450	-260 -510	-460 -710	-77 -117	-48 -88													+450 +200	+510 +260	+710 +460
Over up to	140 160	+125 +100	+90 +65	+68 +43	+52 +27	+28 +3	+12.5 -12.5	0 -25	0 -40	0 -100	0 -250	-14 -39	-43 -83	-85 -148	-145 -245	-210 -460	-280 -530	-520 -770	-85 -125	-50 -90	-28 -68	-12 -52	+12 -28	+20 -20	+40 0	+63 0	+100 0	+250 0	+54 +14	+106 +43	+185 +85	+305 +145	+460 +210	+530 +280	+770 +520
Over up to	160 180	+133 +108	+93 +68													-230 -480	-310 -560	-580 -830	-93 -133	-53 -93													+480 +230	+560 +310	+830 +580
Over up to	180 200	+151 +122	+106 +77													-240 -530	-340 -630	-660 -950	-105 -151	-60 -106													+530 +240	+630 +340	+950 +660
Over up to	200 225	+159 +130	+109 +80	+79 +50	+60 +31	+33 +4	+14.5 -14.5	0 -29	0 -46	0 -115	0 -290	-15 -44	-50 -96	-100 -172	-170 -285	-260 -550	-380 -670	-740 -1030	-113 -159	-63 -109	-33 -79	-14 -60	+13 -33	+23 -23	+46 0	+72 0	+115 0	+290 0	+61 +15	+122 +50	+215 +100	+355 +170	+550 +260	+670 +380	+1030 +740
Over up to	225 250	+169 +140	+113 +84													-280 -570	-420 -710	-820 -1110	-123 -169	-67 -113													+570 +280	+710 +420	+1110 +820
Over up to	250 280	+190 +158	+126 +94	+88	+66	+36	+16	0	0	0	0	-17	-56	-110	-190	-300 -620	-480 -800	-920 -1240	-138 -190	-74 -126	-36	-14	+16	+26	+52	+81	+130	+320	+69	+137	+240	+400	+620 +300	+800 +480	+1240 +920
Over up to	280 315	+202 +170	+130 +98	+56	+34	+4	-16	-32	-52	-130	-320	-49	-108	-191	-320	-330 -650	-540 -860	-1050 -1370	-150 -202	-78 -130	-88	-66	-36	-26	0	0	0	0	+17	+56	+110	+190	+650 +330	+860 +540	+1370 +1050
Over up to	315 355	+226 +190	+144 +108	+98	+73	+40	+18	0	0	0	0	-18	-62	-125	-210	-360 -720	-600 -960	-1200 -1560	-169 -226	-87 -144	-41	-16	+17	+28.5	+57	+89	+140	+360	+75	+151	+265	+440	+720 +360	+960 +600	+1500 +1200
Over up to	355 400	+244 +208	+150 +114	+62	+37	+4	-18	-36	-57	-140	-360	-54	-119	-214	-350	-400 -760	-680 -1040	-1350 -1710	-187 -244	-93 -150	-98	-73	-40	-28.5	0	0	0	0	+18	+69	+125	+210	+760 +400	+1040 +680	+1710 +1350
Over up to	400 450	+272 +232	+166 +126	+108	+80	+45	+20	0	0	0	0	-20	-68	-135	-230	-440 -840	-760 -1160	-1500 -1900	-209 -272	-103 -166	-45	-17	+18	31.5	+63	+97	+155	+400	+83	+165	+290	+480	+840 +440	+1160 +760	+1900 +1500
Over up to	450 500	+292 +252	+172 +132	+68	+40	+5	-20	-40	-63	-155	-400	-60	-131	-232	-385	-480 -880	-840 -1240	-1650 -2050	-229 -292	-109 -172	-108	-80	-45	-31.5	0	0	0	0	+20	+68	+135	+230	+880 +480	+1240 +840	+2050 +1650

Other system of limits and fits

British standard system (BS)

British standard system is the system which is based on ISO systems

BS 1916-1 gives guidance on tolerances, limits and fits for engineering sizes up to 19.69 in. The recommendations apply particularly to fits between cylindrical parts, designated as "holes" and "shafts", in which case the term "size" refers to the diameter of the mating parts.

DIN standard system

A DIN standard is a standard drawn up at the German Institute for Standardization (DIN) in Berlin that sets unified standards for products and processes, such as quality, minimum performance, characteristics, dimensions, etc.

General tolerance for linear and angular dimensions are are coverd in DIN ISI 2768 T1

Japanese Industrial Standards (JIS)

Japanese Industrial Standards (JIS) specifies the standards used for industrial activities in Japan . The standardization process is coordinated by the Japanese Industrial Standards Committee and published through the Japanese Standards Association. (JSA)

JSA - JIS B 0405

General Tolerances - Part 1: Tolerances for Linear and Angular Dimensions Without Individual Tolerance Indications

ANSI Standard Limits and Fits (ANSI B4.1-1967)

ANSI, This American Standard for preferred limits and fits for cylindrical parts presents definitions of terms applying to fits between non threaded cylindrical and makes some recommendations on preferred sizes, fits, tolerances, and allowances for use where they are applicable. The ANSI B4.1 charts data are provided in thousandths (.001) of an inch.

Europe standard

European Committee for Standardization (CEN). Is specify the standard

Tolerance standards vary depending on the manufacturing processes. For example, the most common tolerance standards that are used by engineers in Europe for subtractive manufacturing (e.g. CNC machining) are defined by ISO 2768 and ISO 286

Indian Standard (IS)

This Indian Standard (Second Revision) which is identical with IS0 236-I : 1988 'IS0 system of limits arid fits - Part 1 : Bases of tolerances, deviations and fits' was adopted by the Bureau of Indian Staridards on the recommendations of the Engineering Standards Sectional Committee

IS0 SYSTEM OF LIMITS AND FITS

International Organization for Standardization (ISO)

IS 919 (Part 1) : 1993 IS0 288 - 1 : 1988

This part of IS0 286 gives the bases of the IS0 system of limits and fits together with the calculated values of the standard tolerances and fundamental deviations. These values shall be taken as authoritative for the application of the system.

Geometrical tolerancing

Objectives: At the end of this lesson you shall be able to
- **define geometrical tolerance**
- **state the necessity of using geometrical tolerances**
- **identify the recommended symbols for tolerancing**
- **under the three groups of form, altitude 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

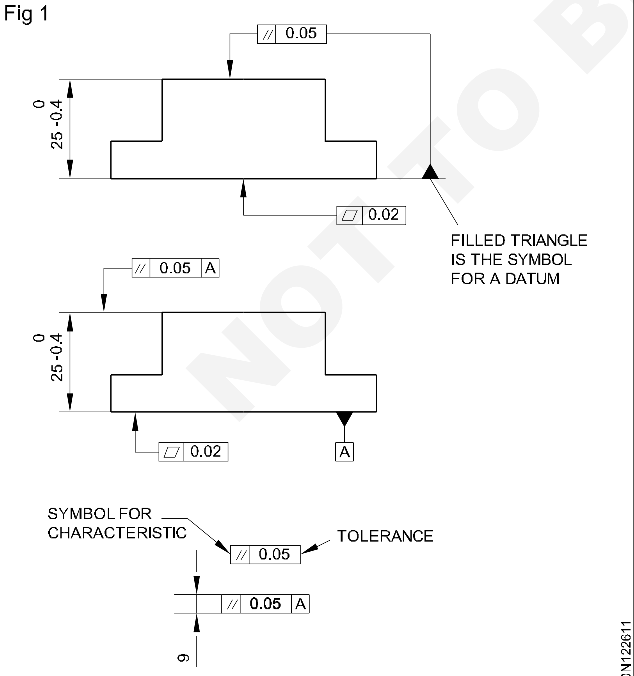
This will help the operator to produce the components, particularly those parts which must fit together precisely.

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



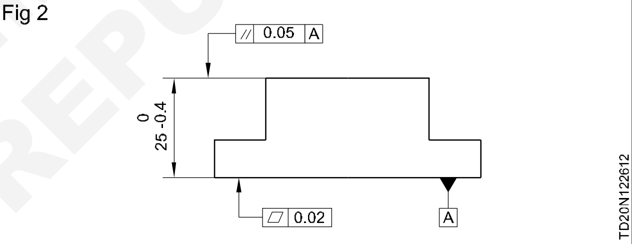
FRAME FOR INDICATING GEOMETRICAL TOLERANCE

Notice that from each of the frames a leader is drawn so that it is normal, it.e. at 90º to the relevant face and ending with an arrow-head against the face.

Notice also that from the 'paralleslism' 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 won 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.

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 (Fig3). Where (3a), (3b), (3c) & (3d) show the use of geometrical tolernaces 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 of the part. This means that the geometrical tolerance 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.

Fig 3

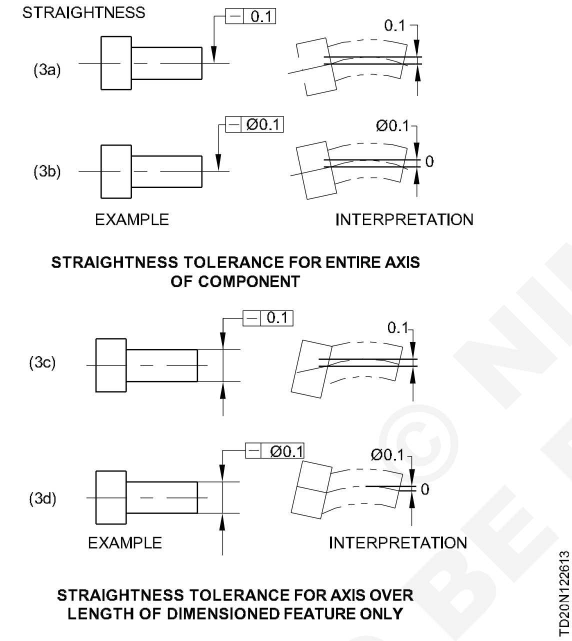
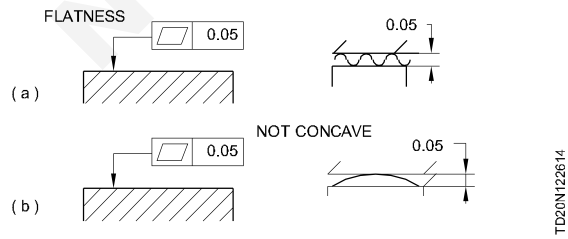


Fig 4 deal with the geometrical tolerance for flatness of a surface, where the sumbol for flatness is followed by the tolerance of 0.05. This Fig (4a) indicates that the actual surface must be between two parallel planes 0.05 apart. If a particular form of direction is prohibited, then this is stated in a note form against the tolerance frame. Eg. "Not concave." (Fig 4b)

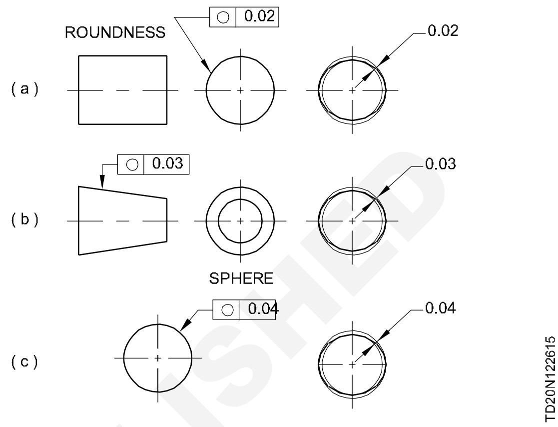
Fig 4



The geometrical tolerance controlling roundness of a part is shown in Fig 5. The interpretation for (5a) and (5b) is that the true form of the periphery of the part at any cross-section perpendicualr to the axis must lie between two concentric circles whose radial distance apart is 0.02 for (5a) and 0.03 for (5b).

For the sphere shown in (5c) the geometrical tolerance applies to concentric circles with the radial distance apart 0.04 at the periphery at any section of maximum diameter.

Fig 5



The symbol controlling cylindricity is shown in Fig 6. 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

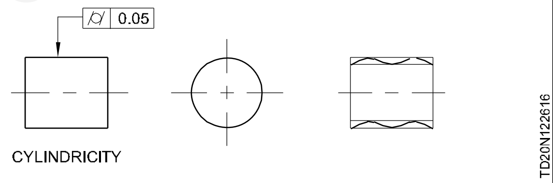
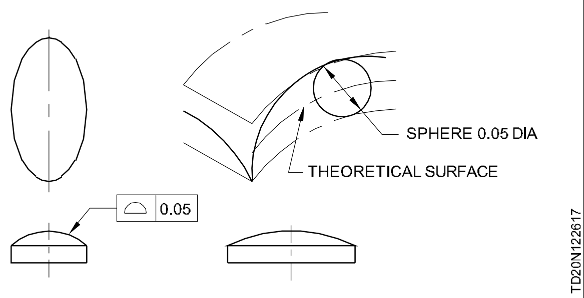


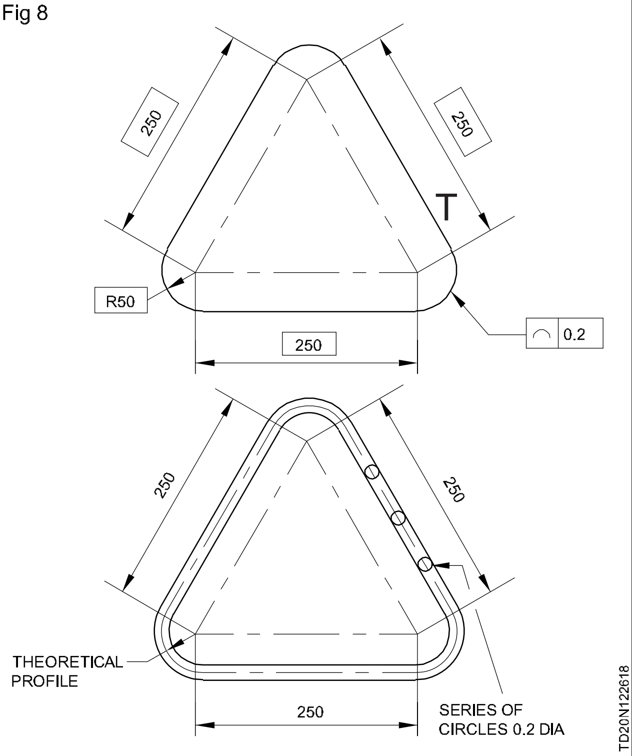
Fig 7. shows the method of applying a geometrical tolerance to a curved surface. The symbol is followed by the tolerance 0.05 which means that the actual surface must lie between two surfaces enveloping a succession of sphere 0.05 diameter whose centre lies on the theoretical surface.

Fig 7



In Fig 8. 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.

Fig 8



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.

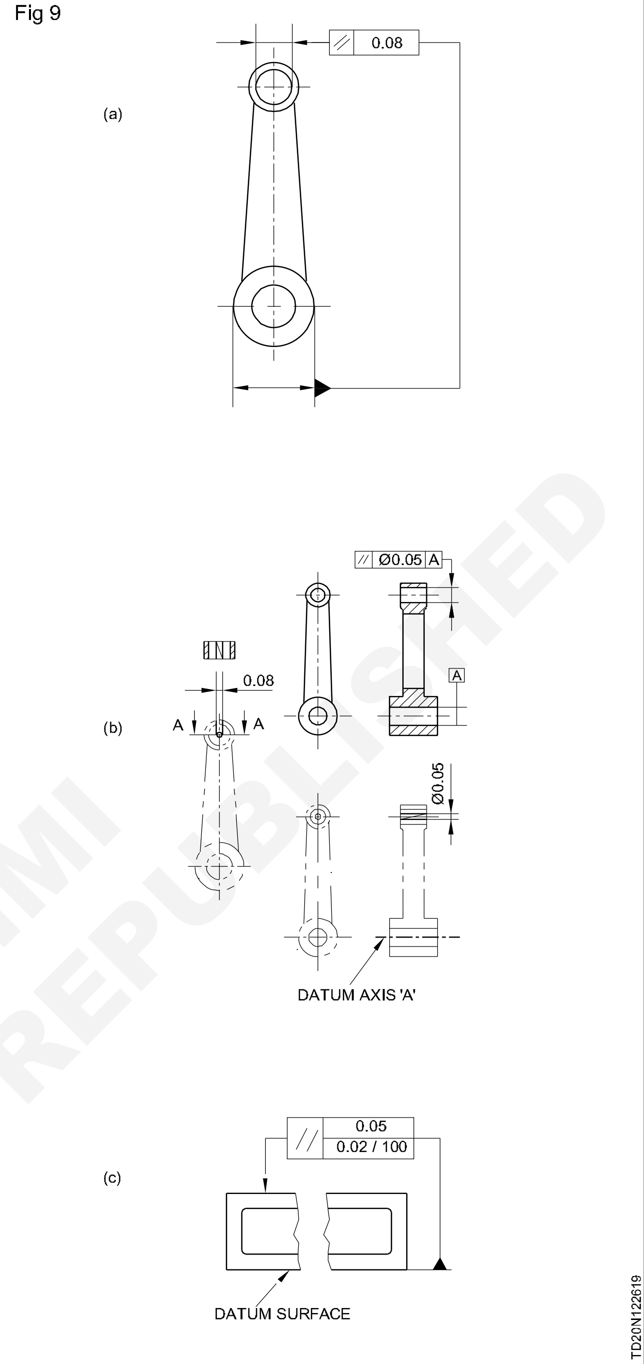
Tolerances of attitude are identified and indicated by the use of symbols for the following.

Characteristic	Symbol
Parallelism	//
Squareness	⊥
Angularity	∠

A typical application of tolerances for these three characteristics is shown in figures 9,10,11,12,13,14,15 and 16. Fig (9a) and (9c) show the application of tolerancing to control 'parallelism.' (9a) shows that the axis of the upper hole must lie between the two lines 0.08 apart which are paralled with the datum axis, i.e., the axis of the lower hole, as indicated by the leader ending in the blacked-in triangle. In (9b) the method uses a separate datum letter 'A' which is added to the frame after the tolerance of 0.05 diameter. (Note the symbol ∅) 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 (9c) shows a component whose upper surface must be between two parallel planes 0.05 apart, paralled with teh 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.

Examples of the application of the geometrical tolerance for 'squareness' are shown in Figs 10, 11, 12 and 13 with 10, 11 and 12 using the separate box method for indicating the datum.

Fig 9



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 holes; (Fig 10).

The axis of the upper cylindrical portion must lie within a cylindrical tolerance zome of 0.1 diameter, the axis of which is perpendicular to the datum surface 'A'. (Fig 11)

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 12)

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. (13)

Fig 10

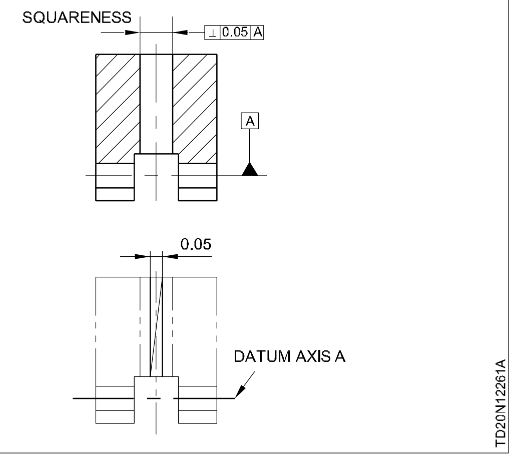


Fig 11

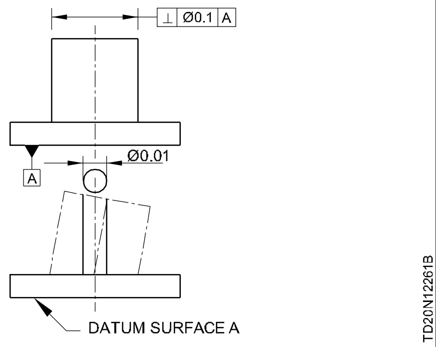


Fig 12

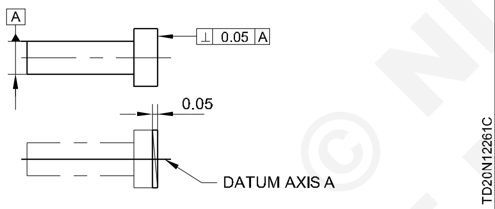
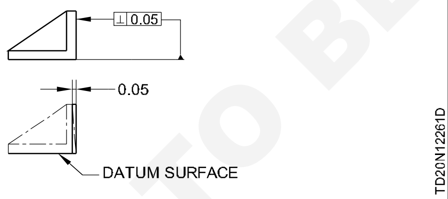


Fig 13



Geometrical tolerances for the control of ANGULARITY are shown in Figs 14, 15 & 16.

The Fig 14 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 60º to the datum surface A.

In Fig 15 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 75º to the datum axis A of the through hole.

Fig 16 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 35º to the base, the datum surface. Notice that the theoretical angle in each example is boxed.

Fig 14

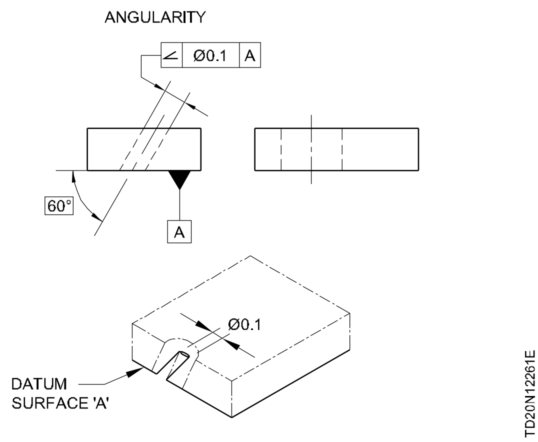


Fig 15

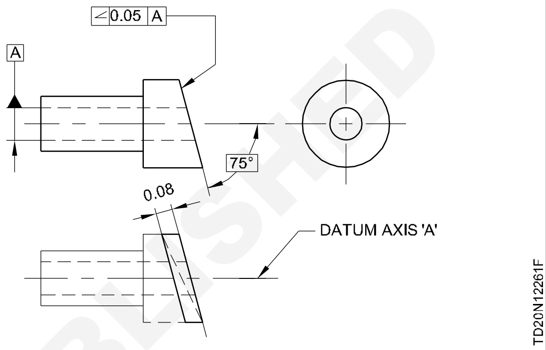
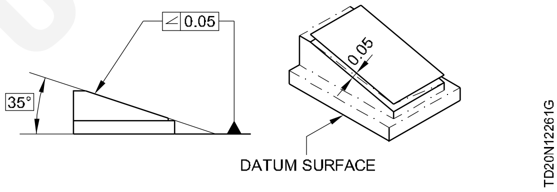


Fig 16



Tolerancces oflocation are identified and indicated by the use of symbols for the following characteristics.

Characteristic	Symbol
Position	⌖
Concentricity	◎
Symmetry	⌯

Figs 17, 18 & 19 show typical examples of these characteristics and symbols. In Fig 17 the holw centre dimensions of 25 and 30 are boxed to show that these are the theoretical dimensions. The geometrical tolerance requires that the holw 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 18 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 17

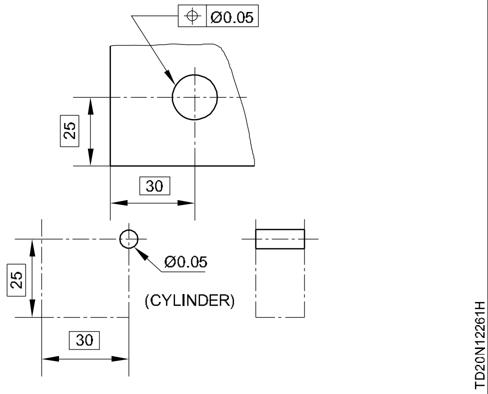
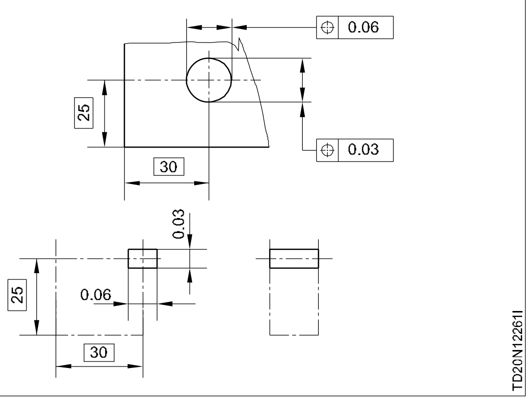
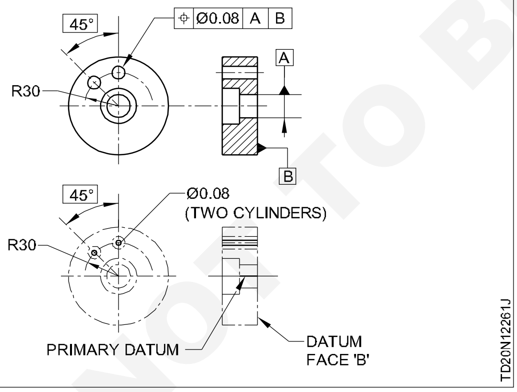


Fig 18



In (Fig 19) the two holes are shown with their true position spaced at 45º on a 30 mm 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 19



Examples of geometrical tolerance for 'CONCENTRICITY' are given in Figs 20,21 & 22. the interpretations are as follows.

In Fig 20 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'.

In Fig 21 the axis of the two end portions must lie within a cylindrical tolerance zone 0.08 dia.

Fig 20

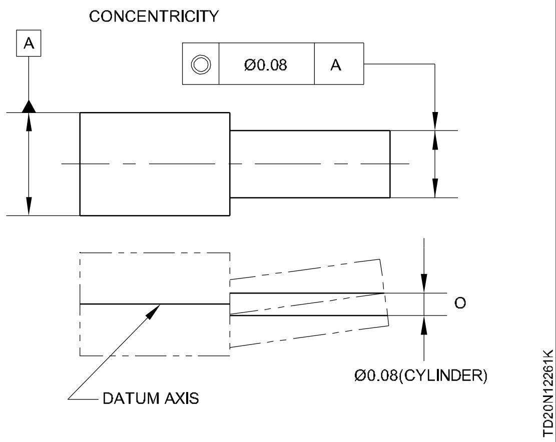


Fig 21

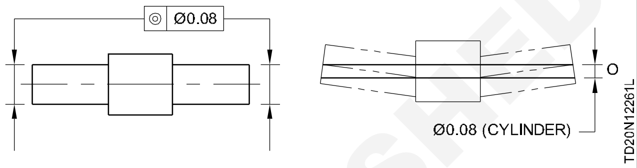
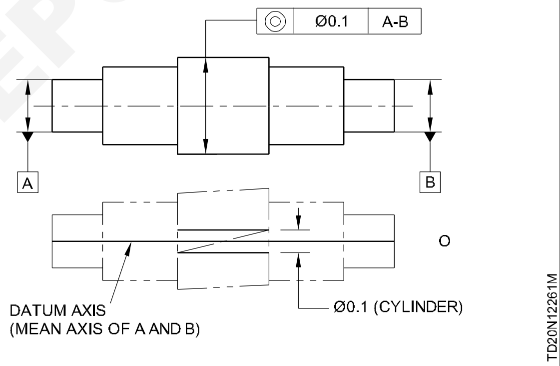


Fig 22 the axis of the large central portion must lie within a cylindrical zone 0.1 diameter which is coaxial with the mean axis of the datum diameters 'A' and 'B'. (Notice that to indicate the requirement of the mean azis the datum letters are separated by a hyphen and enclosed in the same compartment of the tolerance frame)

Fig 22



The geometrical tolerance of 'SYMMETRY' follows in Figs 23, 24 & 25 where the interpretations are:

Fig 23 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 23

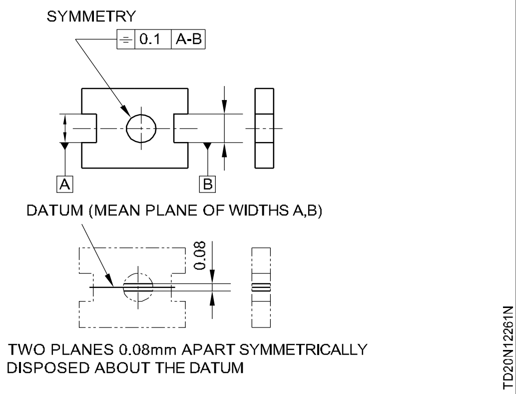


Fig 24 the mean plane of the slot must be between two parallel planes, 0.05 apart symmetically disposed about the mean plane of the datum width 'W'

Fig 25 the median planes of the two end slots must be between two parallel planes, 0.06 apart.

Fig 24

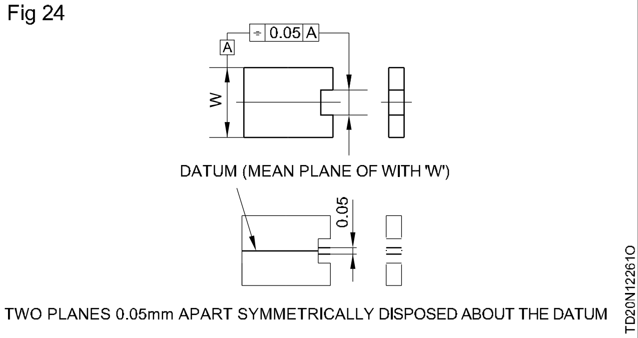
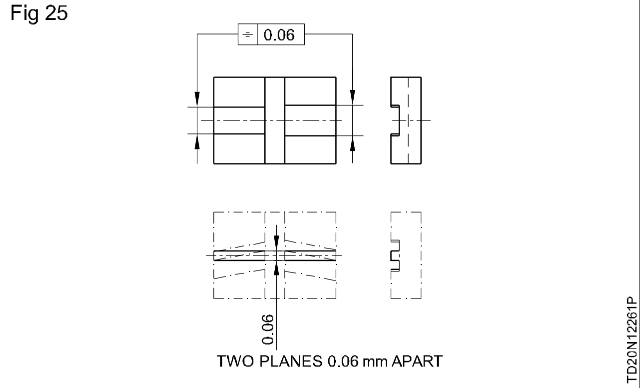
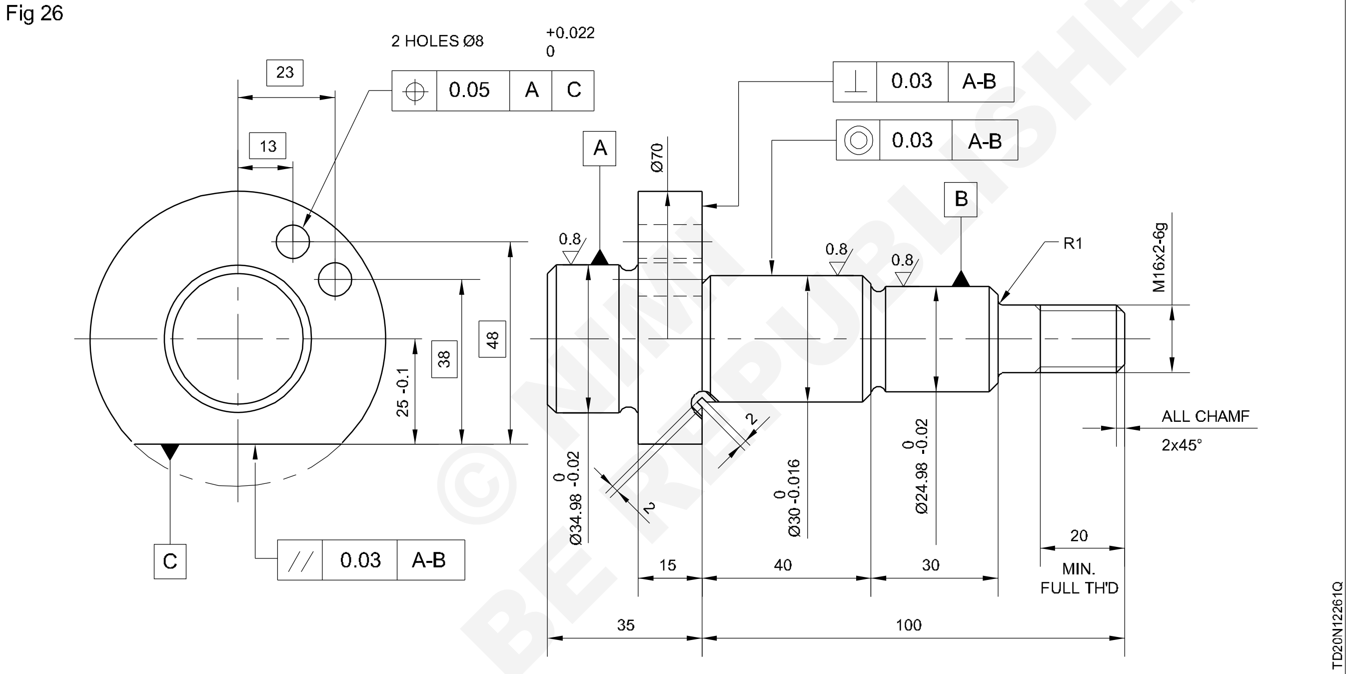


Fig 25



A details drawing which indicates geometrical tolerances (Fig 26)

Fig 26



Physical and mechanical properties of metals

Objectives: At the end of this exercise you shall be able to
- **state the different physical and mechanical properties of materials**
- **state the characteristics of the mechanical properties of metals.**

Properties of metals

Metals have different properties. Depending on the type of application, different metals are selected.

Physical properties of metals

- Colour
- Weight/Specific gravity
- Structure
- Conductivity
- Magnetic property
- Fusibility

Colour

Different metals have different colours. For example, copper is of a distinctive red colour. Mild steel is of a blue/black sheen.

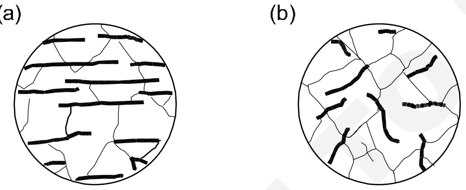
Weight

Metals differ based on their weight. A metal, like aluminium, weighs lighter (specific gravity 2.8) than many others, and a metal, like lead, is heavy (specific gravity 9).

Structure (Figs 1 a and b)

Generally metals can also be differentiated by their internal microstructure. Metals like wrought iron and aluminium will have a fibrous structure, and metals like cast iron and bronze will have a granular structure.

Fig 1 (a) (b)



WROUGHT IRON CAST IRON

TD20N122811

Conductivity

Thermal conductivity and electrical conductivity are the measure of the ability of a material to conduct heat and electricity. Conductivity will vary from metal to metal. Copper and aluminium are good conductors of heat and electricity.

Magnetic property

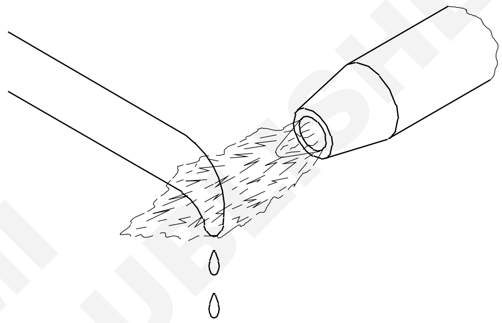
A metal is said to possess magnetic property, if it is attracted by a magnet.

Almost all ferrous metals, excepting some types of stainless steel, can be attracted by a magnet and all non-ferrous metals and their alloys will not be attracted by a magnet.

Fusibility (Fig 2)

It is the property possessed by a metal by virtue of which it melts when heat is applied. Many materials are subject to the transformation in shape (i.e.) from solid to liquid at different temperatures. Tin has a low melting temperature (232^0C)and tungsten melts at a high temperature (3370^0C).

Fig 2



TD20N122812

Mechanical properties

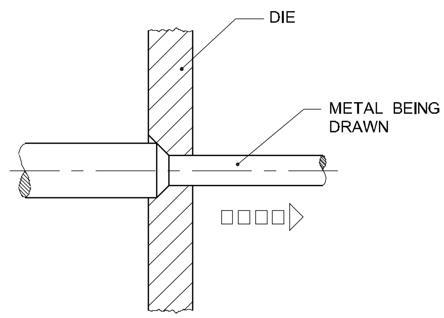
The mechanical properties of a metal are

- ductility
- hardness
- toughness
- elasticity
- malleability
- brittleness
- tenacity

Ductility (Fig 3)

A metal is said to be ductile when it may be drawn out in tension without rupture. Wire-drawing depends upon ductility for its successful operation. A ductile metal must be both strong and plastic. Copper and aluminium are good examples of ductile metals.

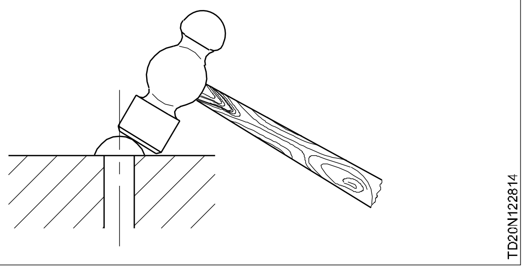
Fig 3



TD20N122813

Malleability (Fig 4): Malleability is the property of permanently extending in all directions without rupture by hammering, rolling etc, to change its size and shape. Lead is a very malleable metal.

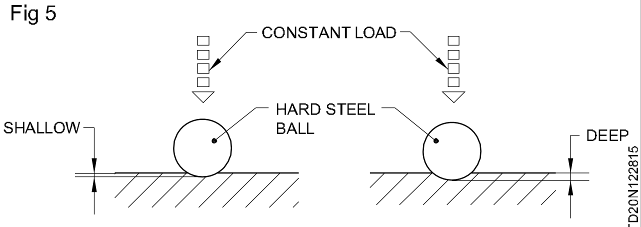
Fig 4



Hardness (Fig 5)

Hardness is a measure of a metal's ability to withstand scratching, wear, abrasion and penetration.

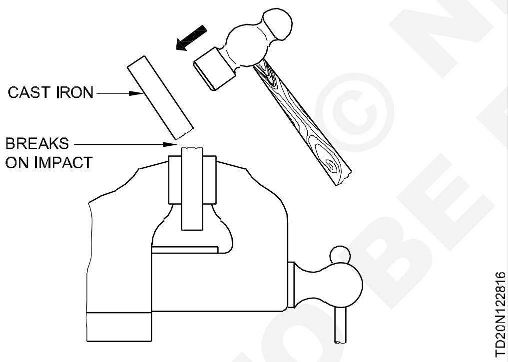
Fig 5



Brittleness (Fig 6)

Brittleness is that property of a metal which permits no permanent distortion before breaking. Cast iron is an example of a brittle metal, and it will break rather than bend under shock or impact.

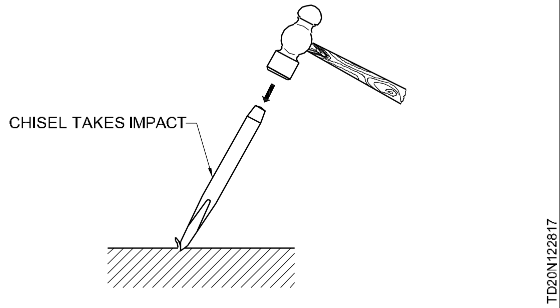
Fig 6



Toughness (Fig 7)

Toughness is the property of a metal to withstand shock or impact. Toughness is the property opposite to brittleness. Wrought iron is an example of a tough metal.

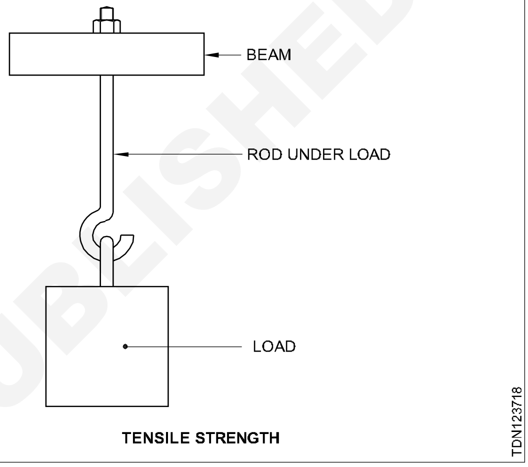
Fig 7



Tenacity (Fig 8)

Tenacity of a metal is its ability to resist the effect of tensile forces without rupture. Mild steel, wrought iron and copper are examples of tenacious metals.

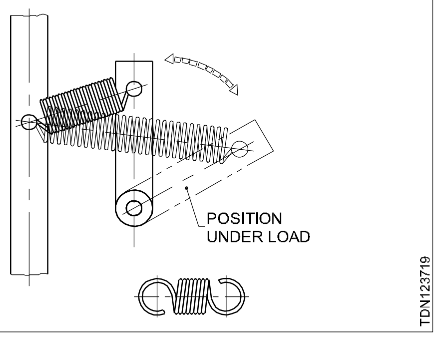
Fig 8



Elasticity (Fig 9)

Elasticity of a metal is its power of returning to its original shape after the applied force is released. Properly heat-treated spring is a good example of elasticity.

Fig 9



Classification of metals and its uses

Objectives: At the end of this lesson you shall be able to
- **state what is ferrous and non ferrous metals**
- **list the different type of ferrous and non ferrous metals**
- **brief the uses of ferrous and non ferrous metals**
- **differentiate between ferrous and non ferrous metals.**

All engineering metals can be classified into two broad categories of metal and non metal. They are ferrous and non ferrous metals.

Ferrous metal: - Metals which contain iron as a major content are called ferrous metal

Non ferrous metal: - Metals that do not contain iron is called non ferrous metal

Pure iron is of little use because it is too soft and ductile

These two categories are further classified as

Ferrous metals

Pig iron

Cast iron

- Grey cast iron,
- white cast iron,
- Alloy cast iron ,
- Malleable cast iron ,
- Nodular cast iron
- Wrought iron

Steel

Alloy steel ,

Stainless steel ,

Carbon steel

- Low carbon steel ,
- Medium carbon steel
- High carbon steel

Non ferrous metals

Copper

Aluminum

Brass bronze

Lead silver

Zinc tin etc

Non metals

Leather

Rubber

Polymers

Wood

Ceramics

Uses of Ferrous Metals

Commonly used examples of ferrous metals include steel, stainless steel, carbon steel, cast iron and wrought iron:

Steel

A combination of iron and carbon, steel is renowned for its strength and machinability. It is widely used in construction, manufacturing and industrial metal fabrication.

Stainless Steel

Stainless steel is an alloy steel made with the addition of chromium to steel, which provides resistance against rust.

Carbon Steel

Carbon steel contains a high carbon content that is added to iron to create an exceptionally hard metal that is used for tools.

Low carbon steel

The low cost and malleability low carbon steels used. For bolts and nuts, forgings, etc

Medium carbon steel

Medium carbon steels are mainly used for making different automotive industry components like gears, axles, shafts but also bolts, nuts, screws etc. Steels ranging from 0.4…0.6% are also suitable for everything related to locomotives and rails.

High carbon steel

The main use of high-speed steels used for various cutting tools: like drills, taps, tool bits, , saw blades, , router bits, punches and dies etc

Cast Iron

Cast iron is a hard and wear resistant metal that is widely used for items including cookware, machine tools, engines, manhole covers and water pipes.

Grey cast iron

This is widely used for the casting of machinery parts like machine bases , tables , slide ways

White cast iron

It is used in components which should be abrasion resistant

Wrought Iron

Unlike most other ferrous metals, wrought iron is able to resist corrosion and oxidation. It is typically used for fences, railings and gates.

Uses of Non Ferrous Metals

Commonly-used non-ferrous metals include aluminium, lead, silver, brass, gold, zinc, copper and tin:

Aluminum

Lightweight and easy to machine, shape and weld, aluminium is used for a range of applications from food cans and cookware to aeroplane parts and cars.

Copper

A good conductor of heat and electricity, copper is highly ductile and malleable. It is widely used for electrical wiring as well as in appliances and vehicles.

Lead

Lead is used in electrical power cables, batteries, pipes, fuels, paint and for soldering.

Tin

Soft and malleable with a low tensile strength, tin is used as a coating to prevent steel from corroding.

Silver

Silver is used for a range of applications, including jewellery, cutlery, electrical contacts and in mirrors.

Brass

Brass is used for fixtures and fittings including taps, hooks, and doorknobs, as well as being used for light fittings and screws, among other uses.

Gold

Used for jewellery, gold also has applications including within the medical industry, in computers and also electronics.

Zinc

A medium strength metal with a low melting point, zinc is used to galvanise iron and steel to prevent rusting.

Difference between Ferrous and Non ferrous Metals

The main differences between ferrous and nonferrous Metals are given in table 1

Sl.No	Ferrous Metals	Non Ferrous Metals
1	Ferrous indicates the presence of iron in a bivalent state.	Non-ferrous metals do not contain any iron.
2	As ferrous contains iron, it shows magnetic feature.	Non-ferrous metals don't show any magnetic feature which means it's non-magnetic.
3	Ferrous metals are less resistant to corrosion.	Non-ferrous metals are more resistant to corrosion
4	One special feature of ferrous metals is it possesses high tensile strength and durability.	One special feature of non-ferrous metals is their malleability.
5	Ferrous metals can be oxidized.	Non-ferrous metals cannot be oxidized.
	Ferrous metals make up the most recycled materials in the world	As per the recycling goes, many non-ferrous materials are relatively insufficient.
6	The price of ferrous metal tends to be lower.	Prices of non-ferrous metals are greater than ferrous metals

Heat treatment of plain carbon steels

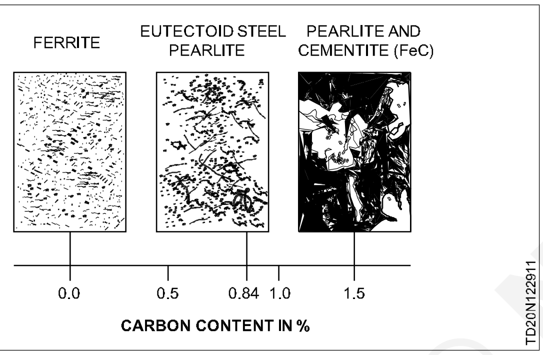
Objectives: At the end of this lesson you shall be able to
- **state the purpose of heat treatment of steel**
- **state the types of structure, constituents and properties of plain carbon steels.**

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 structures of steel (Fig 1)

Fig 1



The structure of steel becomes visible when a piece of the metal is broken. The exact grain size structure can be seen through a microscope. Steel is also classified according to its type of 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% carbon is called 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.

Hypoeutectoid steel

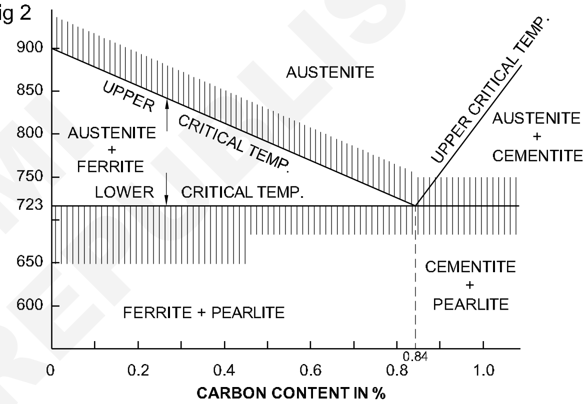
Less than 0.84% carbon steel or hypoeutectoid steel is pearlite and ferrite.

Structure of steel when heated (Fig 2)

If steel is heated, a change in structure commences from 723^0C. 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, very hard and magnetic. It is extremely wear resistant and can cut other metals.

Fig 2



HEAT TREATMENT - RANGES OF STEELS

Heat treatment processes and purposes

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 structure of the steel.

Heating/Quenching steel for heat treatment

Objectives: At the end of this lesson you shall be able to
- **distinguish between the lower critical and the upper critical temperatures**
- **state the three stages in the ehat treatment process**
- **determine the upper critical temperature for different plain carbon steels from the diagram.**

Critical temperatures

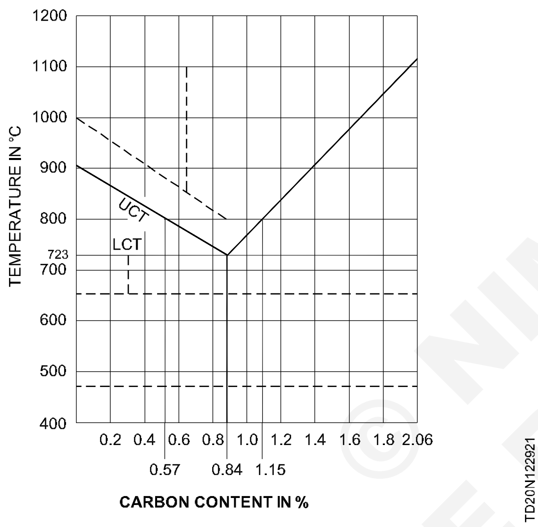
Lower critical temperatures

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)

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 steel 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

The depends upon the cross-section of the steel, its chemical composition, the volume of the charge in the furnace and the arrangement of the charge in the furnace. A good general guide for soaking time in normal conditions is five minutes per 10 mm 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 compositon 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.

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 that 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 teated.

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 furning 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.

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 the steel**
- **state the process of hardening.**

What is hardening?

Hardening is a heat treatment process in which steel is heated tp 30-50^{0}C above the critical range. Soaking 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

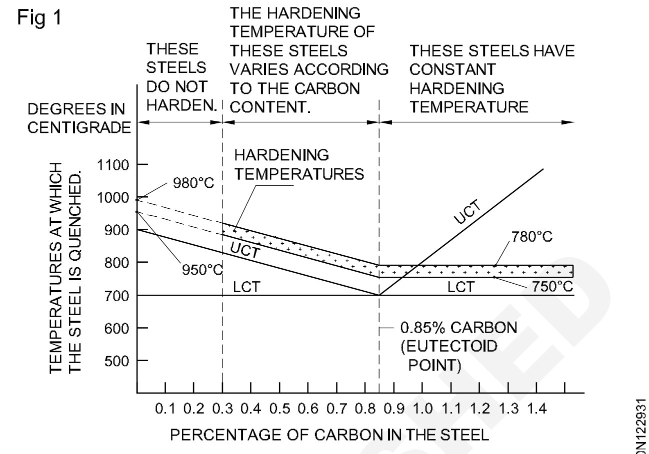
Hardening affects the mechanical properties of steel like strength, toughness, ductility etc.

Hardening adds cutting ability.

Process of hardening

Steel with a carbon content above 0.4% is heated to 30-500°C above the upper critical temperature (Fig 1). A soaking time of 5mts./ 10 mm thickness of steel is allowed. (Fig 1)

Fig 1



HARDENING OF PLAIN CARBON STEELS

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 what is 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^{0}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 (Table 1) 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

Tools or articles	Temperature Centigrade Degrees	Colour
Turning tools	230	Pale straw
Drills and milling cutters	240	Dark straw
Taps and shear blades	250	Brown
Punches, reamers, twist drills	260	Red brown
Rivets, snaps	270	Brown purple
Press tools, cold chisels	280	Dark purple
Cold set for cutting steels	290	Light blue
Springs, screwdrivers	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 annealing of steel**
- **state the purpose of annealing**
- **state the processes 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

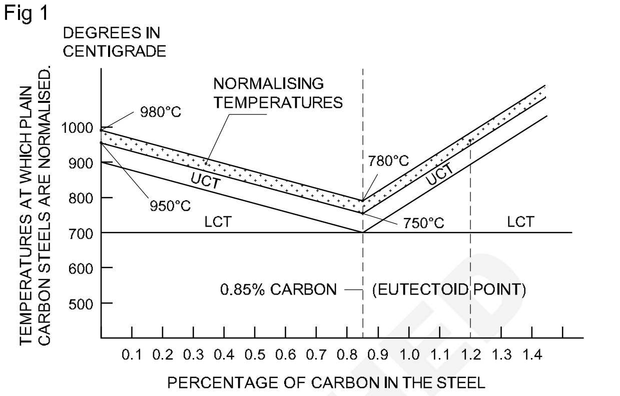
- To soften the steel
- To improve the machinability.
- To increase the ductility.
- To relieve the internal stresses.
- To refine the grain size and to prepare the steel for subsequent heat treatment process.

Annealing process

Annealing consists of heating of hypoeutectoid steels to 30 to 50^0C above the upper critical temperature and 50^0C above the lower critical temperature for hypereutectoid steels. (Fig 1)

Soaking is holding at the heating temperature for 5mts./10mm of thickness for carbon steels.

Fig 1



NORMALISING TEMPERATURES FOR PLAIN CARBON STEELS

The cooling rate for carbon steel is 100 to 150^0C/hr.

Steel, heated for annealing, is either cooled in the furnace itself by switching off the furnace or it is covered with dry sand, dry lime or dry ash.

Normalising 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 components 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.

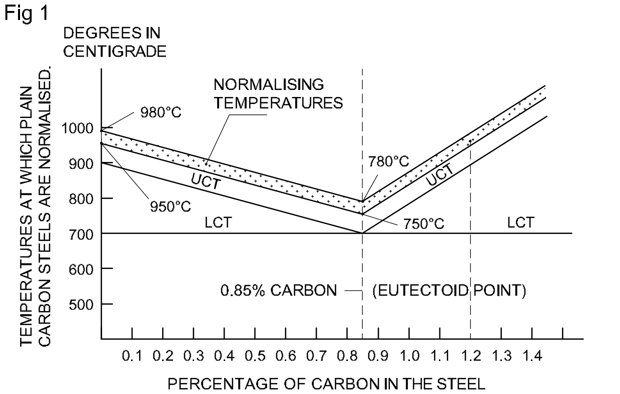
To reduce ductility.

To prevent warping.

Process

To get the best results from normalizing, the parts should be heated uniformly to a temperature of 30 to 400C above the upper critical temperature (Fig 1), 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

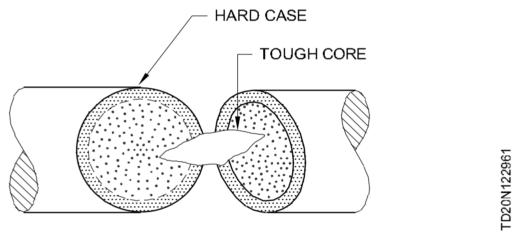
Avoid placing the component in a wet place or wet air, thereby restricting the natural circulation of air around the component. Avoid placing the component on a surface that will chill it.

Surface hardening of steel

Objectives: At the end of this lesson you shall be able to
- **name four different types of surface hardening processes**
- **state the purpose of case hardening**
- **state the purpose of carburising**
- **state the purpose of liquid carburising**
- **state the process of gas carburising.**

Most of the components must have a hard, wear-resisting surface supported by a tough, shock-resisting core for better service condition and longer life. This combination of different properties can be obtained in a single piece of steel by surface hardening. (Fig 1)

Fig 1



Types of surface hardening

- Case hardening
- Nitriding
- Flame hardening
- Induction hardening

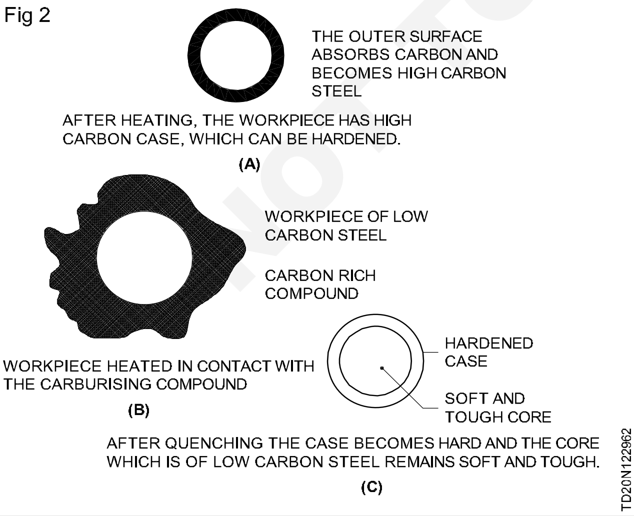
Case hardening

Parts to be hardened by this process are made from a steel with a carbon content of 0.15% so that they will not respond to direct hardening.

The steel is subjected to treatment in which the carbon content of the surface layer is increased to about 0.9%.

When the carburised steel is heated and quenched, only the surface layer will respond, and the core will remain soft and tough as required. (Fig 2)

Fig 2



The surface which must remain soft can be insulated against carburising by coating it with suitable paste or by plating it with copper.

Case hardening takes place in two stages.

- Carburising in which the carbon content of the surface is increased.
- Heat treatment in which the core is refined and the surface hardened.

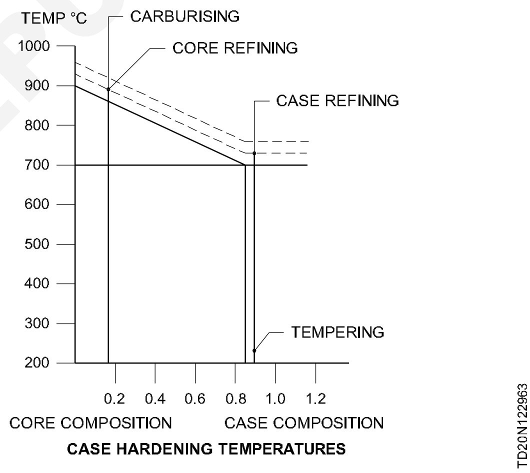
Carburising

In this operation, the steel is heated to a suitable temperature in a carbonaceous atmosphere, and kept at that temperature until the carbon has penetrated to the depth required.

The carbon can be supplied as a solid, liquid or gas.

In all cases, the carbonaceous gases coming from these materials penetrate (diffuse) into the surface of the workpiece at a temperature between 880° and 930°C. (Fig 3)

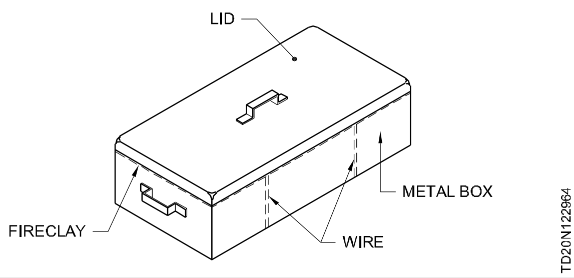
Fig 3



Pack carburising (Fig 4)

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.

Fig 4



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 decarburisation.

Liquid carburising

Carburising can be done in a heated salt bath. (Sodium carbonate, 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 production.

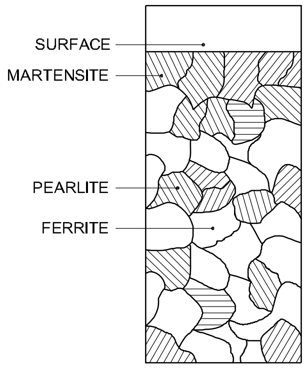
Gas carburising

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' is admitted to the container, and the exit gas is vented.

Fig 5 illustrates the appearance of the structure across its section produced by carburising.

Fig 5



THE MICROSTRUCTURE OF CARBURISED STEEL

Heat treatment

After carburising has been done, the case will contain about 0.9% carbon, and the core will still contain about 0.15% carbon. There will be a gradual transition of the carbon content between the case and the core. (Fig 2)

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 reheated to about 870°C and held at that temperature long enough to produce a uniformity of structure, and is then cooled rapidly to prevent grain growth during cooling.

The temperature of this heating is much higher than that suitable for the case, (Fig 2) and, therefore, an extremely brittle martensite will be produced.

The case and the outer layers of the core must now be refined.

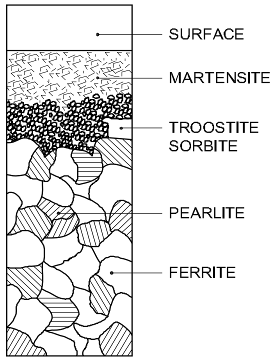
The refining is done by reheating the steel to about 760°C, to suit the case, and quenching it.

Tempering

Finally the case is tempered at about 200°C to relieve the quenching stresses.

Fig 6 illustrates the appearance of the structure across its section produced by case hardening.

Fig 6



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 circulation 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 0.01 to 0.02 mm 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 in 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.

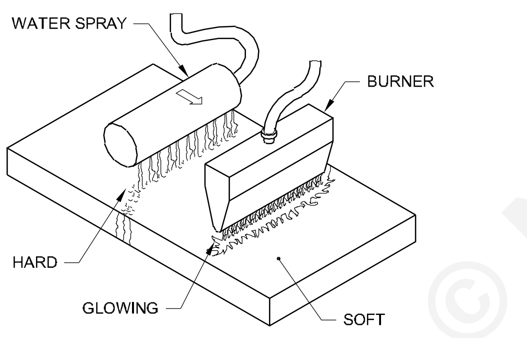
Flame hardening

Objectives: At the end of this lesson you shall be able to
- **state the process of surface hardening using a flame**
- **state the advantages and disadvantages of flame hardening.**

Flame hardening

In this type of hardening, the heat is applied to the surface of the workpiece by specially constructed burners. The heat is applied to the surface very rapidly and the work is quenched immediately by spraying it with water. (Figs 1 & 2) The hardening temperature is generally about 50°C higher than that for full hardening.

Fig 1



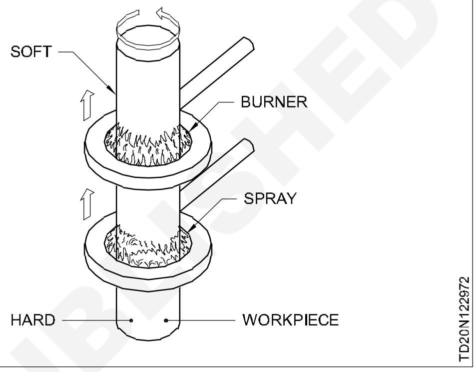
The workpiece is maintained at the hardening temperature for a very short period only, so that the heat is not conducted more than necessary into the workpiece.

Steels used for surface hardening will have a carbon content of 0.35% to 0.7%.

The following are the advantages of this type of hardening.

- The hardening devices are brought to the workpiece.

Fig 2



- It is advantageous for large workpieces.
- Short hardening time.
- Great depth of hardening.
- Easily controlled.
- Small distortion.
- Low fuel consumption.

The following are the disadvantages.

- Not suitable for small workpieces because of the danger of hardening through.
- The workpieces must be stress-relieved before hardening.

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.**

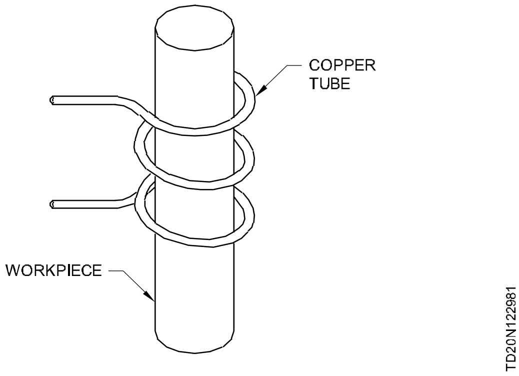
Induction hardening

This is a production method of surface-hardening in which the part to be surface-hardened is placed within an inductor coil through which a high frequency current is passed. (Fig 1) The depth of penetration of the heating becomes less, as the frequency increases.

The depth of hardening for high frequency current is 0.7 to 1.0 mm. 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.

Fig 1



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 production.
- Produce less distortion then convertional heat treatment

The Advantages and Disadvantages of Induction Hardening

Induction hardening is a type of heat-treating process that typically subjects a metal part under induction heating and quenching. After conducting this specific process, the metal part is intended to undergo a martensitic formation, allowing it to obtain features and properties that it does not possess before.

The process of induction heating starts by placing a metal part near a copper coil that has a significant level of alternating current. Through eddy current and hysteresis losses, the said coil can generate heat that is directed at or near the surface of the metal part. Subsequently, the metal part is directed or submerged under a water-based quench, which then initiates its aforementioned martensitic formation.

Induction hardening can bring numerous benefits to the processing of different metal parts. The same process, however, also share some disadvantages that manufacturers may encounter along the way.

Advantages of Induction Hardening

- **Improved Wear Resistance:** One of the advantages of induction hardening is that it can effectively improve the wear resistance of a metal part. Metal parts that have been annealed or treated to a softer condition can significantly obtain improved wear resistance once they undergo this process. After all, any improvement of a part's hardness can likewise improve its wear resistance.
- **Increased Fatigue Life:** The residual compressive stress at the surface of a part out of induction hardening can easily obtain a huge improvement over its strength and fatigue life. The induction hardening allows the part to obtain a hardened structure that occupies more volume than its core.
- **Customised Hardness Level:** With induction hardening, metal parts can now be tempered easily. granting manufacturers the ability to customise their hardness level based on their set requirements. The more these metal parts are tempered, the lower the hardness and brittleness will be.
- Minimised Warpage: Another advantage of induction hardening is that it does not warp the metal parts significantly. You see, other types of hardening processes are known for warping the metal part and its respective surfaces. But with induction hardening, you can easily process materials without significant and noticeable warpage.
- Notable Low Costs: When it comes to overall costing, induction hardening can be done without spending too much money. The process likewise does not require expensive metal parts. So, if you are thinking of processing low cost steels, then you may proceed with induction hardening.

Disadvantages of Induction Hardening

- Occurrences of Cracking: As for the disadvantage of induction hardening, the entire process is more prone to causing some cracks on the surfaces of metal parts compared to other heat treatment processes. Induction hardening can also produce hot spots at specific features like keyways, grooves, cross holes, threads, and many more.
- Greater Distortion Levels: Aside from cracking issues, the process of induction hardening can also cause more distortion on the metal parts than other heat treatment processes. This distortion is brought by the martensitic transformation that the part receives during the process.
- Limitation on Materials: Despite the notable low costs of induction hardening, this process can only work on limited materials. This process can only work with materials that contain enough carbon and other elements so they can support all the features caused by the martensitic transformation.

Definition of machine and machine tool

Objectives: At the end of this lesson you shall be able to
- **explain difference between machine and machine tool**
- **state the history & gradual development of lathe.**

Definition of machine

Machine is a device that performs a related operation to produce desired product. It can be stitching of cloth by a sewing machine, producing a component in a forging machine, or it can be a main production by using a CNC machine technology

A machince tool is defined as a power driven machine capable of holding / supporting the work & tool at sametime, directing / guiding the cutting tool or job or both to perform various metal cutting operations for producing various shapes & sizes

Fundamental of machine tools

Machine tool is a device that utilizes electric energy for shaping and sizing a product by removing excess material in the form of chips, with the help of cutting tool.

Machine tools are used for producing components at a rapid rate. Optimum productivity from machine tool calls for a fairly high degree of skill. Properly carried out operations are capable of producing a large number of components at a fairly rapid rate.

Machine tools and machines are two different things. machine tools when taken as a group can produce a machine tool, which is not true of machines.

Lathe, milling machine, shaping machine, slotter etc., are all machine tools.

Every metal working machine cannot be called a machine tool merely because it removes material. Forging hammers, drawing dies, extruders, rolling machine etc., are not machine tools.

Functions of machine tools

1 To hold and support the workpiece to be machined.

2 To hold and support the cutting tool.

3 To provide requisite motion to the workpiece / tool or both.

4 To regulate the cutting speed and feed of the tool and workpiece.

5 To hold various attachments for different operations. Jobs and tools are held in properly designed devices on a machine tool. Different machine tools are provided with different holding devices.

In a workshop, a machine tool is generally used for producing different shapes and for finishing the surfaces.

Classification of machine tools

1 According to the type of the surface generated.

i Cylindrical work machine tools - Lathes, capstan, turret etc.,

ii Flat surfac machine tools - milling machine, shaping machine planing machine etc.,

2 Classification based on the purpose of the machine tool.

i Single - purpose

ii Multi purpose

iii Special purpose

iv Transfer machine

v Numerically controlled

3 Classification based upon the size of chip

i Machine tools using cutting tools - lathe, milling, planner, slotter etc.,

ii Machine tools using abrasives - honing, lapping Grinding etc.,

Machine tool performance criteria

While designing a machine tool the following factors need consideration.

1 It should be safe and easy to operate.

2 It should be accurate.

3 It should have good production capacity.

4 The operational cost should be low.

5 Controls should be located at convenient points.

6 Blanks should be such that they can be loaded and clamped easily.

Factors in machininig operations

The operations of removing metal by means of the cutting tool using some sort of machine tool in order to obtain a desired shape is called machining.

It includes number of operations such as turning, boring, shaping, milling etc.,

The selection of a machine tool for a particular operation depends upon many factor such as

1 The shape and size of the product required.

2 The quantity of material to be removed.

3 The type of operation to be performed.

4 The number of components required.

5 The type of material to be handled

6 The degree of accuracy required.

Lathe and its parts

Objectives: At the end of this lesson you shall be able to
- **state the main parts of the lathe**
- **state the lever positions**
- **state the various lubrication points.**

Turning and centre lathe

Turning is a machining process to bring the raw material to the required shape and size by metal removal. This is done by feeding a cutting tool against the direction of rotation of the work.

The machine tool on which the job is made to rotate and turning is carried out is known as a lathe.

Constructional features of a lathe (Fig 1)

A lathe should have provision:

- to hold the cutting tool, and feed it against the direction of rotation
- to have parts, fixed and sliding, to get a relative movement of the cutting tool with respect to the rotation of the work
- to have accessories and attachments for performing different operations.

Various Lubrication Points

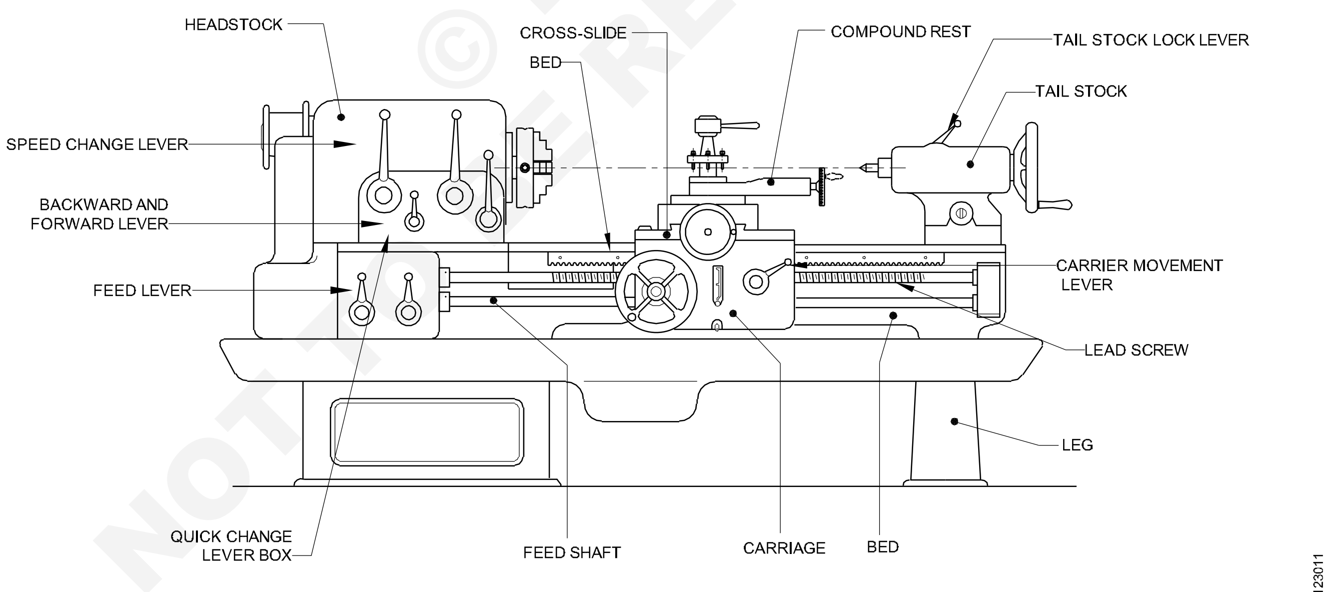
1 Place a few drops of oil on the rockershaft bearing and cams every time the lathe is in use.

2 Countershaft roller bearing - Fill both grease cups with automotive cup grease every two weeks. Give the grease cup caps a turn or so every time the lathe is used.

3 Place a few drops of oil on the rockershaft lever bearings and lever fulcrum bearing every time lathe is used.

4 Motor bearings - sleeve type motors have two oil cups which should be filled once a week with S.A.E. No. 10. Motor oil or equivalent. Ball bearing motors have a sealed in - type bearing - every six months the small headless screw in these bearings should be removed and a moderate quantity of automotive cup grease forced around the bearing.

Fig 1



CENTRE LATHE - MAIN PARTS & LEVER POSITIONS

5 Left and right headstock bearings - oil with No.10 motor oil or equivalent every time the lathe is used.

6 Spindle pulley - every time the lathe is used in backgear, remove the small screw in the bottom of the second step of the idler pulley and oil freely with No. 10 motor oil or equivalent. Replace screw.

7 Back gear spindle - every time the back gears are used, remove the small screw in the centre of the back gear spindle and oil freely with No 10 motor oil or equivalent. Replace screw.

8 Back gears and change gears - A small amount of keystone No. 122 heavy outer gear lubricant or equivalent applied to the gear teeth will aid in obtaining smoother, more quiet operation. Be sure to remove all oil in the gear teeth before applying this lubricant or it will not adhere.

9 Change gear bearings - put a few drops No.10 motor oil or equivalent on the change gear bearings each time the lathe is used.

10 Lead screw stub bearing and reversing gears - put a few drops of No. 10 motor oil or equivalent in the three oil holes on the top of the reversing gear box every time the lathe is used.

11 Carriage traverse gear case - every time the lathe is in use, put a few drops of No 10 motor oil in oil hole on top of gear case on back of carriage apron.

12 Carriage hand wheel bearing - put a few drops of No 10 motor oil or equivalent in the ball spring oil hole every time the lathe is used.

13 Cross feed gear bearing - put a few drops of oil in the ball spring oil hole every time lathe is used.

14 Half - nut lever bearing - put a few drops of No. 10 motor oil or equivalent in the ball spring oil hole every time the lathe is used.

15 Thread dial-once a week put a few drops of No.10 motor oil or equivalent around the rim of the top of the thread dial.

16 Wipers (front and back) - saturate the felts in the four wipers, located on the carriage with oil every time the lathe is used.

17 Cross slide screw - Put a few drops of No. 10 motor oil or equivalent in the oil hole above the front cross slide screw bearing after removing the small screw. Replace the screw. This should be done every time the lathe is used. Clean the cross slide screw regularly with a small stiff brush. Oil the screw threads by running the compound rest back and forth.

18 Cross feed gears - put a few drops of oil in the oil hole above the cross feed screw after removing the small screw. Replace the screw. This should be done every time the lathe is used.

19 Cross slide ways - clean regularly and apply a liberal quantity of No. 10 motor oil or equivalent to the ways whenever the lathe is used.

20 Compound slide screw - every time the lathe is used put a few drops of No. 10 motor oil or equivalent in the oil hole on top of compound rest and above thecompound screw bearing.

21 Compound slide ways - clean regularly and apply a liberal quantity of No. 10 motor oil or equivalent to the ways whenever the lathe is used.

22 Lead screw - about once a month clean the lead screw threads with kerosene and a small stiff brush and apply a small amount of No. 10 motor oil or equivalent.

23 Rack (on bed, under front way) - about once a month apply a small amount of cup grease to the rack after cleaning with kerosene and a small stiff brush.

24 Lead screw bearing (right end of lathe) - put a few drops of No. 10 motor oil or equivalent in the oil hole on top of the bearing every time the lathe is used.

25 Place a few drops of oil between the handwheel and screw bearing when ever using lathe.

26 Tailstock centre lubricant - fill the small cup on the tailstock with a mixture of white lead and oil and apply to the tailstock center whenever turning between centres. If white lead is not available, used a liberal amount of cup grease on the center.

27 Tailstock ram - keep the outside surface of the tailstock ram well oiled.

28 Lathe bed ways - keep the bed ways oiled at all times with No. 10 motor oil or equivalent and free from chips. Wipe off the ways before using and cover with fresh oil. Always leave a generous film of oil on the ways when the lathe is not in use. The lathe should be completely covered when not in use. During all grinding operations cover bed ways with canvas or carboard.

Keep all the lathe bearing surfaces perfectly clean. Dirt is the natural enemy of accurate lathe work.

Lubrication points on lathe

Objectives: At the end of this lesson you shall be able to
- **describe the lubrication points**
- **brief the schedule of lubrications.**

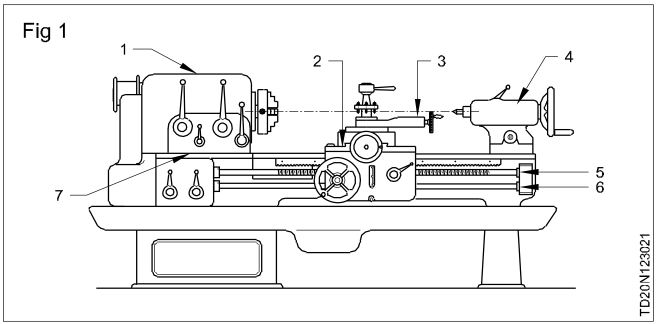
Lubrication of machines and equipments

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.

The lubrication points of lathe as shown in Fig 1.

Type of lubricant, period of lubrication and method of lubrication.



1 Head stock gear box filling the oil twice a year gear box oil.

2 Cross slide oiling daily by gun oily.

3 Top slide oiling daily by gun oily.

4 Tailstock oiling daily by gun oiler.

5&6 Lead screw and feed rod bearing grease by grease gun

7 Bed ways oiling daily by gun oiler use lubrication oil.

Lever position of lathe

Objectives: At the end of this lesson you shall be able to
- **describe the levers to be changed for different speeds**
- **brief to change the lever for feed and direction.**

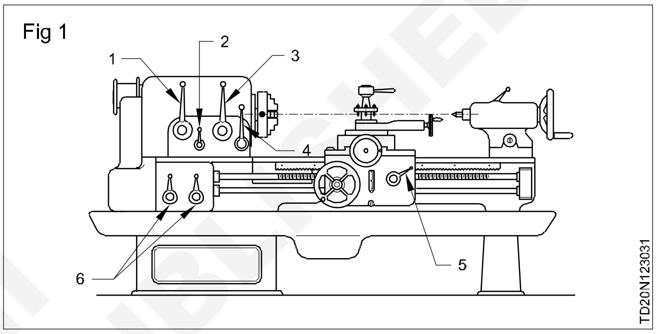
The lever available on lathe to change the different speed, feed and direction are shown in Fig 1.

Lever 1&3 to get the different spindle speed lever 2 is used to change the direction of the spindle.

Lever 4 Start and stop lever

Lever 5 Used to engage lead screw for thread cutting and disengaged feed rod for automatic feed.

Lever 6 Used to engage according to pitch of thread while thread cutting.



Introduction to Lathe

Objectives: At the end of this lesson you shall be able to
- **state the different types of lathes and their uses**
- **state the parts and their function**
- **state the method of specifying a centre lathe.**

Lathe introduction

Lathe is a machine widely used for wood works and machining of metal parts. Lathe is a machine which turns the work piece against a machine tool. The lathe is used for facing, turning, knurling, taper cutting, threading, gear cutting and many other metal and wood works.

Lathes are classified depending upon their application and functionality.

Light duty lathe - These machine find their application in automobile, electronic, electrical industries and are manufactured from quality tested raw materials.

Medium duty lathe - These machines are powerful than the light duty lathes and can work on bigger work pieces and have more strength than the light duty lathes.

Heavy duty lathe - these machines are manufactured from hightest grades of materials like iron and steel. They are designed for high precision heavy duty operations.

All geared lathe - In all geared lathe, all the rotating components of the machines are driven by the same source at different speeds by using gears to perform various operations.

Imported lathe - imported lathes are high quality lathes used for high precision operations.

Depending upon the modes of operation, the lathe can be classified as

Manual lathe - In these lathes, the tool handling is done manually and so the precision of work also depends upon the skill of the person handling the machine.

CNC lathe - CNC lathes are completely automated lathes. We just have to feed the instructions into the computer and the lathe will perform the operations according to the data fed to the computer.

Types of lathe

1 Speed lathe
   a Wood working
   b Centering
   c Polishing
   d Spinning
2 Engine lathe (or) centre lathe
   a Belt drive
   b Individual motor drive
   c Gear head lathe
3 Bench lathe
4 Tool room lathe
5 Capstan and turret lathe
6 Special purpose
   a Wheel lathe
   b Gap bed lathe
   c T-lathe
   d Duplicating lathe
7 Automatic lathe
8 CNC Lathe

Speed lathe

- The speed lathe has been so named because of very high speed of the head stock spindle.
- It consists of head stock, tailstock and tool post mounted on adjustable slide.
- Tool is fed into the work by hand control.
- It has no gear box lead screw and carriage.
- Different speeds are obtained by cone pulley (1200-3600rpm)
- Wood working, spinning, polishing, centering operations can be performed.

Engine lathe (or) centre lathe

- The term engine is because of that early lathes were driven by steam engine.
- It consists of basic parts like bed, head-stock and tail stock but head-stock is more robust and has additional drive mechanism for multiple speeds.
- Engine lathe can feed cutting tool both in cross and longitudinal directions with the help of carriage, feed rod, and leadscrew.
- Belt drive lathe gets power from an over head line shaft equipped with speed cone and one or more back gears.
- Individual motor driven lathe gets power from individual motor.
- A geared head lathe gets its power from constant speed motor and all speed changes are obtained by shifting various gears located in the headstock.

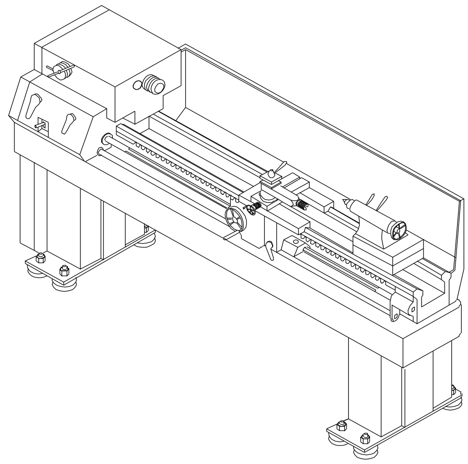
Bench lathe

- It is mounted on bench and has the same features like engine lathe.

Tool room lathe (Fig 1)

- It has the same features like engine lathe and has very low to high speed up to 2500rpm.

Fig 1



- It has taper turning attachment, draw in collet attachment, thread chasing dial, relieving attachment, steady and follower rest, pump for coolant.
- Used for precision work on tools, dies, gauges.

Capstan and turret lathe (Fig 2 & 3)

- Wheel lathe is used for finishing the journal and turning the thread on locomotive wheels.
- The gap bed lathe can accommodate the jobs having extra diameter.

Fig 2

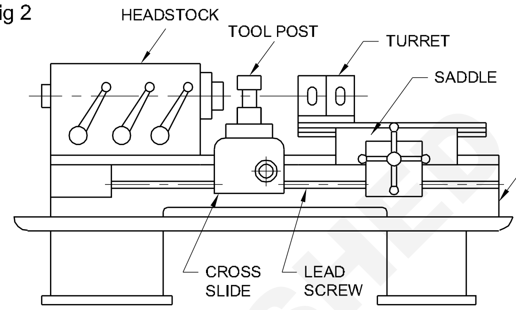
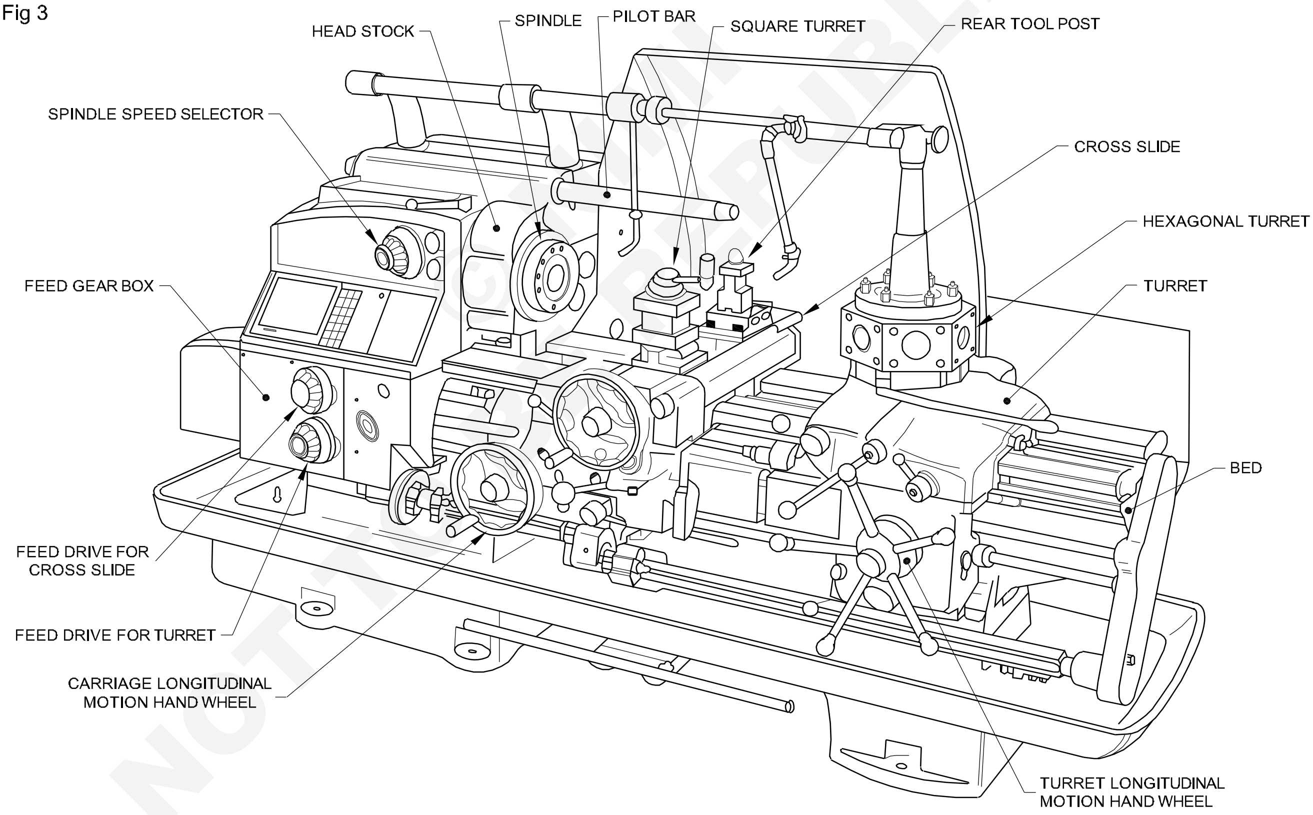


Fig 3



- T- lathe is intended for machining the rotors for jet engines, axis of bed is right angles to the axis of head stock spindle.
- Duplicating lathe is used for duplicating the shape of given template using mechanical or hydraulic system.

Special purpose lathe

- These are high speed, heavy duty, mass production lathes with complete automatic control.
- Once the tools are set and machine is started it performs automatically all the operations to finish at a time.
- Change of tools, speeds and feeds can be done automatically, operator can run 5 to 6 machines at a time.

- **CNC Lathe** (CNC)
- Complex shapes machined easily.
- High production rate.
- Accuracy and repeatability is achieved.
- Less operation skill and involvement.
- Reduced space.

Centre lathe parts

The following are the main parts of a lathe.

Headstock

Tailstock

Carriage

Cross-slide

Compound slide

Bed

Quick change gearbox

Legs

Feed shaft

Lead screw

Head stock

The following are the functions of a headstock. (Fig 1)

- Provide a means to assemble the work-holding devices.
- Transmit the drive from the main motor to the work.
- Accommodate shafts, gears and levers for a wide range of varying work speeds.
- Ensure arrangement for lubricating the gears, shafts and bearings.

Tailstock

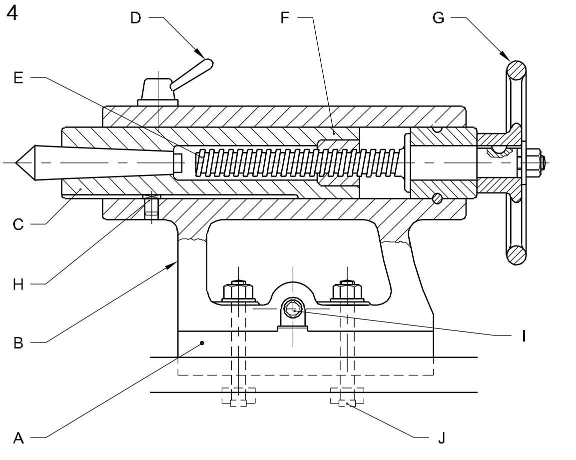
It is a sliding unit on the bed-ways of the lathe bed. It is situated on the right hand side of the lathe. It is made in two parts namely the 'base' and the 'body'. The base bottom is machined accurately and has 'V' grooves corresponding to the bed-ways. It can slide over the bed and can be clamped at any position on the bed by means of the clamping unit. The body of the tailstock is assembled to the base. Graduations are marked on the rear end of the base and a zero line is marked on the body.

When both zero lines coincide, the axis of the tailstock is in line with the axis of the headstock.

The body and base are made out of cast iron. The parts of a tailstock are: (Fig 4)

- base (A)
- body (B)
- spindle (barrel) (C)
- spindle locking lever (D)
- operating screw rod (E)

Fig 4



- operating nut (F)
- tailstock hand wheel (G)
- key (H)
- adjusting screws (J)
- clamping unit (I)

Functioning of a tailstock: By rotating the hand wheel, the barrel can be moved forward or backward. The barrel can be locked in any required position. The hollow end of the barrel at the front is provided with a Morse taper to accommodate the cutting tools with a taper shank. Graduations are sometimes marked on the barrel to indicate the movement of the barrel. With the help of the adjusting screws, the body can be moved over the base laterally, and the amount of movement may be read approximately referring to the graduations marked. This arrangement is to offset the centre of the tailstock as required for taper turning.

Purpose of the tailstock

- To accommodate the dead centre to support lengthy works to carry out lathe operations.
- To hold cutting tools like drills, reamers, drill chucks provided with taper shank.
- To turn external taper by offsetting the body of the tailstock with respect to the base.

Lathe bed

Functions of a lathe bed

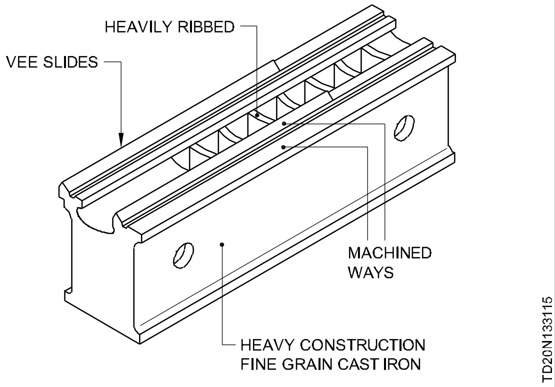
The functions of a lathe bed are:

- to locate the fixed units in accurate relationship to each other
- to provide slideways upon which the operating units can be moved.

Constructional features of a lathe bed (Fig 5)

The lathe bed generally consists of a single casting. In larger machines, the bed may be in two or more sections accurately assembled together. Web bracings are employed to increase the rigidity. For absorbing shock and vibration, the beds are made heavy.

Fig 5

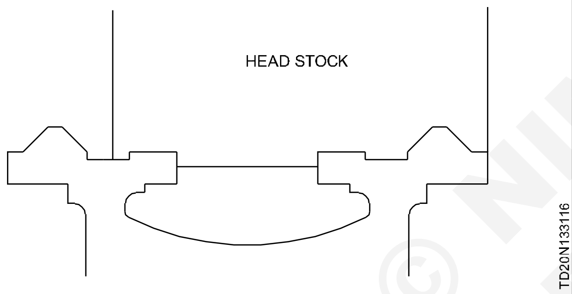


A combined swarf and coolant tray is provided on the lathes. This may be an integral part with the lathe bed.

The bed is generally carried by cast iron or welded sheet metal legs of box section. This provides the necessary working height for the lathe. Very often the electrical switchgear unit and the coolant pump assembly are housed in the box section of the legs at the headstock end.

Bed-ways (Fig 6)

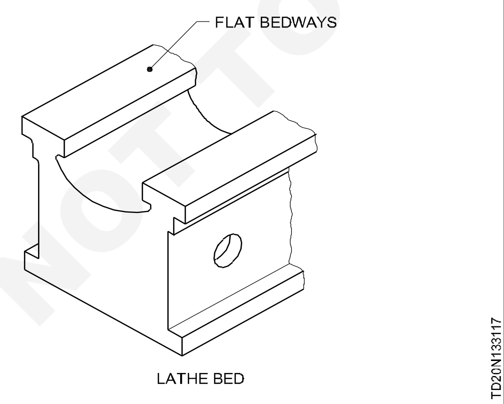
Fig 6



The bed-ways or slideways assist in accurate location and sliding of the accessories/parts mounted on this. The bed-ways are of three types.

- Flat bed-ways (Fig 7)

Fig 7



- 'V' bed-ways (Fig 8)
- Combination bed-ways (Fig 9 & 10)

Normally the bed-ways stop at a distance away from the headstock with a gap at this point. This enables to mount larger diameters of works.

Fig 8

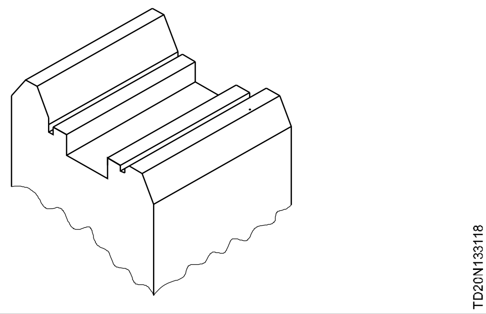
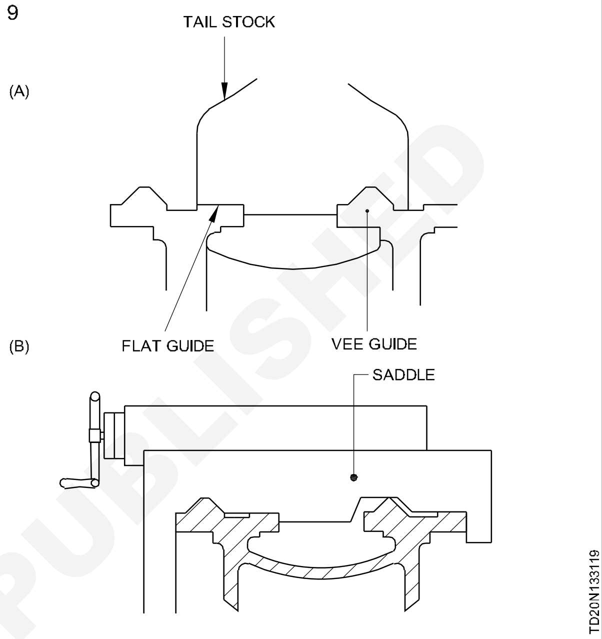


Fig 9

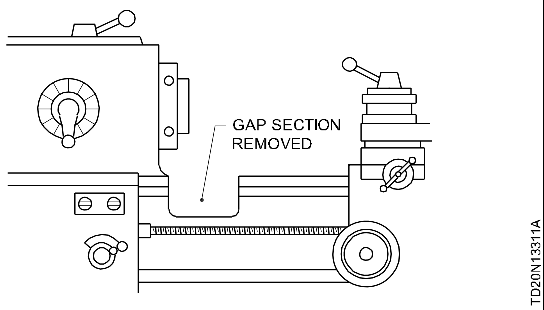


Some lathes have detachable section of the bed, which can be fitted when desired, to enable the saddle to operate close to the headstock. (Fig 10)

Some bed-ways are fine finished by grinding. Some lathes have their bed-ways hand-scraped. Some have their bed-ways hardened and ground. The wear-resisting qualities of bearing surfaces are improved by employing chilled iron castings.

The beds are mostly made up of close grained grey cast iron.

Fig 10



Lathe carriage

Carriage (Fig 11)

The carriage is the lathe feature that provides the method of holding and moving the cutting tool. It consists of two major parts.

- The apron
- The saddle

The apron (Fig 11)

The apron is bolted to the front end of the saddle. It contains the mechanism for moving and controlling the carriage.

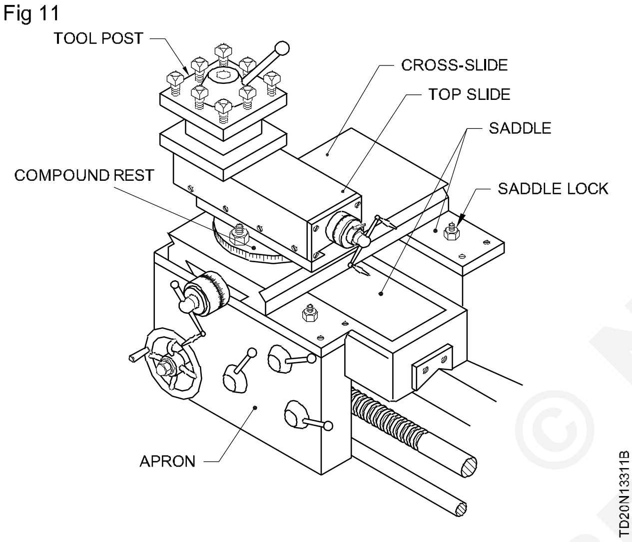
The saddle (Fig 11)

The saddle is that part of the carriage that fits the bed slideways and moves along the bed between the head and the tailstock.

It is an assembly consisting of the following parts.

- Cross-slide
- Compound rest
- Top slide
- Tool post

Fig 11



Cross-slide (Figs 11 & 12)

This part is mounted on the top of the saddle base and it moves along the saddle base in a direction perpendicular to the bed. This movement is accomplished by means of a screwed spindle and hand wheel.

Compound rest (Figs 11 & 12)

This is fitted on the top of the cross-slide and may be swivelled horizontally, clockwise or anticlockwise through 360°.

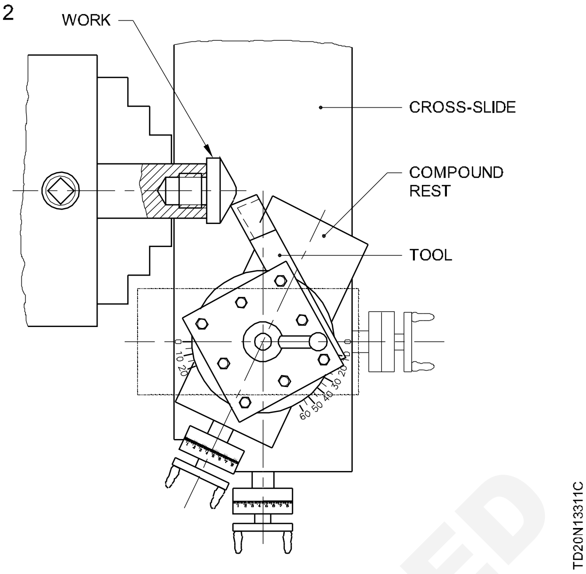
Top slide (Fig 11)

This part is connected to the compound rest by means of a screwed spindle and it has a short travel on the compound rest. It provides a means of supporting the tool post which holds the cutting tool.

By swivelling the compound rest the top slide may be set to a desired angle to the cross-slide and fed at that angle, and turn tapers.

In the normal case the compound rest is set so that the top slide is at right angles to the cross-slide and in this position the setting angle is 0°.

Fig 12



Feed shaft (Fig 13)

Usually the carriage is moved manually by means of a hand wheel. But the carriage can also be moved by power (automatic feed). The power for automatic feed comes from the headstock and is transmitted to the carriage through the feed shaft.

Lead screw (Fig 13)

Centre lathes, equipped with power feed, also have a provision for screw cutting. A special threaded spindle called lead screw, mounted on the front of the lathe bed and driven through the quick change gearbox, helps to cut screw threads.

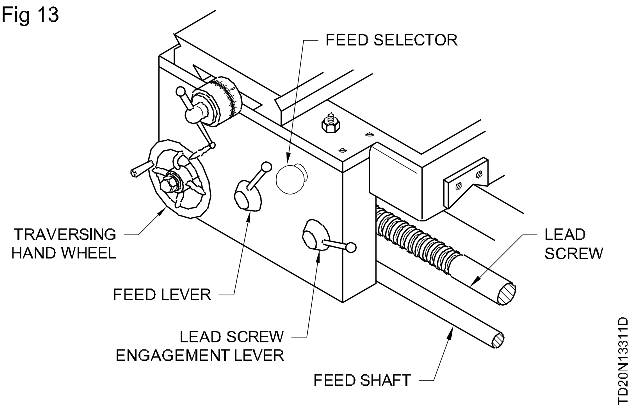
In some small lathes the lead screw and feed shafts are combined.

Both the lead screw and the feed shaft pass through the apron of the carriage. Controls on the apron enable the feed shaft or the lead screw to be connected to the carriage at the operator's will.

Feed lever

This lever is used to engage and disengage the automatic feed mechanism which provides automatic feeding for both facing and turning operations.

Fig 13



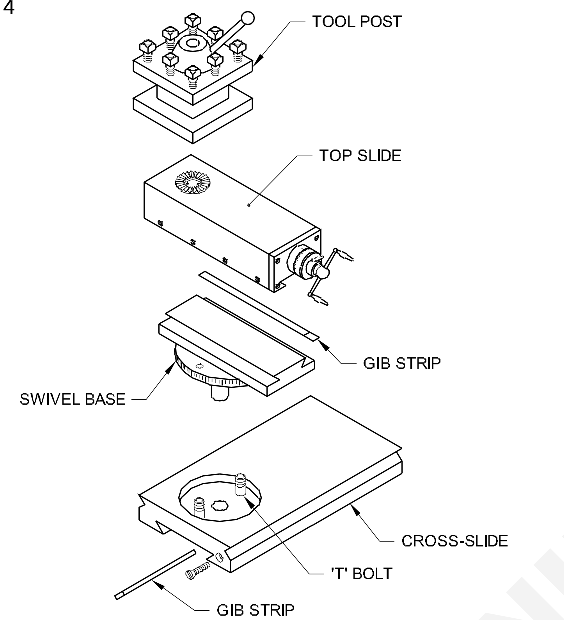
Tool posts

The tool post fitted on the top slide holds and supports the tool firmly. (Fig 14)

The commonly used types of tool posts are:

- American type tool post or single way tool post
- indexing type tool post or square tool post
- quick change tool post.

Fig 14

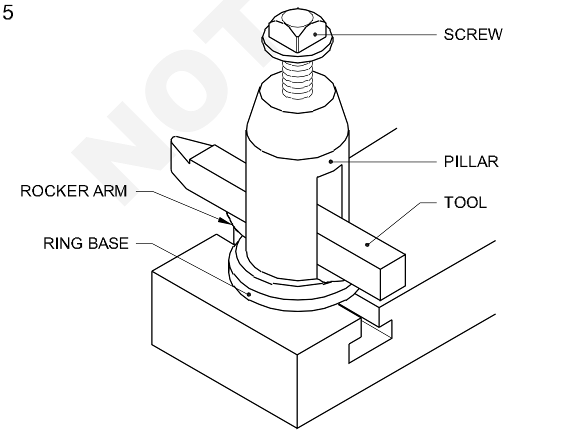


Single way tool post (Fig 15)

It consists of a circular tool post body and a pillar with a slot for accommodating the tool or tool-holder. A ring base, a rocker arm (boat piece) and a tool-clamping screw complete the assembly of this type of tool post.

The tool is positioned on the boat piece and is clamped by the screw. The centre height of the tool tip can be adjusted with the help of the rocker arm and the ring base. Only one tool can be fixed in this type of tool post. The rigidity of the tool is less as it is clamped with only one screw.

Fig 15



AMERICAN (PILLAR) TYPE TOOL POST

Indexing type tool post (Fig 16)

It is also called a square tool post or a four-way tool post. Four tools can be fixed in this type of tool post, and any one of them can be brought into the operating position. The indexing may be manual or automatic.

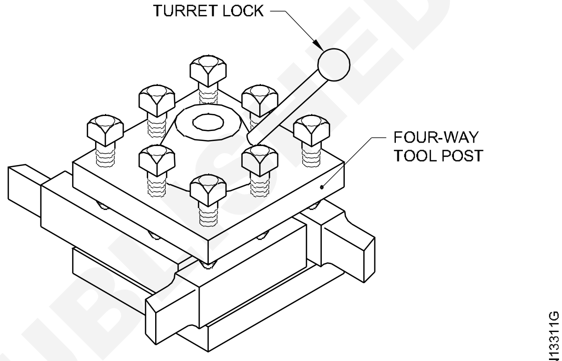
The advantages are as follows.

Each tool is secured in the tool post by more than one screw and, therefore, the rigidity is more.

Frequent changing of the tool for different operations need not be done as all the four tools can be clamped at the same time.

The disadvantage is that skill is required to set the tools, and it takes more time to set to the centre height.

Fig 16

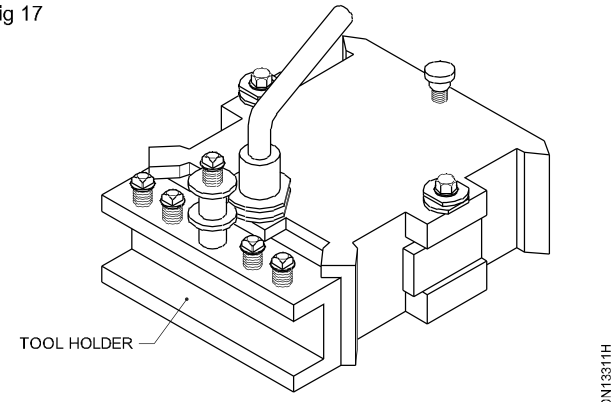


INDEXING TYPE TOOL POST

Quick change tool post (Fig 17)

Modern lathes are provided with this type of tool post. Instead of changing the tools, the tool-holder is changed in which the tool is fixed. This is expensive as a number of tool-holders may be needed to preset various tools. But it can be set to the centre height easily, and has the best rigidity for the tool.

Fig 17



QUICK CHANGE TOOL POST

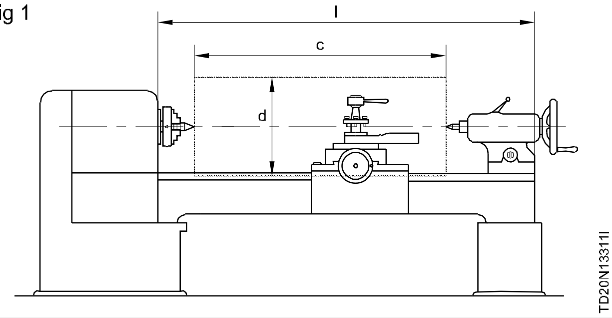
Centre lathe specification

The size of a lathe is specified by (Fig 18)

- the length of the bed (l)
- the maximum diameter (swing) of the work that can be turned (d)
- the length between centres (c) and

- the pitch of the lead screw.
- the maximum diameter of bar that passes through hole of the head stock spindle.
- number of spindle speeds and feeds available.
- power input.
- floor space requried.

Fig 1



Safety to be observed while working on a lathe

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

- **state the precautions to be observed before starting work on a lathe, during work and after.**

Before starting the work

Ensure that the lubricating system is functioning.

The mating gears should be in proper mesh and the power feed levers are in neutral position.

The work area should be clean and tidy.

The safety guards should be in place.

During work

Never try to stop a rotating chuck with your hand. A rotating chuck is dangerous.

Switch off the machine before making any adjustment on the lathe.

It is dangerous to leave the chuck key in the chuck. Remove it immediately after use. (Fig 1)

Fig 1

Single point tools are sharp and dangerous. Be extra careful when using them.

Chips are sharp and dangerous. Never remove them with your bare hands. Use a chip rake or brush.

You must always know where the emergency stop switch.

After work

Clean the lathe with a brush and wipe with cotton waste.

Oil the bedways and lubricating points.

Clean the surroundings of the lathe, wipe the dirt and coolant and remove the swarf.

Stop lathe before taking measurements of any kind.

Follow job specifications for speed and depth of cut for material being turned.

Do not lean on machine. Stand erect, keep your face and eyes away from flying chips.

Do not make adjustments while the machine is operating unit unitl the machine has come to complete stop. Do not leave lathe unattended while it is running.

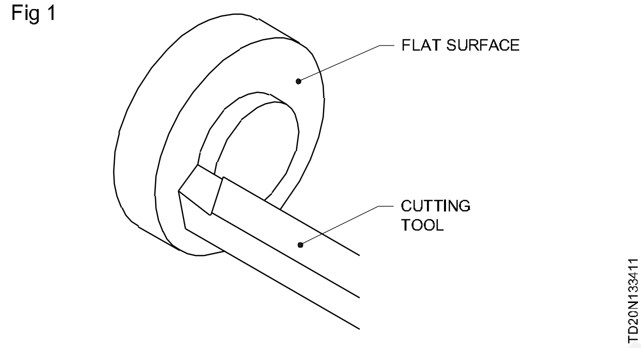
Lathe operations - facing

Objectives : At the end of this lesson you shall be able to
- **list the various types of operations of lathe machine**
- **state the meaning of each operation**
- **brief the purpose, defects of each operations.**

Facing

This is an operation of removing metal from the work-face by feeding the tool at right angles to the axis of the work. (Fig 1)

Fig 1



Purpose of facing

- To have a reference plane to mark and measure the step lengths of the work.
- To have a face at right angle to the axis of the work.
- To remove the rough surface on the faces of the work and have finished faces instead.
- To maintain the total length of the work.

Facing may be rough or finish facing. Rough facing is done to remove the excess metal on the face of the work by coarse feeding with more depth of cut, leaving sufficient metal for finishing.

Rough facing is done by feeding the tool from the periphery of the work towards the centre of the work. Finish facing is the operation to have a smooth face by removing the rough surface produced by the rough facing.

Finish facing is done by feeding the tool from the centre of the work towards the periphery. (Figs 2a and 2b)

Rough facing is done by choosing a spindle r.p.m. according to the average diameter of the work, with the recommended cutting speed, coarse feed and more depth of cut.

Finish facing is done by choosing a cutting speed about twice that of the cutting speed for roughing, with a fine feed rate of 0.05 mm approximately and with a depth of cut of not more than 0.1 mm.

The following are the defects found in facing a work. (Fig 3)

step lengths of the work.

Fig 2

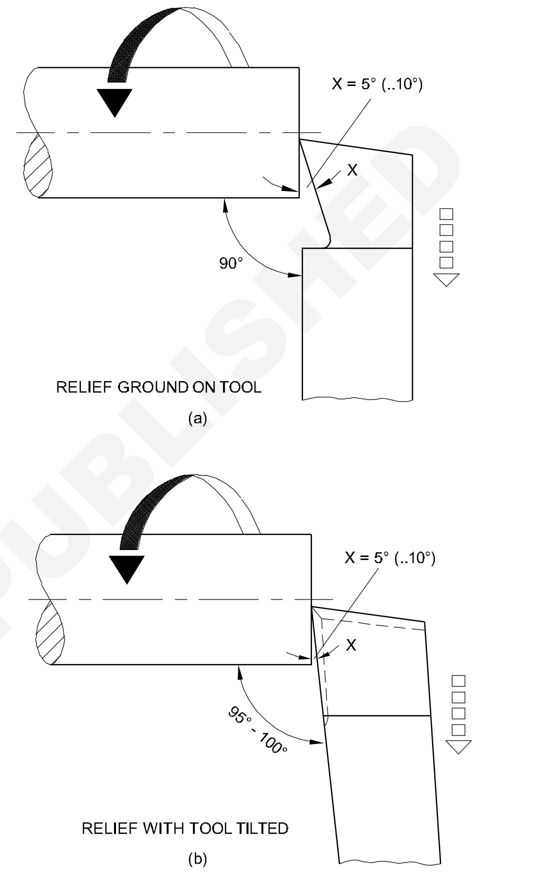
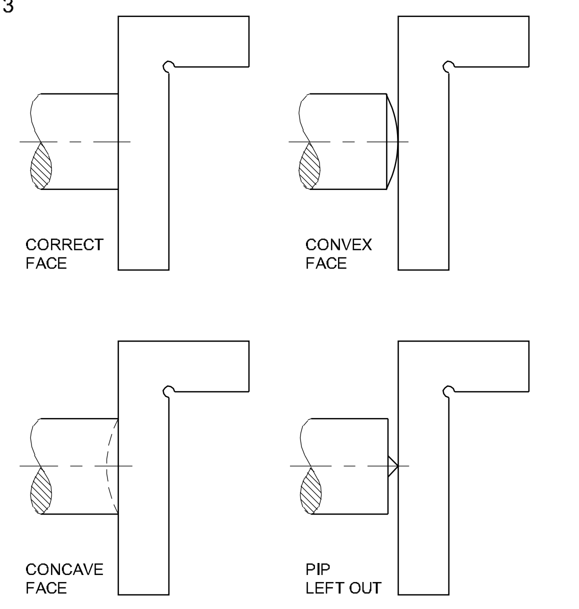


Fig 3



A concave face

This is caused by the tool digging into the work during the feeding as the tool is not clamped rigidly. By clamping the tool rigidly with minimum overhang, this defect can be avoided.

A convex face

This is caused by the blunt cutting edge of the tool and the carriage not being locked. To avoid this defect, re-sharpen the tool and use it; also lock the carriage to the bed of the lathe.

A pip left in the centre

This is due to the tool not being set to the correct centre height. By placing the tool to the correct centre height, this defect can be avoided.

Plain turning & Step turning

Plain turning (Parallel turning)

This operation involves removal of metal from the work and it has a cylinder for the full travel of the tool on the work, keeping the same diameter throughout the length.

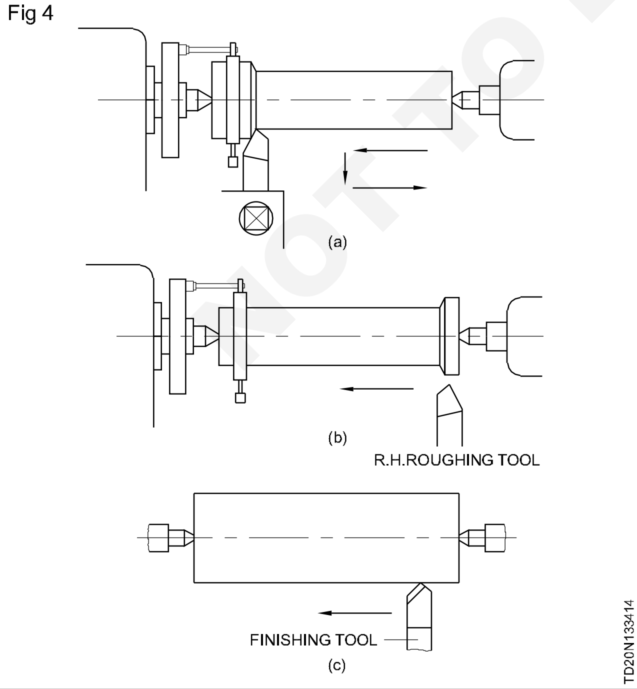
Plain turning is done in two stages.

- Rough turning, using roughing tool or knife tool. (Figs 4a and 4b)
- Finish turning using a finishing tool. (Fig 4c)

It is done after rough turning, and it aims to bring the size of the work within the specified accuracy and with better surface finish. The tool to be used in this case will be a finish turning tool. Finish turning is also carried out from the tailstock end towards the headstock end.

The spindle speed is calculated according to the material being turned, diameter of the work, the tool material and the recommended cutting speed diameter of the works.

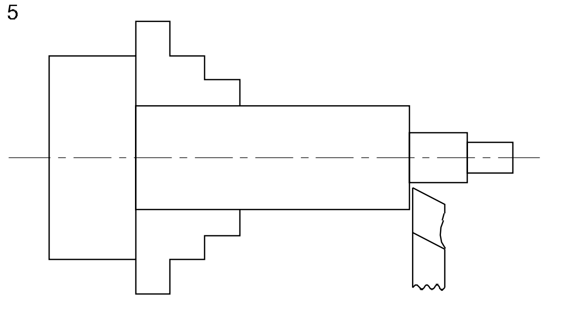
Fig 4



Step turning

Step turning is an operation performed on lathe machine where the excess material is removed from the work piece to obtain various steps at different diameters. (Fig 5)

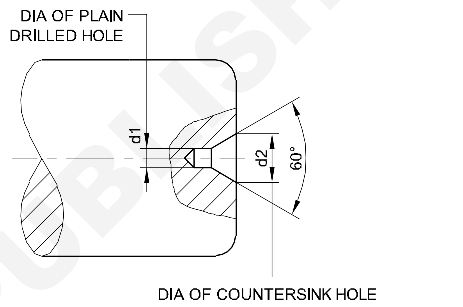
Fig 5



Lathe Operations - Centre Drilling

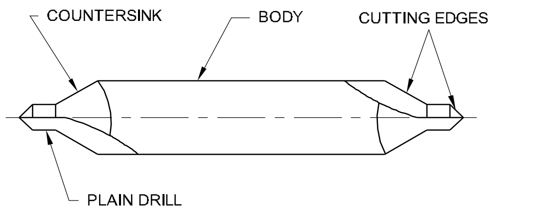
Centre drilling (Fig 6)

Fig 6



It is an operation of drilling and countersinking a hole on the face of the work, and on the axis of the work.It is done by a cutting tool known as centre or combination drill held in a drill chuck. The drill chuck is mounted in a tailstock spindle and the feeding of the drill to the work is done by rotating the tailstock hand wheel. The spindle speed for the work rotation is calculated, taking into consideration the plain drilling diameter and the recommended cutting speed for the drilling.(Fig 7)

Fig 7



Defects in centre drilled holes

The two major defects in centre drilling are:

- Insufficient depth of plain drilled portior (Fig 8)
- centre drilling done to deep. (Fig 9)

Fig 8

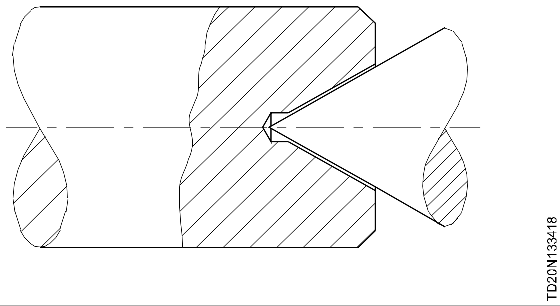
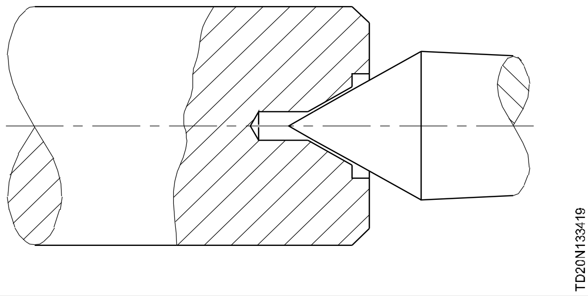


Fig 9



The first depth results in making the tip of the centre to contact the work surface, and the conical portion of the centre does not have any contact with the bearing surface of the centre drilled hole. Undue friction and overheating will be noticed which will damage the tip of the centre. Sometimes, breakage is also possible and the broken part of the centre may get welded to the centre hole.By feeding the centre drill upto 3/4 th of the 60° countersink, this defect is avoided.

When the centre drill feeding is too much, a plain drilled portion by the body of the centre drill will be formed at the nose of the bearing surface of the centre hole, and the area of contact between the bearing surface and the work - supporting centre will be the only point of contact, as illustrated in Fig 10. This will not provide proper support to the work and any operation if carried out, may result in dimensional inaccuracy, chatter and poor surface finish.

To rectify this defect, face the work, work permits, and feed the centre drill to the recomended length.

Centre drills

It is made of high steel and is cylindrical in shape.At both the ends, it has a plain drill and countersink as its integral part. It is hardened and ground. It is available in standard sizes.

Classification as per Indian Standard

Indian standard classifies centre drills into three types.

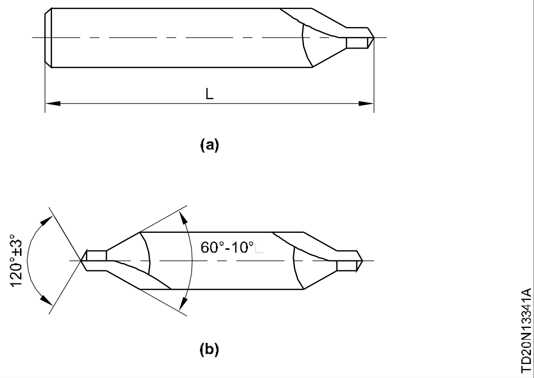
They are Type A, Type B and Type D.

The differences lies in the formation of the countersink by each type

Uses and specifications

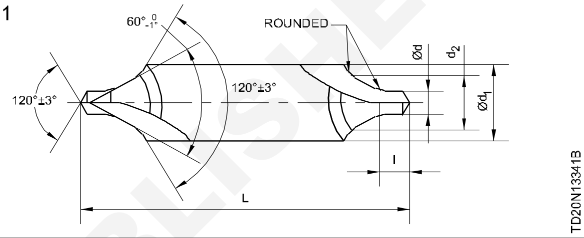
Type ‘A’ centre drill is used to produce centre holes with plain drilled portion and countersink.It is designated as centre drill A.

Fig 10



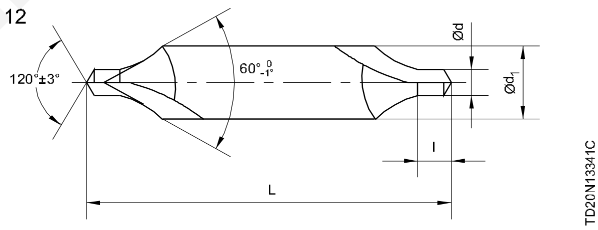
Type ‘B’ centre drill is used to produce a centre hole with a plain drilled portion and a countersink, and has a further conical portion to form additional countersinking to protect the centre hole. (Fig 11)

Fig 11



The third type, ‘D’ is designated as centre Drill R 1.6 x 4.0 IS: 6710.This also has provision to provide a protected centre hole. This has an enlarged radius, machined along with the countersinking portion: (IS: 6710) (Fig 12)

Fig 12



Methods of centre drilling

Centre drilling can be done on a

- drilling machine
- lathe
- centering machine.

The selection of a particular method of centering depends on the size and shape of the component.

A drill chuck is used to hold the centre drill, Sometimes special holders/collets are also used. In mass production, the raw material ends are centered using a centre drillling machine.

While central drilling ensure the following

Choose the right size and type of centre drill according to the diameter of the work.

Select the recommended cutting speed and calculate the spindle r.p.m. taking into consideration the small plain drilling diameter of the centre drill.

The job diameter does not have influence on the spindle speed

The face of the work should be free from any ‘pip’.

The alignment of lathe centres is perfect

The drill chuck used has no damages on the holding fingers and also on the morse taper shank.

All the three jaws of the chuck act simultaneously.

The overhang of the tailstock barrel should be as minimum as possible.

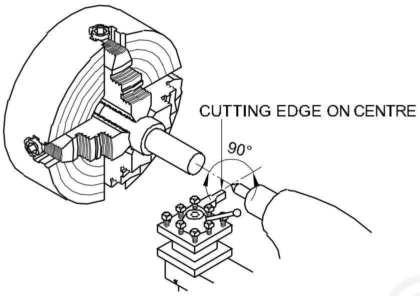
If any one of the above points is not observed, htere are possibilities to have a centre hole with defects,and breakage of centre drill might result.

Parting off operation: Parting off or cutting off is the operation of severing a finished part from the rough or finished stock.

Setting of parting tool

Set the parting tool exactly on the centre with as little back-rake as possible. (Fig 13)

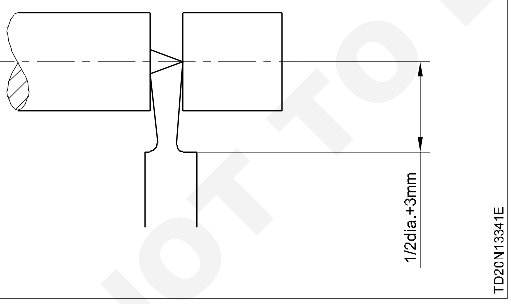
Fig 13



CORRECT CUTTING EDGE POSITION

Adjust the parting off tool so that it extends one half of the diameter of the work plus about 3mm for clearance from the tool-holder (Fig 14)

Fig 14



If the cutting tool is too high, it will not cut through the workpiece. If it is too low, the work may be bent and the cutting tool damaged.

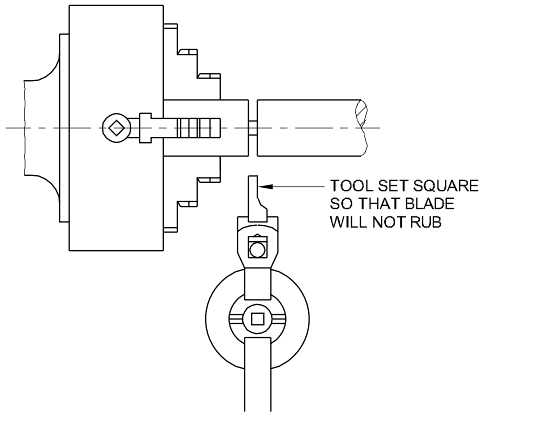
Procedure

Select the correct type of tool for a specified job.

Hold the work with the minimum overhang in a chuck.

Set the tool square with the work so that it does not rub against the sides of the groove, as it is fed into the work. (Fig 15)

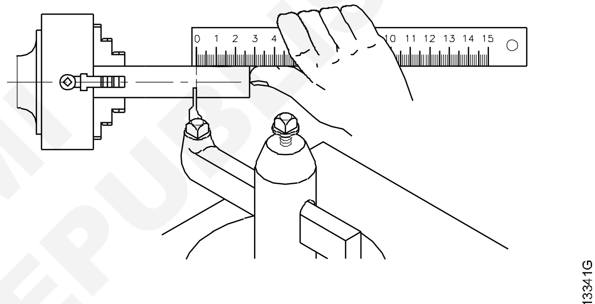
Fig 15



Set the spindle speed to half the speed for turning.

Move the carriage so that the right hand side of the blade is at the point where the work is to be cut off. (Fig 16)

Fig 16



PARTING TOOL POSITIONED TO CUT OFF A DEFINITE LENGTH OF WORKPIECE

Start the lathe and feed the tool steadily into the work using the cross-slide handle.

Continue to feed the tool into the work until the part is severed.

Precautions

The work should protrude from the chuck jaws, sufficiently enough to permit the cut to be made as close as possible to the chuck jaws.

The work must always be held securely in a chuck or a collet.

If the workpiece is held between centres, it may bend or break and fly aut of the lathe during parting off. (Fig 17)

Use a right hand offset tool-holder. (Fig 18)

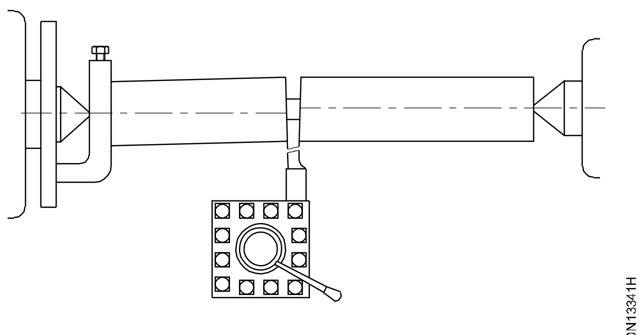
A work having more than one diameter should be gripped on the larger diameter while parting.

Intermittent feed tends to dull the tool’s cutting edge.

Heavy feed causes jamming and tool breakage.

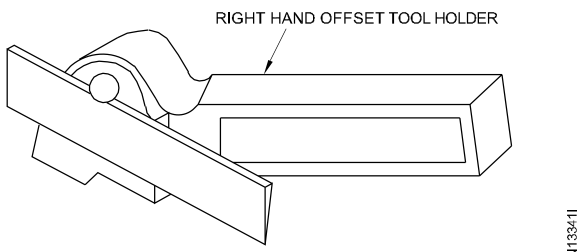
Use sufficient coolant on steel. Brass and cast iron should be cut off dry.

Fig 17



INCORRECT METHOD OF PARTING

Fig 18



INSERTED-BLADE TYPE PARTING TOOL

Make sure the saddle is locked during the entire operation.

Reduce the rate of feed, when the work is almost cut off.

While parting off long work, it should be supported with the tailstock centre.

If the machine is in good condition, the automatic cross feed may be used.

When the tool has penetrated to about the depth of its width, withdraw it and move it sideways with the compound slide and feed again.

The above operation should be repeated frequently to minimise the tendency of the tool to dig in and cause trouble.

When the parting off operation is almost completed, hold the workpiece by hand to prevent it from falling, so that damage can be avoided.

Grooving

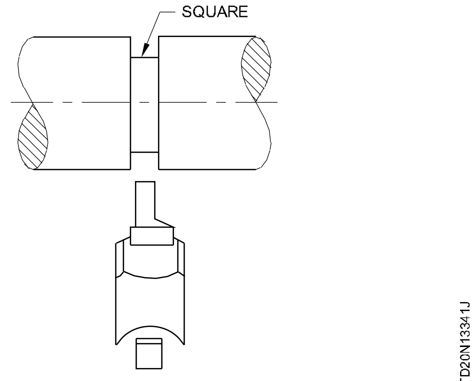
Grooving is the process of turning a grooved form or channel on a cylindrically turned workpiece. The shape of the cutting tool and the depth to which it is fed determine the shape of the groove.

Types of grooves

Square grooves

Square grooves are frequently cut at the end of a section to be threaded in order to provide a channel into which a threading tool may run. A square groove cut against a shoulder allows a matching part to fit squarely against the shoulder. (Fig 19)

Fig 19



When a diameter is to be finished to size by grinding, a groove is generally cut against the shoulder to provide clearance for the grinding wheel and to ensure a square corner.

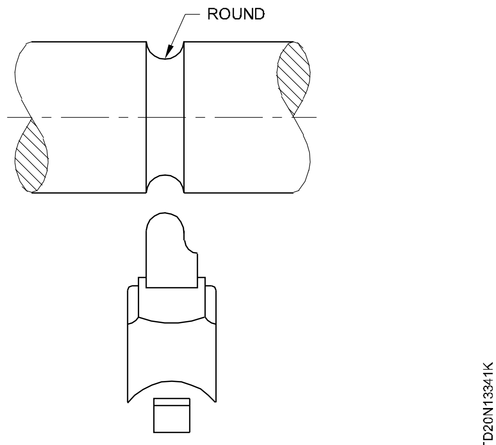
Square grooves are cut with a tool bit ground to the width of the square groove to be formed.

A square groove also serves the purpose of providing space for forks of shift levers in sliding gear assemblies.

Round groove

Round grooves serve the same purpose as square grooves. They are generally used on parts subjected to stress. The round groove eliminates the sharpness of the square corners and strengthens the part at the point where it tends to fracture. A tool bit with a round nose ground to the required radius is used to cut round grooves. (Fig 20)

Fig 20

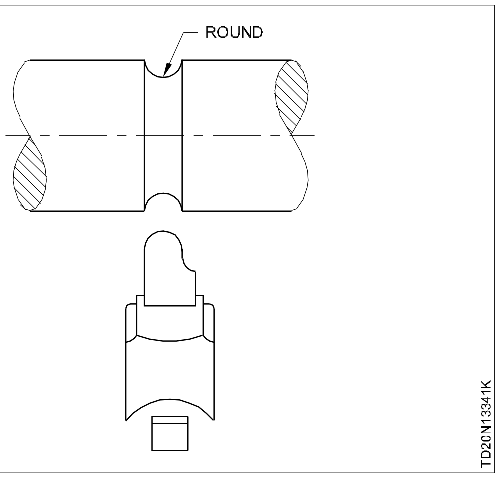


'V' Shaped groove

'V' shaped grooves are most commonly found on pulleys driven by `V' belts. The `V' shaped groove eliminates much of the slip which occurs in the other forms of the belt drive. A `V' groove may also be cut at the end of a thread to provide a channel into which the threading tool may run. (Fig 21)

A tool bit ground to the desired angle is used to cut a shallow `V' groove. Larger `V' grooves such as those found on pulleys should be cut with the lathe compound rest set to form each face of the groove individually.

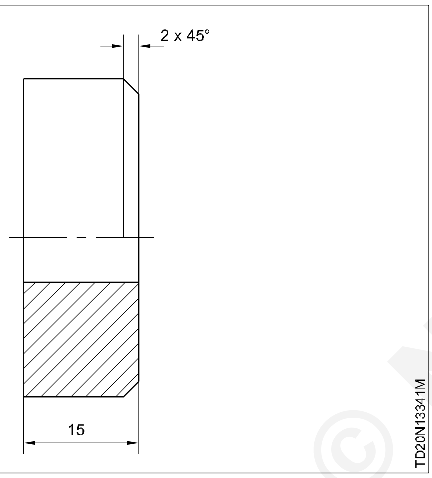
Fig 20



Chamfering (Fig 22)

It is the operation of bevelling the edge of a cylinder or a bore to a definite length.

Fig 22



Purpose of chamfering

The following are the purposes of chamfering.

- To remove burrs and sharp edges from the turned components to make their handling safe.
- To permit for easy assembly of mating components (a shaft and a hole).
- To avoid the formation of feather edges on the threaded parts.
- To provide better appearance.

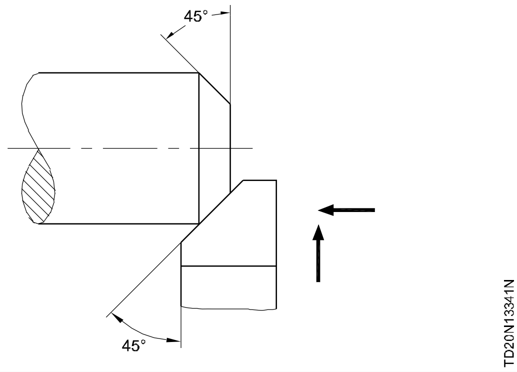
Methods of chamfering

Form tool method

Chamfering can be done with the help of a cutting tool, having its edges ground to the angle of the chamfer and by directly applying the tool on to the edge of the work. (Fig 23)

The tool is fed either longitudinally or crosswise. Depth is calculated by the graduated collars.

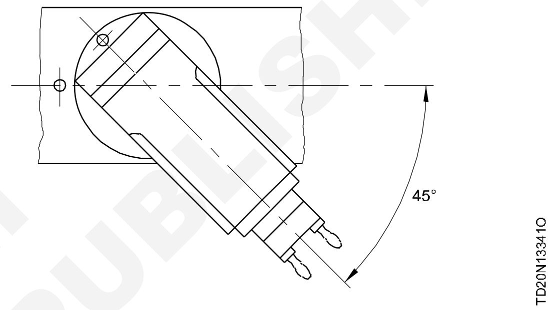
Fig 23



Compound slide method

Chamfering is also done by a tool fixed on the tool post, swivelling the compound slide to the angle of chamfer. Then the carriage is locked and the tool is fed by the top slide. (Fig 24)

Fig 24

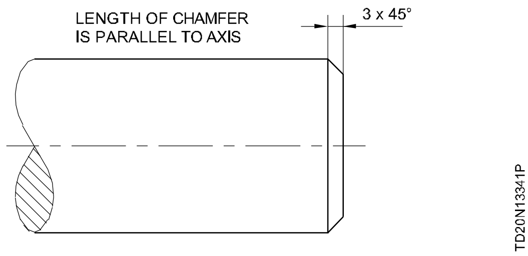


Filing method

This is a crude method, and is adopted only when the accuracy and finish are not a criterion. This is limited to external chamfer. The file is held on to the edge of the work to the angle of the chamfer and moved along the edge of the rotating work. A gentle pressure is applied until the required length of chamfer is achieved.

Length of chamfer is the distance measured parallel to the axis. (Fig 25)

Fig 25



Boring tool and boring

Objectives : At the end of this lesson you shall be able to
- **explain the different types of boring tools**
- **state the advantages of the different boring tools.**

Boring is the process of enlarging and truing an existing drilled or core hole with a single point cutting tool.

Necessity of boring a hole

- To enlarge a drilled hole larger than the drill size as drills are available in standard sizes only.
- To obtain concentricity of the hole.
- To maintain accuracy of the hole.
- To obtain better surface finish.
- To remove the error formed by drilling, and to facilitate the reaming operation.

Boring tools and holders

Boring is an internal operation performed on the drilled or core holes. The cutting edge of a boring tool is ground similar to the left hand plain turning tool. But the operation being performed is from right to left. (Fig 1)

Parts of a rough boring tool (Figs 1,2 & 3)

Types of boring tools

The following are the different types of boring tools.

- Solid forged tools
- Boring bars with bits

Fig 1

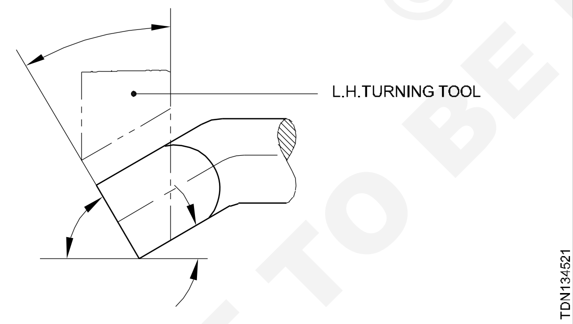


Fig 2

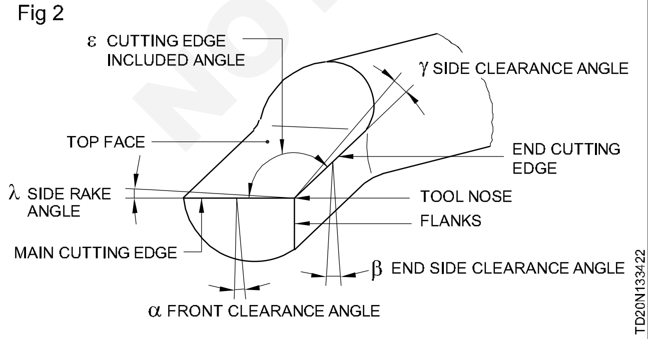
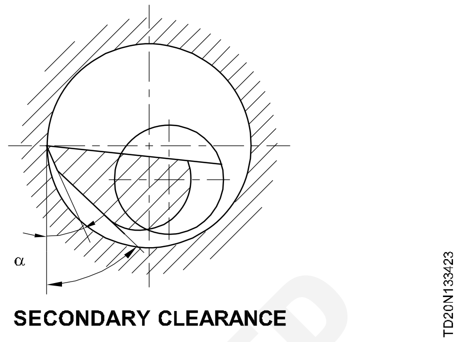
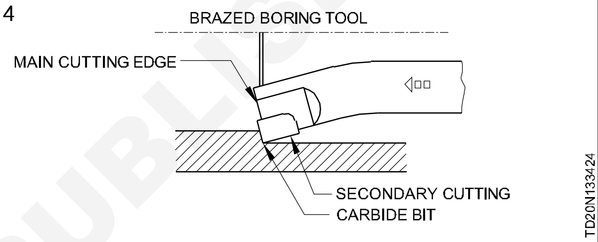


Fig 3



- Brazed tools (Fig 4)
- Throw-away bits inserted in special holders.

Fig 4

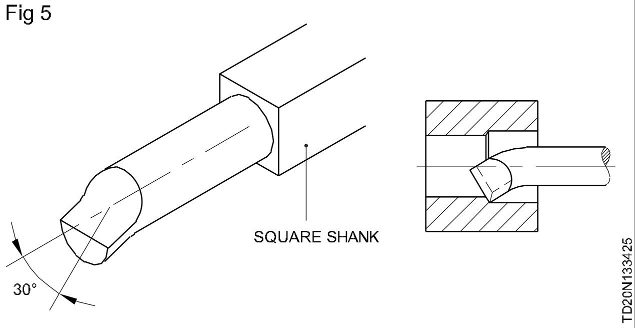


Solid forged tools (Fig 5)

The solid forged boring tool is generally made of high speed steel, with the end forged and ground to resemble a left hand turning tool. They are light duty tools and are used on small diameter holes. They are held in special tool holders which are mounted in the tool post.

Occasionally tungsten carbide or high speed steel tips are brazed to low carbon bars, for economy.

Fig 5

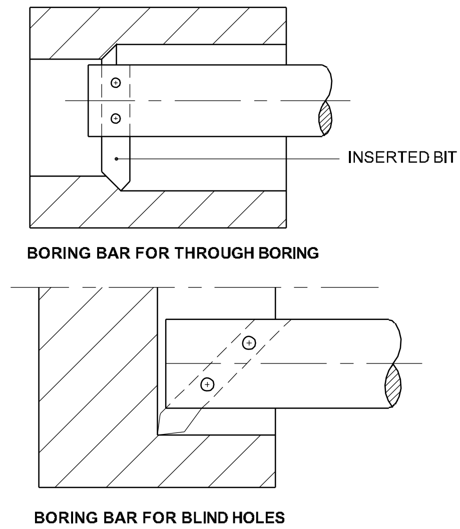


Boring bars with inserted bit (Fig 6)

The boring bar tool-holder is mounted in the tool post and is used for heavier cuts than those for the forged boring tool.

The square tool bits are set at angles of 30°, 45° or 90° in the broached holes in the bar.

Fig 6



The boring bars may be plain type or end cap type. The cutting tool of the plain type is held in position by a set screw. The cutting tool of the end-cap type is held in position by the wedging action of a hardened plug.

The round or square section tool bits may be inserted in boring bars, the size depending on the diameter of the bar.

The tool bit may be square to the axis of the bar for plain boring or at an angle for facing shoulder, or threading up to a shoulder.

The bar is held in a split or 'V' block holder.

The advantages of Different boring tools

Solid boring tools

Available with square and round shank.

Enables to mount on the tool post easily.

Re-grinding is easy.

As the tool is integral, alignment is easy.

Can be easily forged to the required shape and angle.

Boring bars and inserted bits

Used for heavy duty boring operation.

Used for deep boring operation.

Tool changing is faster, thereby re-sharpening time is avoided.

Cost is less because the boring bar is made out of low carbon steel.

Boring tools can be set square to the axis of the boring bar or at an angle very quickly.

Lathe cutting tools

Objectives : At the end of this lesson you shall be able to
- **explain difference between end-cutting and side-cutting tools**
- **state the features of each type**
- **explain the different shapes of cutting tools used for lathe operations**
- **state the characteristics and uses of different shapes of lathe tools.**

Cutting tools - classification

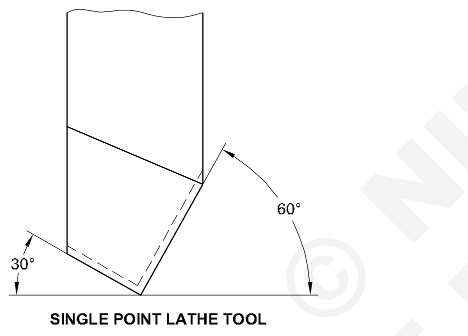
Cutting tools are classified as:

- single point cutting tools
- multi-point cutting tools
- form tools.

Single point cutting tools

The single point cutting tools have one cutting edge which performs the cutting action. Most of the lathe cutting tools are single point cutting tools. (Fig 1)

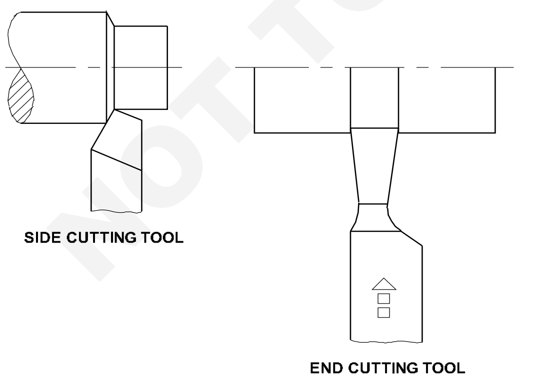
Fig 1



Single point cutting tools used on lathes may be grouped into:

- side-cutting tools
- end-cutting tools. (Fig 2)

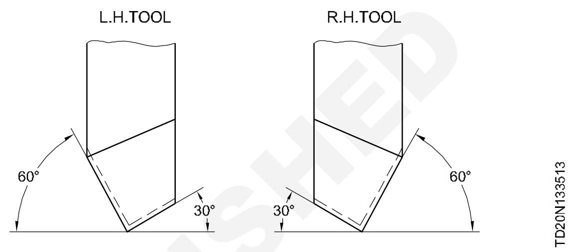
Fig 2



Side cutting edge tools have their cutting edges formed on the side of the cutting tool, and are used on lathes for most of the operations. They are again classified as right hand tools and left hand tools. (Fig 3) A right hand tool operates from the tailstock end towards the headstock, and a left hand tool operates from the headstock end towards the tailstock. The cutting edge is formed accordingly.

End-cutting tools have their cutting edge at the front end of the tools and are used on lathes for plunge-cut operations.

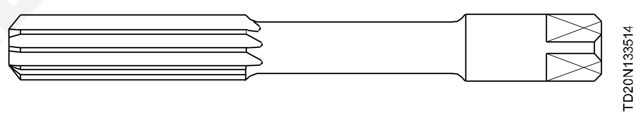
Fig 3



Multi-point cutting tools (Fig 4)

These tools have more than one cutting edge and remove metal from the work simultaneously by the action of all the cutting edges. The application of multi-point cutting tools on the lathe is mostly done by holding the tool in the tailstock and feeding it to the work.

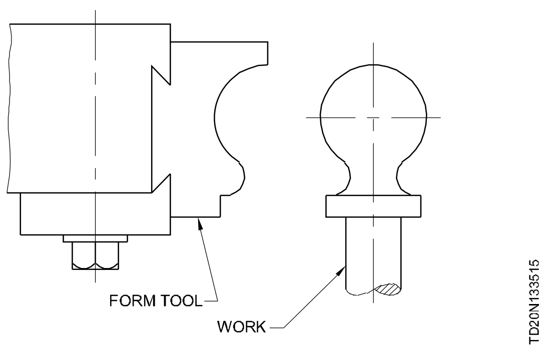
Fig 4



Form tools (Fig 5)

These tools reproduce on the work the form and shape of the cutting edges to which they are ground. Form tools perform the operations on the work by the plunging action, and are fixed on the tool post, square to the axis of the work, and are fed by the cross-slide. They may have their cutting edges formed on square or rectangular section tool blanks acting radially. The form tools may be either circular form tools or tangential form tools. They may require special holders to which they are fixed, and the holders themselves are clamped on the tool posts for the operations.

Fig 5



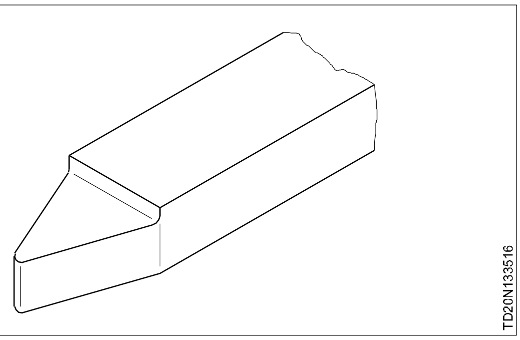
Lathe cutting tool - types

The tools used on lathes are:

- solid type tools
- brazed type tools
- inserted bits with holders
- throw-away type tools (carbide).

Solid tools (Fig 6)

Fig 6

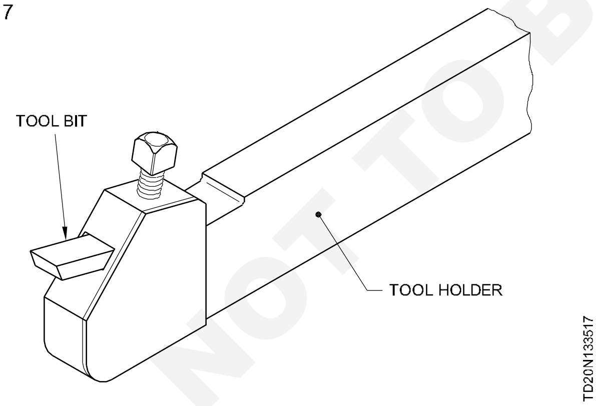


These are tools having their cutting edges ground on solid bits of square, rectangular and round cross-sections. Most of the lathe cutting tools are of the solid type, and high carbon steel and high speed steel tools are used. The length and cross-section of the tool depend upon the capacity of the machine, the type of tool post and the nature of the operation.

Inserted bits with holders (Fig 7)

Solid high speed steel tools are costly, hence, sometimes inserted bits are used. These bits are small in size, and are inserted in the holes of the holder. These holders are held and clamped in the tool posts to carry out the operations. The disadvantage in this type of tools is that the rigidity of the tool is poor.

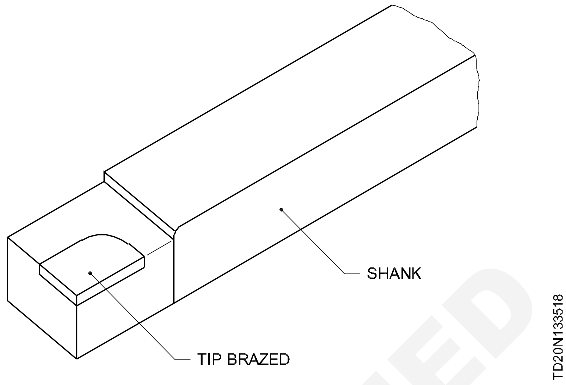
Fig 7



Brazed tools (Fig 8)

These tools are made of two different metals. The cutting portion of these tools is of cutting tool materials, and the body of the tools does not possess any cutting ability, and is tough. Tungsten carbide tools are mostly of the brazed type. Tungsten carbide bits of square, rectangular and triangular shape are brazed to the tips of the shank. The tips of the shank metal pieces are machined on the top surface according to the shape of the tips so as to accommodate the carbide bits. These tools are economical, and give better rigidity for the tools than the inserted bits clamped in the tool-holders. This is applicable to high speed steel brazed tools also.

Fig 8

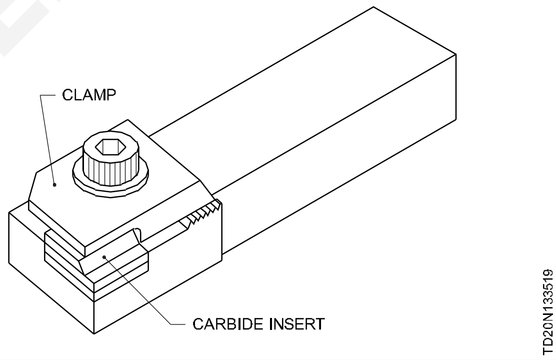


Throw-away type tools (Fig 9)

Carbide brazed tools when blunt or broken need grinding which is time absorbing and expensive. Hence, they are used as throw-away inserts in mass production. Special tool-holders are needed and the carbide bits of rectangular, square or triangular shapes are clamped in the seating faces and machined on this type of special holders.

The seating faces are machined in such a way that the rake and clearance needed for the cutting bits are automatically achieved when the bits are clamped.

Fig 9



Lathe cutting tools shapes

Lathe cutting tools are available in a variety of shapes for performing different operations. Some of the lathe cutting tools generally used are shown in Figs 10 &11.

Fig 10

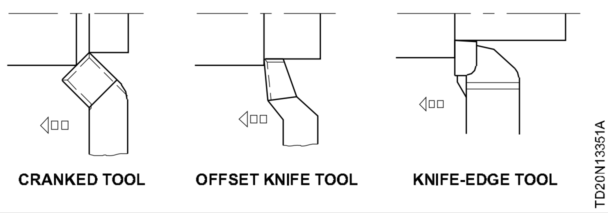
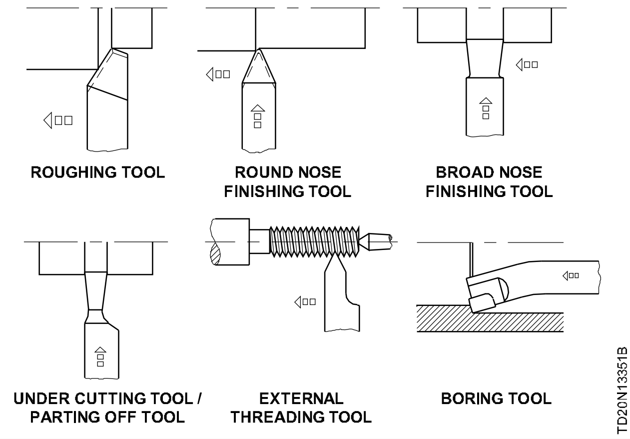


Fig 11

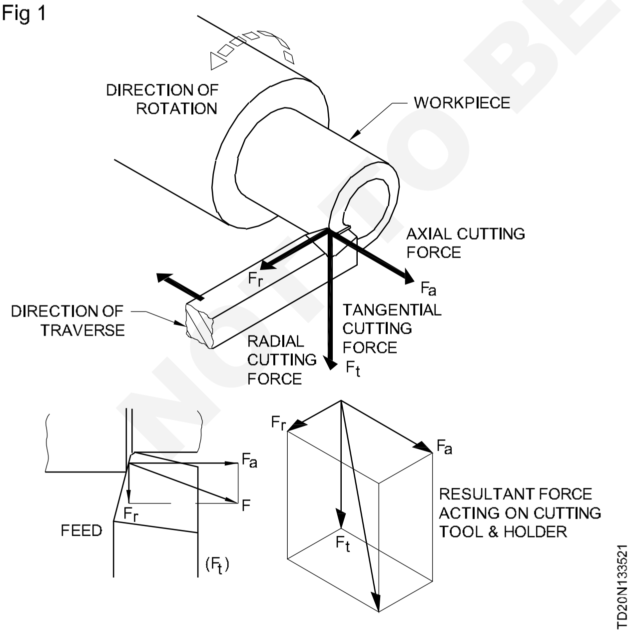


Lathe tools - angles and their functions

Objectives : At the end of this lesson you shall be able to
- **state the angles of a lathe cutting tool**
- **state the characteristics of a rake angle**
- **state the characteristics of a clearance angle**
- **state the characteristics of a relief angle.**

A lathe tool in action is shown in Fig 1. Notice that the section of the tool is basically a square or rectangle with a modified shape at the cutting end. If it is not modified and used, full face surface will make contact with the work and the tool can hardly penetrate into the work. (Fig 2) Otherwise the surface of the tool will only rub against the workpiece and hardly any cutting will take place. But by shaping the tool as shown in Fig 3, the full surface contact will be eliminated and a cutting edge will be created. The angle which is responsible for the creation of the cutting edge is called the clearance angle.

Fig 1



The tool in Fig 1 has some more angles ground in it to make the metal removal more efficient, and these angles are shown in respect of general roughing tool in Fig 4.

Fig 2

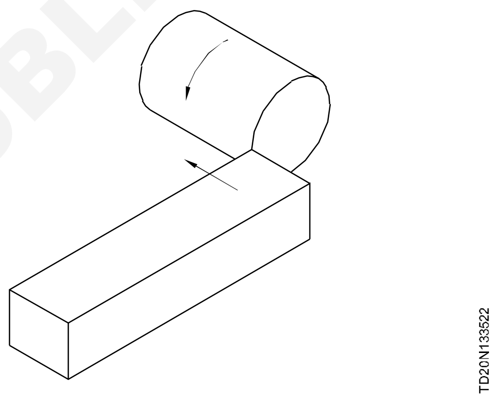
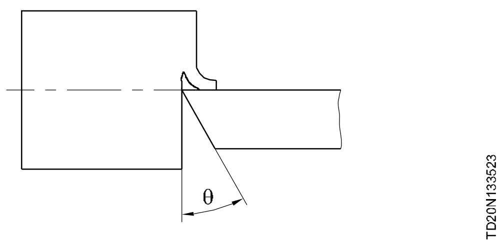


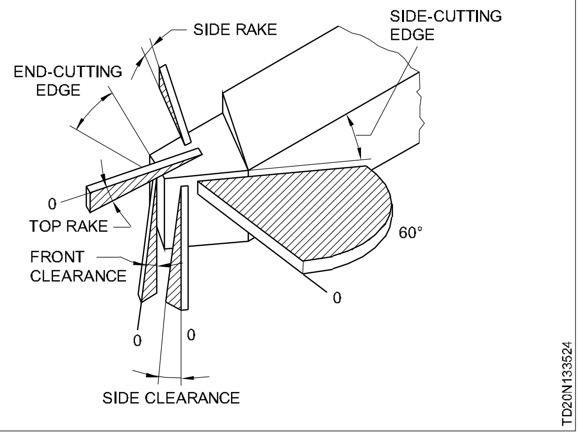
Fig 3



The angles are:

- side cutting edge angle
- end cutting edge angle
- top rake angle
- side rake angle
- front clearance angle
- side clearance angle.

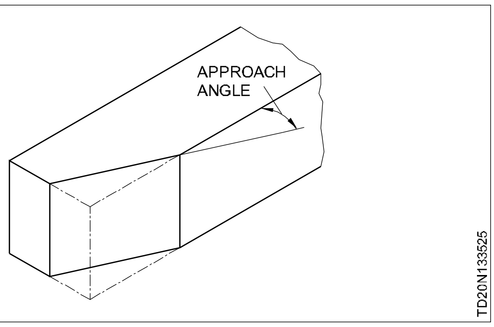
Fig 4



Side cutting edge angle

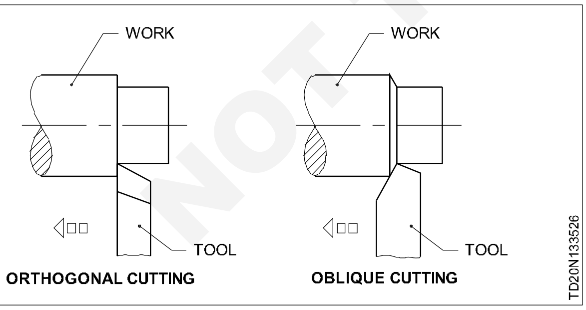
(Approach angle) (Fig 5)

Fig 5



This is ground on the side of the cutting tool. The cutting will be oblique. The angle ground may range from 25° to 40° but as a standard a 30° angle is normally provided. The oblique cutting has certain advantages over an orthogonal cutting, in which the cutting edge is straight. More depth of cut is given in the case of oblique cutting since when the tool is fed to the work, the contact surface of the tool gradually increases as the tool advances, whereas, in the case of orthogonal cutting, the length of the cutting edge for the given depth fully contacts the work from the beginning itself, which gives a sudden maximum load on the tool face. The area, over which heat is distributed, is more in oblique cutting. (Fig 6)

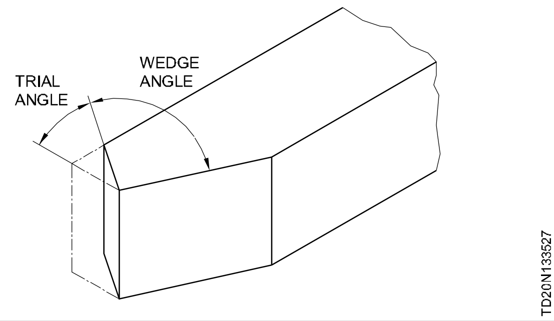
Fig 6



End-cutting edge angle (Trial angle) (Fig 7)

The end-cutting edge angle is ground at 30° to a line perpendicular to the axis of the tool, as illustrated in Fig 4. The side-cutting edge angle and the end-cutting edge angle, when ground, form a nose (wedge) angle of 90° for the tool.

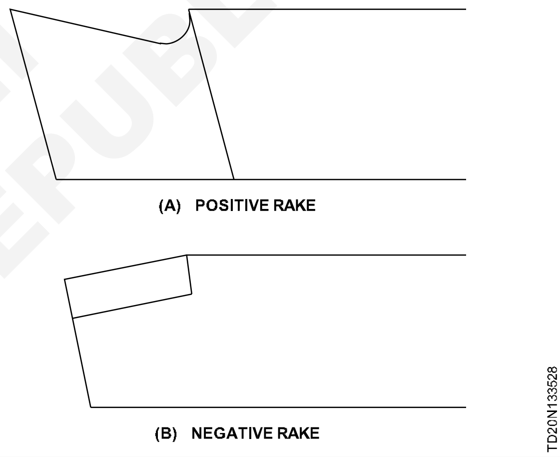
Fig 7



Top or back rake angle (Fig 8)

The rake angle ground on a tool controls the geometry of chip formation for any given material. It controls the mechanics of the cutting action of the tool. The top or back rake angle of the tool is ground on the top of the tool, and it is a slope formed between the front of the cutting edge and the top face. If the slope is from the front towards the back of the tool, it is known as a positive top rake angle,(Fig 8A) and, if the slope is from the back of the tool towards the front of the cutting edge, it is known as the negative back rake angle. (Fig 8B)

Fig 8



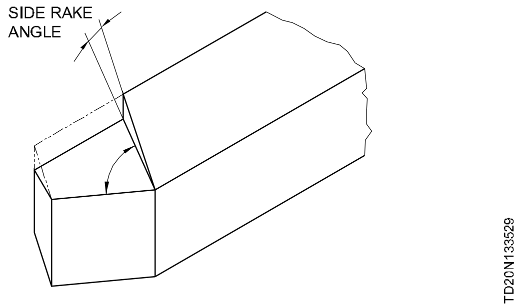
The top rake angle may be ground positive, negative or zero according to the material to be machined. When turning soft, ductile materials which form curly chips, the positive top rake angle ground will be comparatively more than for turning the hard brittle metals.

When turning hard metals with a carbide tool it is the usual practice to give a negative top rake. Negative top rake angle tools have more strength than the positive top rake angle tools.

Side rake angle (Fig 9)

The side rake angle is the slope between the side of the cutting edge to the top face of the tool widthwise. The slope is from the cutting edge to the rear side of the tool. It varies from 0° to 20° according to the material to be machined. The top and side rakes, ground on a tool, control the chip flow and this results in a true rake angle which is the direction in which the chip that shears from the work passes.

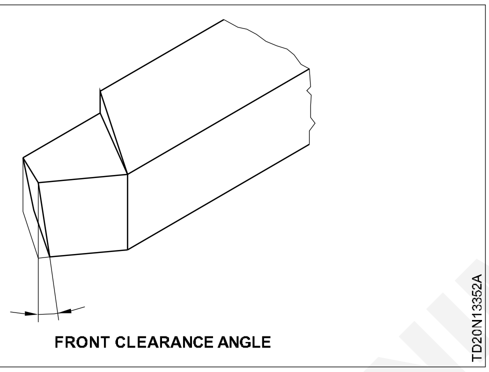
Fig 9



Front clearance angle (Fig 10)

This angle is the slope between the front of the cutting edge to a line perpendicular to the axis of the tool drawn downwards. The slope is from the top to the bottom of the tool, and permits only the cutting edge to contact the work, and avoids any rubbing action. If the clearance ground is more, it will weaken the cutting edge.

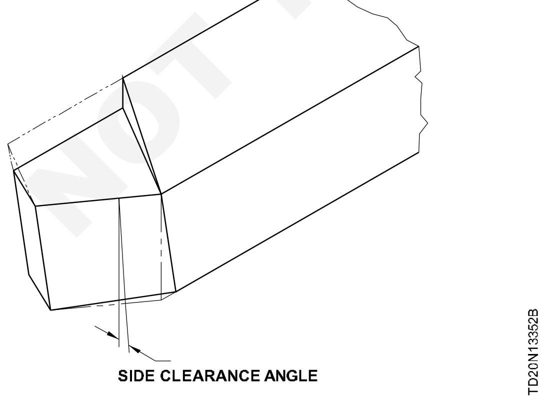
Fig 10



Side clearance angle (Fig 11)

The side clearance angle is the slope formed between the side cutting edge of the tool with a line perpendicular to the tool axis drawn downwards at the side cutting edge of the tool. The slope is from the top of the side cutting edge to the bottom face. This is also ground to prevent the tool from rubbing with the work, and allows only the cutting edge to contact the work during turning. The side clearance angle needs to be increased when the feed rate is increased.

Fig 11



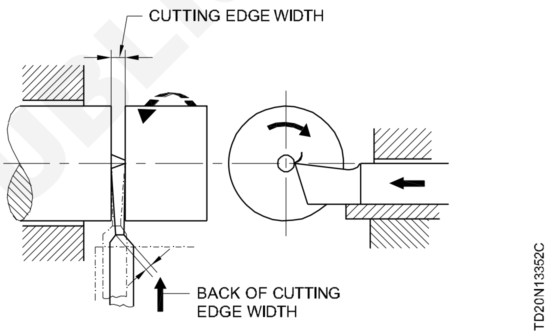
When grinding the rake and clearance angles, it is better to refer to the standard chart provided with the recommended values, and then grind. However, actual operation will indicate the performance of the tool and if any modification is needed for the angles ground on the tool.

Side relief angle (Fig 12)

This angle is ground on the parting and the undercutting tools on both sides. This will provide the width of the cutting edge slightly broader than the back of the cutting edge.

This permits clearance between the sides of the tool and the groove walls formed by the plunging action of the tool, thereby, preventing the tool from getting jammed in the groove and causing breakage. The relief is kept as minimum as possible. Too much of relief will weaken the tool cutting edge, and also permit the chips to clog in the gap, causing the tool in both cases to break. Side relief is also provided sometimes to the main cutting edge of the facing tools, permitting only the cutting point performing the operation, when the tool axis is set perpendicular to the lathe axis. The side relief angle normally does not exceed 2°.

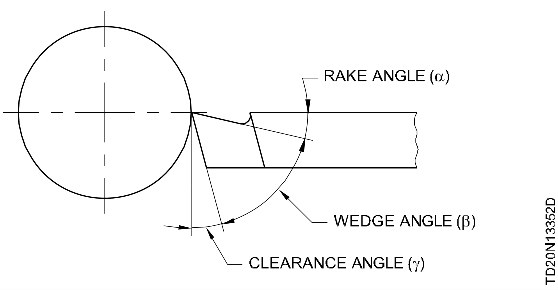
Fig 12



Relationship between rake, clearance and wedge angles (Fig 13)

The rake angle (μ), clearance angle (g) and the wedge angle (b) have close relationship for efficiency in cutting. Excessive rake angle reduces the wedge angle, which helps in good penetration and it is particularly useful for cutting soft metals. A decreased wedge angle weakens the tool strength. Therefore, for cutting hard metals, the rake angle is zero or negative. The clearance angle is generally fixed depending on the geometry of the surface being cut.

Fig 13



Different Types and Specifications of Carbide Tools

Objectives: At the end of this lesson you shall be able to
- **identify the different types of carbide tools**
- **state the specifications of carbide tools.**

Types of chip

Cemented carbide tools are available as brazed tipped tools and throwaway tips held in specially designed tool holders. Standard shapes of carbide-tipped turning and facing tools are shown in the figures 5.8.1 to 5.8.9 carbide tipped cut off and boring tools are also available. These tools are re-sharpened as needed using special silicon carbide and diamond wheels.

Standard terms for carbide tools as specified in ISO.

ISO 1 Straight turning tool

ISO 2 Cranked turning tool

ISO 3 Offset facing tool

ISO 4 Wide nose square turning tool

ISO 5 Offset turning and facing tool

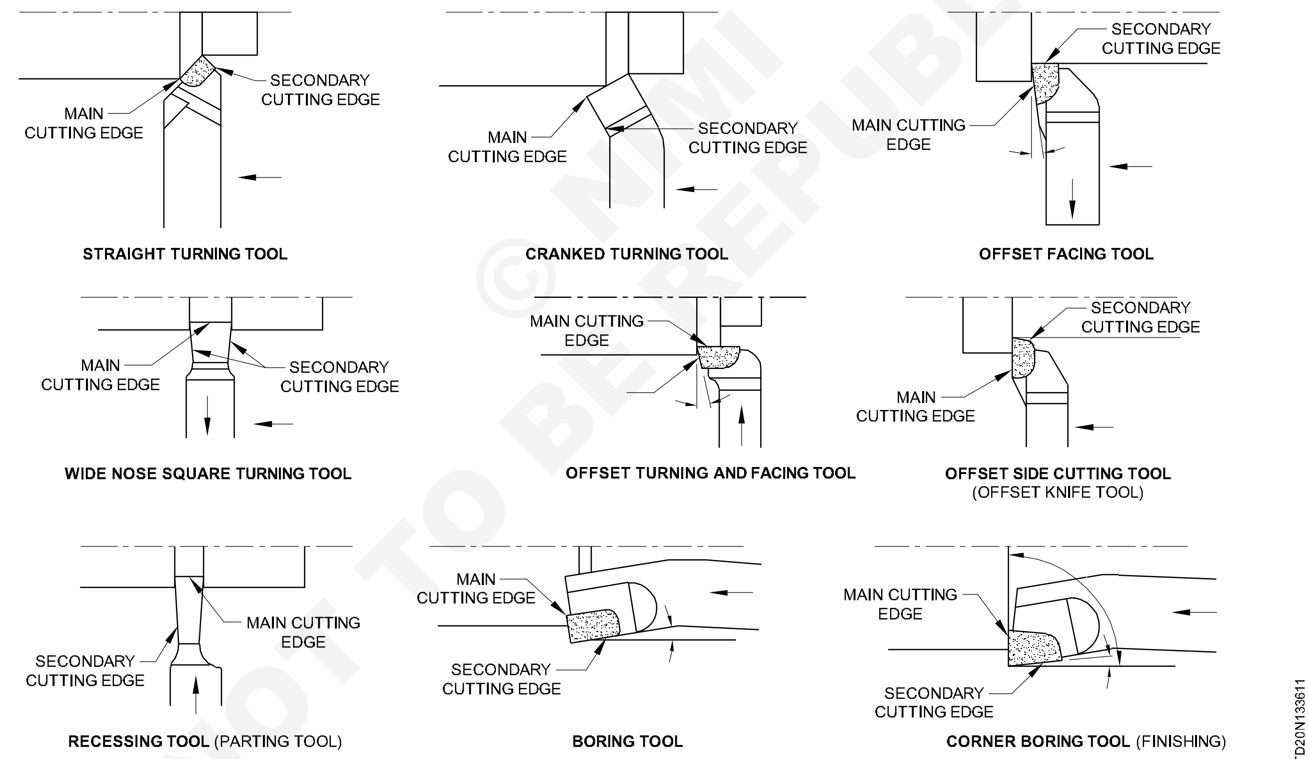
ISO 6 Offset side cutting tool (Offset knife tool)

ISO 7 Recessing tool (parting tool)

ISO 8 Boring tool

ISO 9 Corner boring tool (finishing)

Fig 1



The carbide tools are specified according to

1 The operations (rough and finish)

2 Right hand or left hand

3 Material being turned and machine conditions (Refer Table 1)

Classification

Classifications of carbide tool according to their range of application (1S: 2428-1964)

The method of referring to a straight ISO carbide tool by a manufacturer.

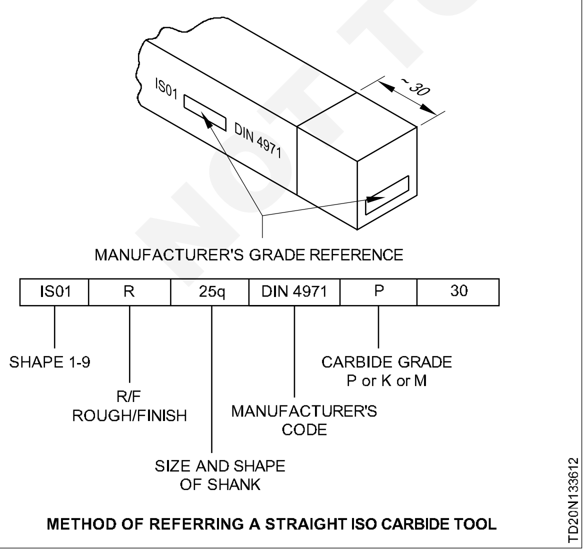
Table 1

Designation		Increasing direction of the characteristic of		Range of application	
Identification colour		Carbide tip	Cutting	Materials to be machined	Machining conditions
P01	BLUE	Resistance to wear Toughness	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) Planning: with low feed rate.
P30				Steel, steel casting, malleable cast iron, forming long chips	Turning, milling, (Cutting speed and feed: medium) Planning: with low feed rate
P40				Steel, steel casting, with sand inclusions or shrinkage cavities.	Turning, planning, shaping, Cutting speed: low, Feed: high. Rake angle: high, for machining under unfavorable conditions and work on automatic machines
P50				Steel, steel casting of medium or low tensile strength with sand inclusions or shrinkage cavities	Turning, planning, shaping, Cutting speed: low, Feed: high. Rake angle: high, for machining under unfavorable conditions and work on automatic machines
M10	YELLOW	Resistance to wear Toughness	Cutting speed Feed	Steel, steel casting, manganese steel, grey cast iron, alloyed cast iron	Turning, Cutting speed: medium to high. Feed: low to medium
M20				Steel, steel casting, austenite, manganese steel, grey cast iron, sphrodised cast iron and malleable cast iron	Turning, milling, Cutting speed: medium. Feed: medium
M30	YELLOW	Resistance to wear Toughness	Cutting speed Feed	Steel, steel casting, austenite, steel grey cast iron, heat resisting alloys	Turning, milling, planning, cutting speed: medium. Feed: medium or high
M40				Free cutting steel, low tensile, steel, brass and light alloy	Turning, profile turning, parting off especially in automatic machines

K01	RED	Resistance to wear / Toughness	Cutting speed / Feed	Very hard grey cast iron, chilled castings of hardness up to 60 HRC. Aluminum alloys with high silicon content, hardened stool, plastics of abrasive type, hard hoard 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, aluminum 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 to HB, non-ferrous metals such as copper, brass, aluminum: laminated wood of abrasive type.	Turning, milling, planning, roaming, broaching
K30				Soft grey cast iron, low tensile strength steel, laminated wood.	Turning, planning, shaping, milling. Rake angle: large, even under unfavorable conditions
				Soft or hard natural wood, non-ferrous metals.	Turning, milling, planning, shaping. Rake angle: large, even under unfavorable machining conditions

The method of referring to a straight ISO carbide tool by a manufacturer is shown in Fig 2

Fig 2



METHOD OF REFERRING A STRAIGHT ISO CARBIDE TOOL

Types of chips, chip brakers and tool life

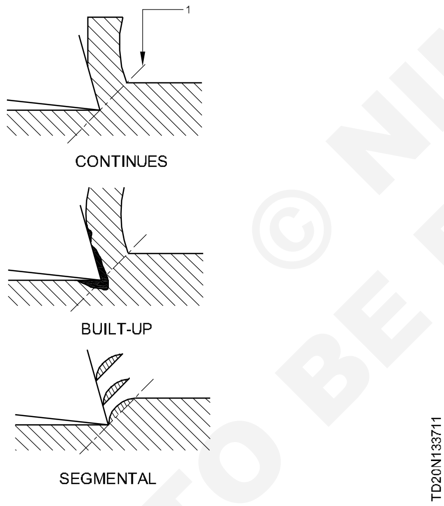
Objectives: At the end of this lesson you shall be able to
- **state the types of chips**
- **state the function of chip breakers and their uses**
- **state the relationship between cutting speed and tool life**
- **explain tool life index equation**
- **state the maximum cutting speed for a given tool life.**

Types of chip

The form and dimension of a chip in metal machining indicate the nature and quality of a particular machining process, but the type of chip formed is greatly influenced by the properties of the material being cut and various cutting conditions.

In engineering manufacture particularly in metal machining processes hard brittle metals have a very limited use, and ductile metals are mostly used. Chips of ductile metals are removed by varying proportions of tear, shear, and flow. This results in three general types of shapes (Fig 1)

Fig 1



1 The discontinuous or segmental form.

2 The continuous or ribbon type.

3 The continuous with built-up edge.

Discontinuous or segmental chips consist of elements fractured into fairly small pieces ahead of the cutting tool. This type of chip is obtained in machining most brittle materials, such as cast iron and bronze. These materials rupture during plastic deformation, and form chips as separate small pieces. As these chips are produced, the cutting edge smoothes over the irregularities, and a fairly good finish is obtained. Tool life is also reasonably good, and the power consumption is low. Discontinuous chips can also be formed on some ductile metals only under certain conditions particularly at very low speeds and if the coefficient of friction is low. With ductile metals however the surface times is bad end the tool life is short

Conditions tending to promote its formation include: brittle metal, greater depth of cut, low cutting speed and small rake angle.

Continuous chips consist of elements bonded firmly together without being fractured. Under the best conditions the metal flows by means of plastic deformation, and gives a continuous ribbon of metal which, under the microscope, shows no signs of tears or discontinuities. The upper side of a continuous chip has small notches while the lower side, which slides over the tool face, is smooth and shiny. The continuous form is considered most desirable face, is smooth and shiny. The continuous form is considered most desirable for low friction at the rool-chip interface, lower power consumption, long tool life and good surface finish.

Factor favourable to its formation are: ductile metal, such as mild steel, copper, etc., fine feed, high cutting speed, large rake angle, keen cutting edge, smooth tool face and an efficient lubrication system.

The term built-up edge implies the building up of a ridge of metal on the top surface of the tool and above the cutting edge. It appears that, when the cut is started in ductile metals, a pile of compressed and highly stressed metal forms at the extreme edge of the tool. Owing to the high heat and pressure generated there, this piled up metal is welded to the cutting tip and forms a "false" cutting edge tool. This is usually referrd to as the "built up edge". This weld metal is extremely strain hardened and brittle. So the weaker chip metal tears away from the weld as the chip moves along the tool face. The buit-up becoming unstable, breaks down and some fragments leave with the chip as it passes off and the rest adheres to the work surface producing the characteristic rough surface. The built-up edge appears to be a rather permanent structure as long as the cut is continuous at relatively high speeds and has the effect of slightly altering the rake angle. At very high speeds, usually associated with sintered carbide tools, the the built-up edge is very small or nonexistent, and a smooth machined surface results.

Conditions tending to promote the formation of built-up edges include: low cutting speed, low rake angle, high feed, lack of cutting fluid and large depth of cut.

The chip breakers and their uses

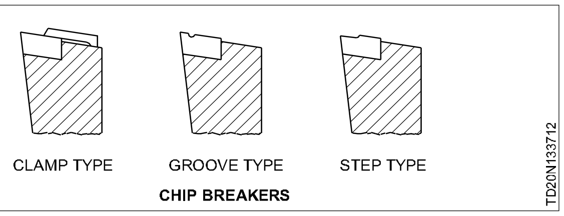
Chip breaker

Chip breaker 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 2)

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

Fig 2



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

- 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.

Tool life and factors effecting tool life

Objectives: At the end of this lesson you shall be able to
- **State what is life**
- **explain the factors effecting the tool life.**

Tool life is a most important factor in the evaluation of machinability,

it is the period of time in which the tool cuts effectively and efficiently.

The methods of expressing tool life.

There are many ways of expressing the tool life, such as

- Time unit - It is the most commonly used tool life unit.
- Volume of material removed by tool during its total lifespan.
- Number of work pieces machined by a tool.

Tool life most commonly expressed in minute, expected life of some tool material is given below

Cast tool steel - 120 minute

High speed steel tool - 60 to120 minute

Cemented carbide tool - 420 to 480minute

Factor affecting tool life:

- Cutting speed
- Feed and depth of cut
- Tool geometry
- Tool material
- Work material
- Nature of cutting
- Rigidity of machine tool and work
- Cutting fluids

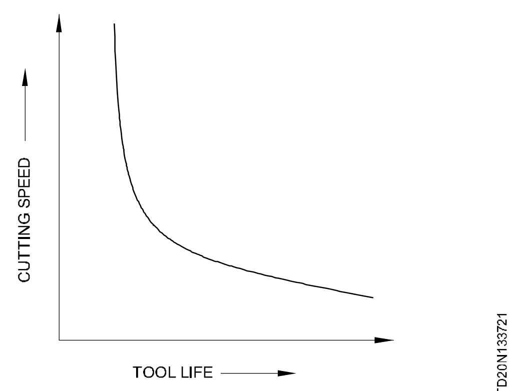
Cutting speed:

It is major for affecting tool life.

It varies inversely with tool life which leads to parabolic curve as shown in Fig 1

Tool life relation is invented by F.W.Taylor.

Fig 1



$VT^n = C$

V = cutting speed in m/min,

T = Tool life in min,

n = Tool life index

C = machining constant

N and C are constant for a given set of conditions

Feed Rate and Depth of Cut:

According to the Taylor's tool life equation, tool life decreases when feed rate increases. Also, same for the depth of cut.

Tool Geometry:

The tool geometry greatly affects the tool life. the effect of all the tool parameters on tool life is as follows

- Back Rake Angle.
- Principal Cutting Edge.
- Clearance Angle.
- Nose Radius.

Back Rake Angle:

Larger the rake angle smaller will be the cutting angle and larger will be shear angle, this reduces the cutting force and power, and hence less heat generated during cutting, means reduced cutting temperature, results in longer tool life.

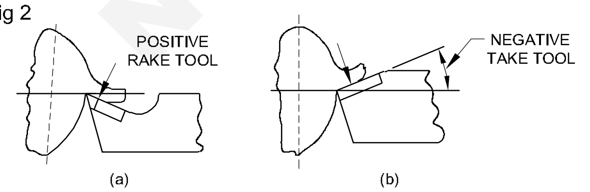
But on the other hand, increasing the rake angle results in mechanically weak cutting edge the positive rake tool experiences shear stress and the tip is likely to be sheared-off.

Negative rake increases cutting force and power, hence more heat and temperature generated results in smaller tool life.

Therefore, there lies an optimum value of the back rake which depends upon tool material and work material. It ranges from -5° to +15°. An optimum value of rake angle is about 14° which gives maximum tool life.

Fig. 2 shows cutting process using positive and negative rake tools. The positive rake tool experiences shear stress and the tip is likely to be sheared off. Whereas tool with negative rake experiences

Fig 2



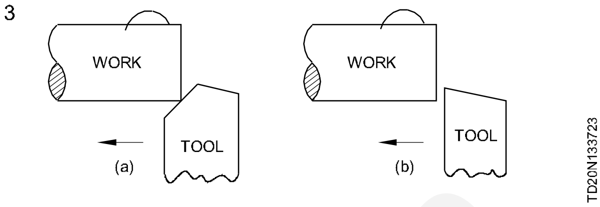
CUTTING PROCESS USING POSITIVE AND NEGATIVE RAKE TOOLS

compressive stress. The carbide and ceramic tools are generally given negative rake because they are weak in shear and good in compression

Principal Cutting Edge:

Fig 3 shows two different arrangements of principal cutting edge angles. Fig. 3 (a), the contact is gradually starting from a point quite away from the tip. Therefore, the tool experiences the cutting force gradually and over a larger area. Hence the tool is safer and tool life is more as compared to the Fig. 2(b) in which the principal cutting edge angle is 90°.

Fig 3



Clearance Angle:

Increase in clearance angle results in significant reduced flank wear, so increased tool life. But the cutting edge will become weaker as the clearance angle is increased. Therefore an optimum value is required. The best compromise is 5° (with carbide tools) to 8° (with H.S.S. tools) for common work materials.

Nose Radius:

The nose radius improves tool life and surface finish.

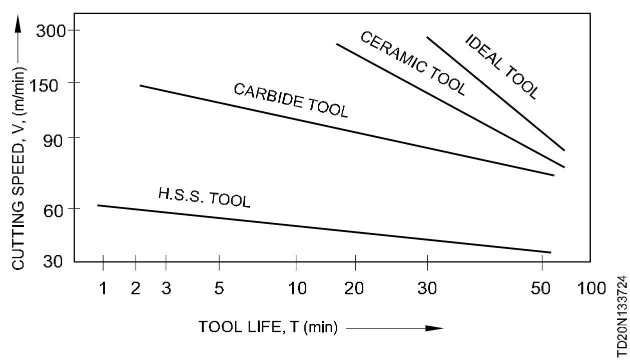
There is an optimum value of nose radius at which the tool life is maximum.

- If the radius exceeds optimum value, the tool life decreases.
- Larger radius means larger area of contact between the tool and work piece. Due to which more frictional heat is generated, results in increased cutting force. Due to which the work piece may starts, vibrating, hence if rigidity is not very high, brittle tools (carbides and ceramics) will fail due to chipping of cutting edge.

Tool Material:

- The major requirements of cutting tool materials are: Hot hardness, impact toughness, and wear resistance. For better tool life, the material must have the above properties. Fig. 4 shows tool life variation against cutting speeds for different tool materials.

Fig 4



It is very clear from the figure; at any cutting speed the tool life is maximum for ceramic tool and lowest for the high speed steel tool. So using ceramic tool maximum volume of material could be removed at any cutting speed for a specific tool life.

Hardness of Work piece:

As the hardness increases, the permissible velocity decreases for a given tool life. For example, the tool life is 50 minutes for cutting less hard material, now if say harder material is to be cut then to maintain the tool life as 50 minutes, the cutting velocity should be reduced proportionate.

Nature of Cutting:

If the cutting is intermittent, the tool bears impact loading, results in chance of its quick failure. In continuous and steady cutting, the tool life is more.

Rigidity of Work piece-Machine Tool System:

Higher is the rigidity of system higher will be the tool life. Lower the rigidity of the system, higher is the chance of tool failure, by vibrations of tool or work piece. Rigidity is the prime requirement in case of intermittent cutting especially when brittle tools are used.

Type of Cutting Fluid and its Method of Application:

Application of suitable cutting fluid obviously increases tool life or in other words, for the same tool life, allowable cutting speed increases. Fig. 9.30 shows the effect of cutting fluid on tool life for different tool materials. The tool life even increases by 150 per cent at some speeds. All types of cutting fluids do not have equal effect, some of them more, some are less

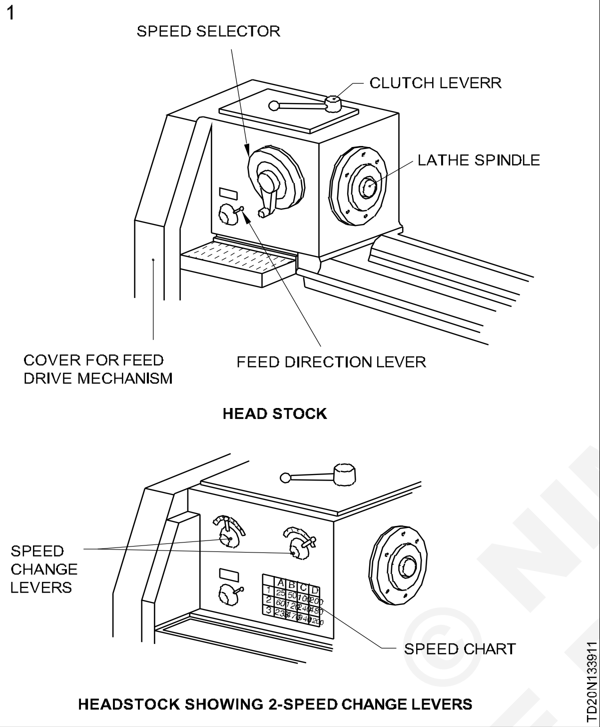
Lathe driving mechanism for all gear types

Objectives: At the end of this lesson you shall be able to
- **state the functions of headstock**
- **explain the difference between cone pulley headstock and all geared headstock.**

Headstock

It is a fixed unit of lathe on the left hand side. (Fig 1)

Fig 1



Its main functions are to:

- provide a means to assemble work - holding devices

Transmit the drive from the main motor to work to make it revolve

Accommodate shafts with fixed and sliding gears for providing a wide range of work speeds

Have shift levers to slide gears to bring in mesh for different speeds

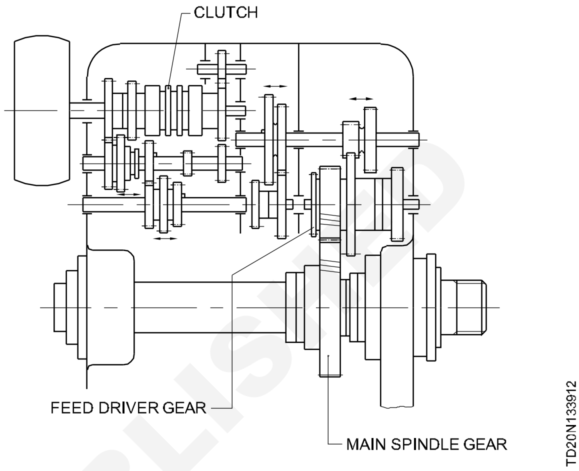
Have a means for lubricating the gears, shafts and bearings.

Constructional features of all-geared headstock (Fig 2)

It is a box-section alloy iron casting having a top cover which can be removed, if needed, It has an input shaft which is connected by means of 'V' belts to the main motor, and runs at constant speed. It is equipped with clutches and a brake.

There may be two or more intermediate shafts on which sliding gears are mounted. The main spindle is the last driven shaft in the headstock assembly. The nose of the spindle is outside the headstock casting and is designed to accommodate the work-holding devices.

Fig 2



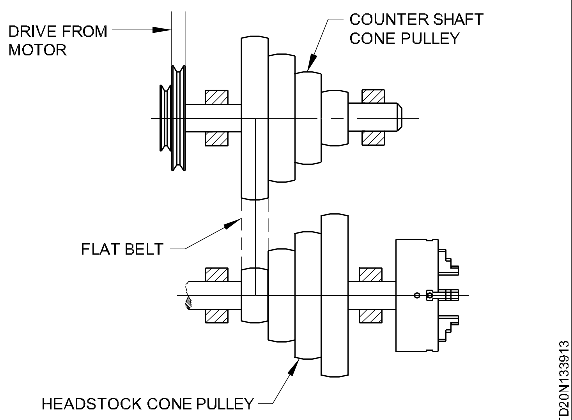
The levers operating the forks of the sliding gears are situated outside in the front of the headstock casting. A sight glass is provided on the top to indicate the functioning of the automatic lubricating system and side of sight glass is provided oil length of the machine.

Cone pulley headstock (Fig 3)

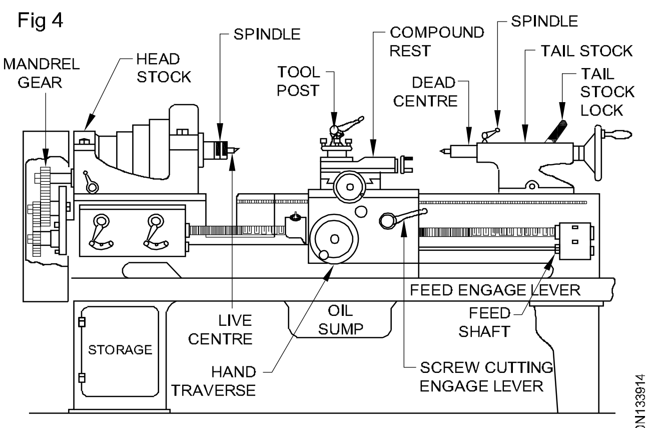
It has a stepped cone pulley mounted on the main spindle and is free to revolve. It is connected by means of a flat belt to a similar cone pulley, the steps arranged in a reversing order. This cone pulley gets the drive from the main motor.

The spindle is mounted on the bush bearings in the headstock casting and a gearwheel called 'bullgear' keyed to it. A pinion is coupled to the cone pulley. The back gear unit has a shaft which carries a gear and a pinion. The number of teeth of the gear and pinion on the back gear

Fig 3



Shaft corrresponds to the number of teeth on the bull gear and the pinion on the cone pulley. The axis of the back gear shaft is parallel to the axis of the main spindle, and the back gear is brought in engagement or disengagement with the cone pulley system by means of a lever. The back gear unit is engaged to have reduced spindle speeds. (Fig 4)



CORE PULLEY HEAD STOCK

A three-stepped cone pulley headstock provides three direct ranges of speeds through the belt connection. and with the back gear in engagement, three further ranges of reduced speeds.

Advantages

Easy for maintenance.

Can take up heavy load.

Less noise during functioning.

During overloads, the belt slips off, and hence, no major damage to the lathe is caused.

Positive drive when the back gear is in engagement.

Disadvantages

Number of spindle speeds limited to the the number of steps in the cone pulley.

Takes time to change spindle speeds.

Needs adjustments of bush bearings.

Back gear

Objectives: At the end of this lesson you shall be able to
- **state the construction details of Back gear assembly**
- **state the function and purpose of Back gear.**

Back gear

As its name implies “back gear” is a gear mounted at the back of the head stock. It is used to reduce the speed.

Necessity of back gear: For machining big job, taking rough cut use need more power at reduced speed of spindle the back gear provides this feature in a lathe

Use of backgear

It enables to rotate the chuck at very slow speed

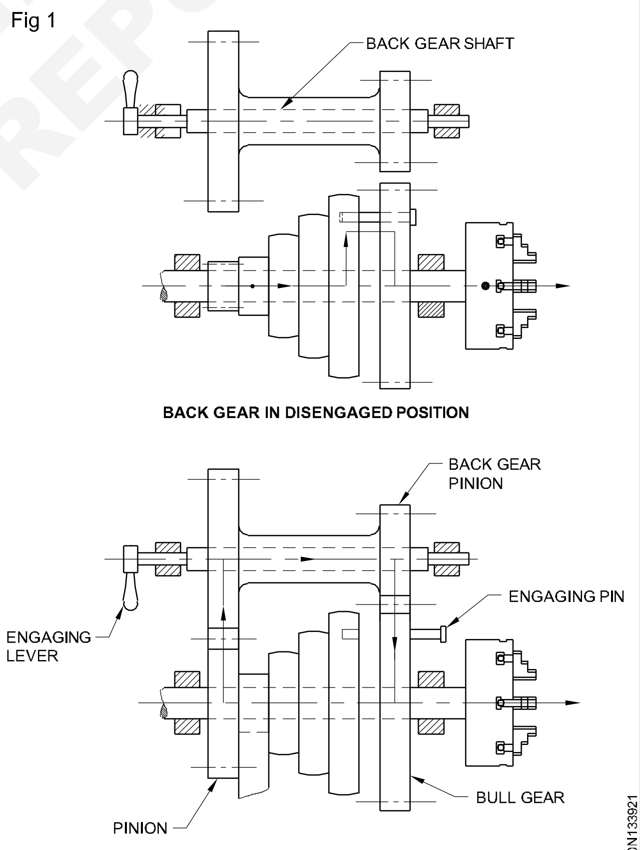
It provides increased turning power

It is highly suitable for turning large diameter casing (200 rpm)

It reduces the rpm but increases the torque

Even the largest face plate mounted job can be turned succesfully

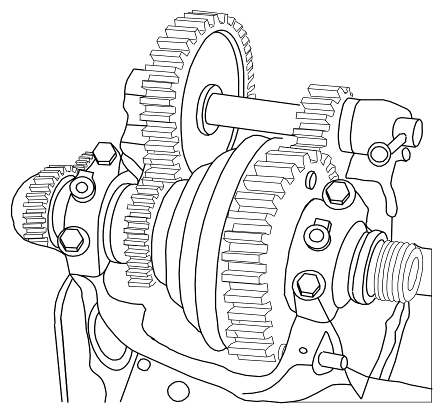
The spindle is mounted on the bush bearings in the headstock casting and a gear wheel called ‘bullgear’ keyed to it. A pinion is coupled to the cone pulley. The back gear unit has a shaft which carries a gear and a pinion. The number of teeth of the gear and pinion on the back gear shaft corresponds to the number of teeth on the bull gear and the pinion on the cone pulley. The axis of the back gear eccentric shaft is parallel to the axis of the main spindle, and the back gear is brought in engagement or disengagement with the cone pulley system by means of a lever. The back gear unit is engaged to have reduced spindle speeds. (Fig 1)



BACK GEAR IN ENGAGED POSITION

A three-stepped cone pulley headstock provides three direct ranges of speeds through the belt connection, and with the back gear in engagement, three further ranges of reduced speeds (Fig 2).

Fig 2



BACK GEAR

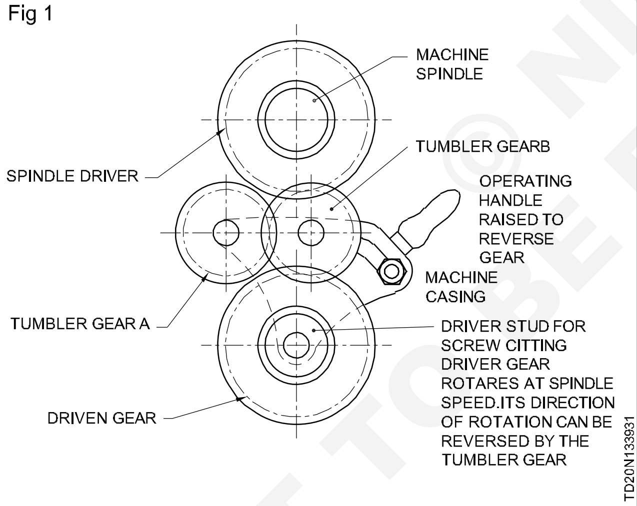
Tumbler gear set

Objectives: At the end of this lesson you shall be able to
- **state the purpose of the tumbler gear mechanism**
- **state the construction details of the tumbler gear mechanism.**

Tumbler gear mechanism (Fig 1)

The tumbler gear mechanism is used for changing the direction of rotation of the lead screw and feed shaft. It is normally situated between the spindle drive and the feed gear box. It consists of 3 gears arranged in a simple gear train, mounted on a bracket can be shifted into 3 positions.

Fig 1



For forward rotation of the lead screw and feed shaft.

For neutral position (no rotation of lead screw and feed shaft)

For the reverse rotation of the lead screw and feed shaft.

In practice, the first driver gear of a screw cutting train is not fitted directly to the lathe spindle but is mounted on a driver stud which rotates at the same speed as the spindle.

The driving gear on the spindle drives the fixed stud gear, and, since they have the same speed, they must be of the same size. Tumbler gear A is always in mesh with the driven gear and in mesh with the fellow tumbler gear B. In the figure, the drive is directed through the tumbler gear A, and tumbler gear B is idle.

If the tumbler bracket is moved upwards, tumbler gear A rolls around the driven gear until it is out of mesh with the driver gear, and tumbler gear B moves into mesh with the driver, reversing the direction of the driven gear. Thus the two trains available are:

Forward: Driver —> A —> Driven

Reverse: Driver —> B —> A —> Driven.

In yet another position of the tumbler bracket, tumbler gears A or B do not mesh with the driver gear and no drive is transmitted to the driven gear. No feed movement or thread cutting is possible.

Feed machanism of lathe

Objectives: At the end of this lesson you shall be able to
- **explain the parts of the feed machanism**
- **state the functional features of the feed mechanism.**

Feed mechanism (Fig 1)

The feed mechanism of a lathe enables automatic feeding longitudinally and transversely as needed. By automatic feeding the finish on the work will be better, the feeding of the tool will be uniform by a continuous rate and it takes less time to finish the operation while manual labour is avoided.

The feed mechanism comprises the following.

Spindle gear (A)

Tumbler gear unit (B)

Fixed stud gear (C)

Change gear unit (DEFG)

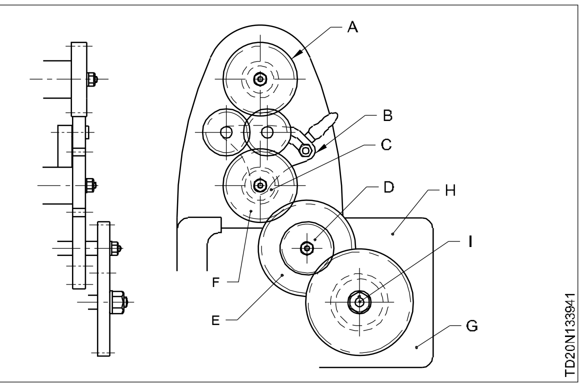
Quick change gearbox (H)

Feed shaft/ lead screw (I)

Apron meachinsm (not in figure)

The proportionate tool movement for each revolution of work is achieved through all the above units of the feed mechanism.

Fig 1



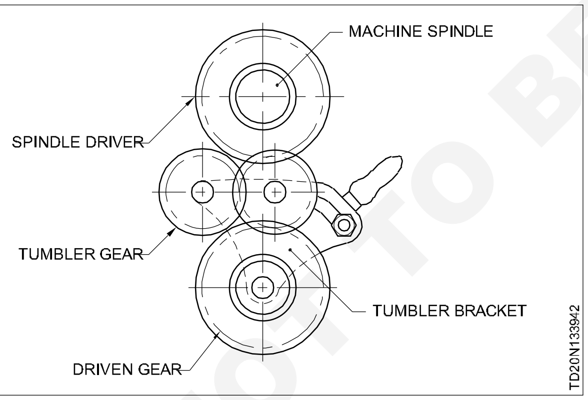
Spindle gear

The spindle gear is fitted to the main spindle, and it is outside the headstock casting. It revolves along with the main spindle.

Tumbler gear unit

The tumbler gear unit itself consists of three gears, each having the same number of teeth and it connects the spindle gear to the fixed gear. It is also called the reversing gear unit as it is used to change the direction of feed of the tool for the same direction of rotation of the spindle. It can be engaged and spindle. It can be engaged and disengaged with the fixed spindle gear by the operation of the hand lever provided in the unit. (Fig 2)

Fig 2



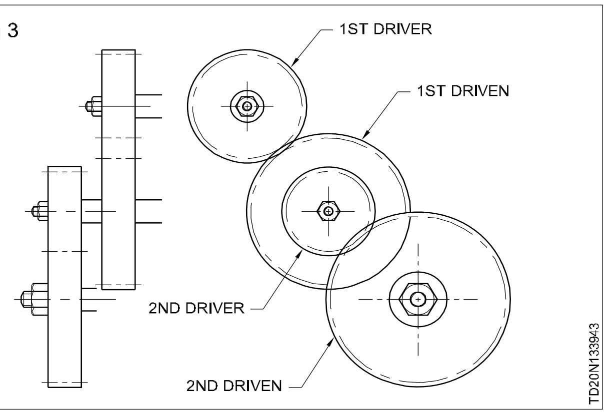
The fixed stud gear

The fixed stud gear gets the drive from the main spindle gear through the tumbler gear unit and runs at the same number of revolutions per minute as the spindle gear on most lathes.

Change gear unit

The fixed stud gear transmits its drive through a change gear unit to the quick change gearbox. The change gear unit has provision for changing the driver, the driven and the idler gears from the set of change gears available for the purpose of feed changing as an additional unit. (Fig 3)

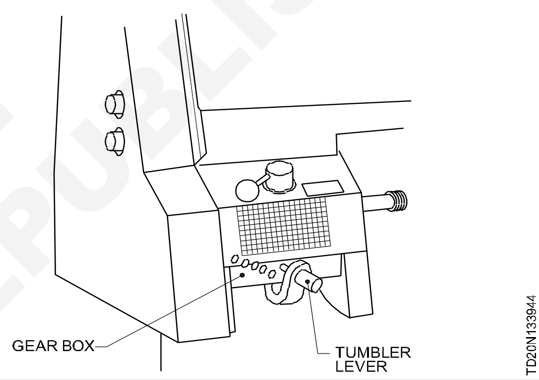
Fig 3



Quick change gearbox

The quick change gearbox is provided with levers outside box casting, and by shifting the levers, different gears are brought in mesh so that different feed rates can be given to the tool. A chart listing the different feed rates for the different positions of the levers is fixed to the casting, and by referring to the table, the levers may be engaged in position for the required feed rate. (Fig 4)

Fig 4



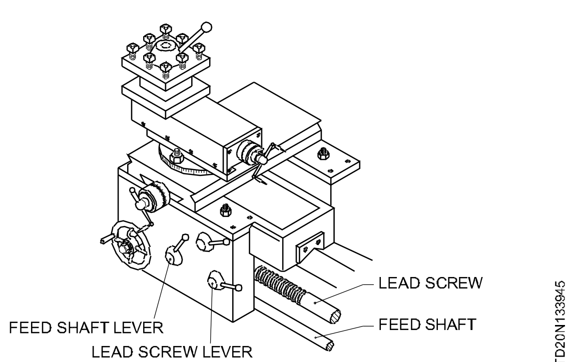
The feed shaft

The feed shaft gets its drive from the quick change gearbox, and through the apron mechanism, the rotary movement of the feed shaft is converted into the linear movement of the tool.

The apron mechanism

The apron mechanism has an arrangement for transmitting the drive from the feed shaft to the saddle for longitudinal movement of the tool or to the cross-slide for the transverse movement of tool. (Fig 5)

Fig 5



Cutting speed and feed, depth of cut and recommended speed

Objectives: At the end of this lesson you shall be able to
- **explain 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)

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 pxD. 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

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

Where V = cutting speed in metre/min

π = 3.14

D = diameter of the work in mm.

N = r.p.m.

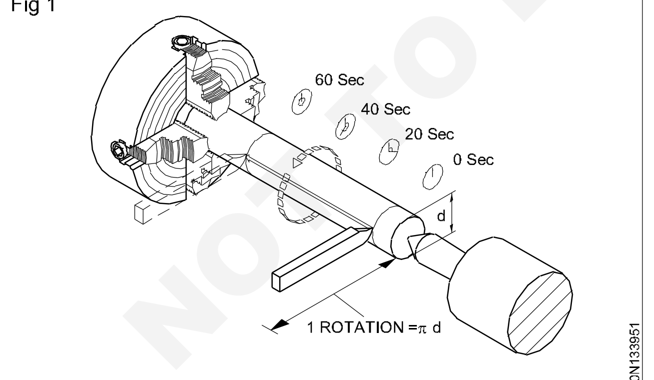
Example

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

$$V = \frac{\pi DN}{1000} \quad N = \frac{1000V}{\pi D}$$

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

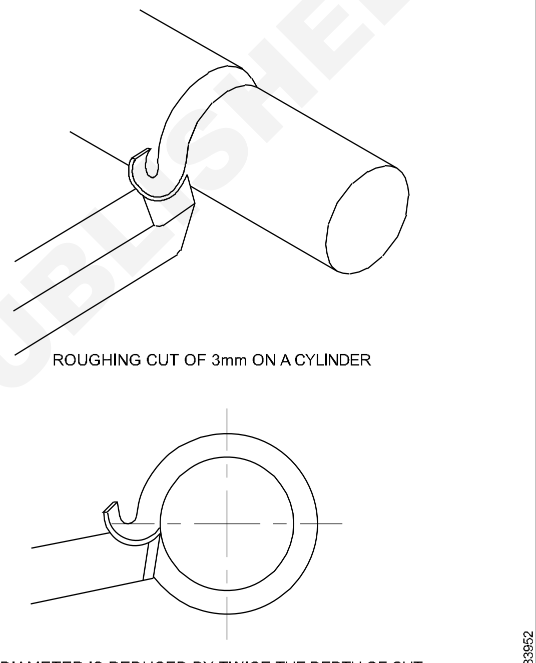
Fig 1



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 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)

Fig 2



DEPTH OF CUT

Factors governing the cutting speed

Finish required

Depth of cut

Tool geometry

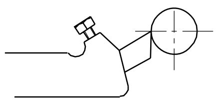
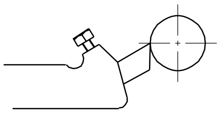
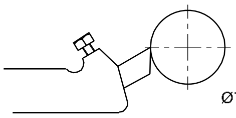
Properties and rigidity of the cutting tool and its mounting

Properties of the workpiece material

Rigidity of the workpiece

Type of cutting fluid used & Rigidity of machine tool

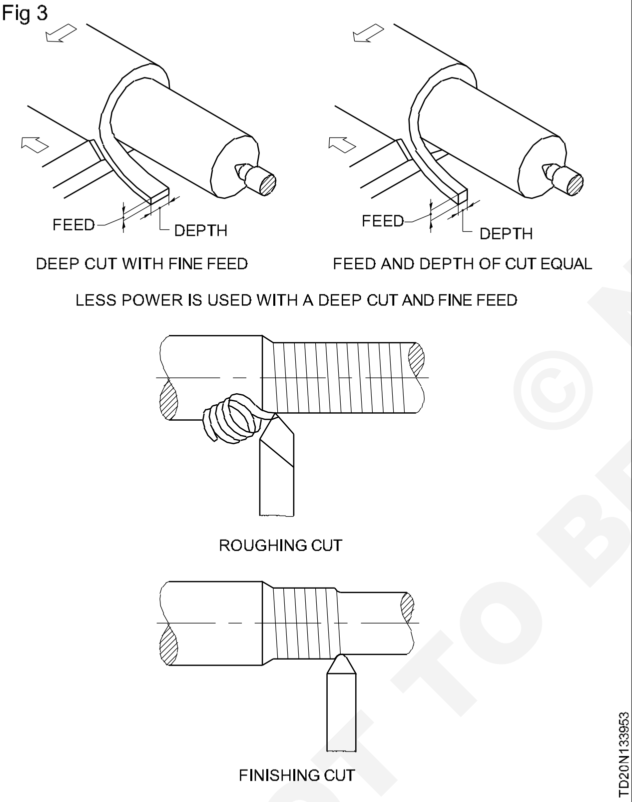
Relationsship of r.p.m to the cutting speed on different diameter (Fig 3)

Cutting speed 120m/min	Length of metal passing cutting tool in one revolution	Calculated r.p.m of spindle
	.... 78.5 mm	1528
	.... 157.0 mm	764
	.... 235.5 mm	509.5

Feed (Fig 4)

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.

Fig 3



Factors governing feed

Tool geometry

Surface finish required on the work

Rigidity of the tool

Coolant used.

Depth of cut (Fig 4)

It is defined as the perpendicular distance measured between the machined surface (d) and the unmachined surface (D) expressed in mm.

$$\text{Depth of cut} = \frac{D - d}{2}$$

Rate of metal remove

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 are given in table 1

Table 1

Material being turned	Feed	Cutting speed
Aluminium	0.2 - 1.00	70-100
Brass (alpha) - ductile	0.2 - 1.00	50-80
Brass (free cutting)	0.2 - 15	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.

Orthogonal cutting and oblique cutting

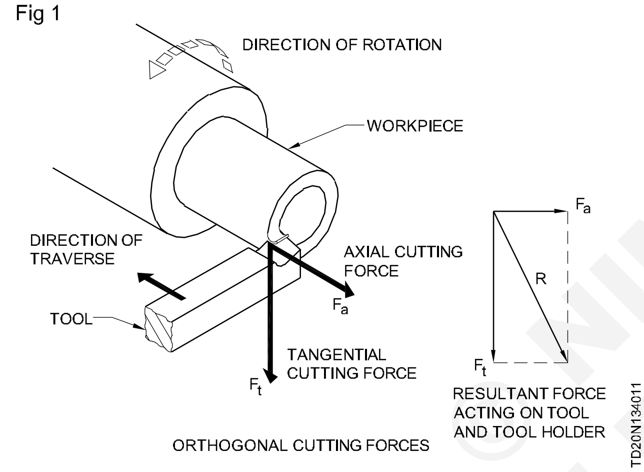
Objectives: At the end of this lesson you shall be able to
- **explain difference between orthogonal cutting and oblique cutting**
- **state the advantage and disadvantages of orthogonal cutting**
- **state about the work holding devices**
- **state the advantages and disadvantages of oblique cutting.**

Orthogonal cutting (Fig 1)

Orthogonal cutting is a process of cutting operation where only two forces are acting on the tool which are shown in figure 1, They are:

- tangential force
- axial force.

Fig 1



In orthogonal cutting, the cutting edge is parallel to the axisof the tool.

The major force is the tangential force, but the following factors affect the magnitude of both the forces.

Rake angle

Depth of cut

Rate of feed

Resistance to cutting by the material of the work piece

Friction between the chip and the tool

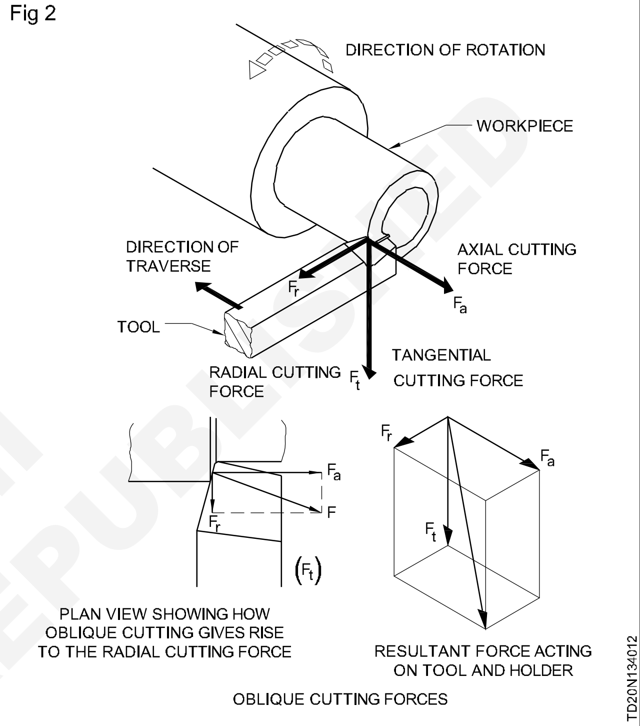
Orthogonal cutting is suitable for finish turning where the depth of cut is less. The chattering during turning is much reduced. The main disadvantage is at the start of the cutting itself as the load will act suddenly on the tool.

Oblique cutting (Fig 2)

Oblique cutting is a process of cutting operation where three forces are acting on the tool as shown in the figure. They are:

- axial force

Fig 2



- tangential cutting force
- radial force.

Though the radial force does not affect the power consumed, it gives stability to the cutting operation. This force keeps the cross-slide screw and nut in contact. In oblique cutting the cutting edge is at an angle to the axis of the tool.

Advantages of oblique cutting

For the given depth of cut the load is distributed at the start of the cutting operation.

As the length of the cutting edge is more, load per unit area is less.

It is suitable for rough turning.

Disadvantages of oblique cutting

The tool has to be ground for the approach angle. So more cutting tool material is consumed.

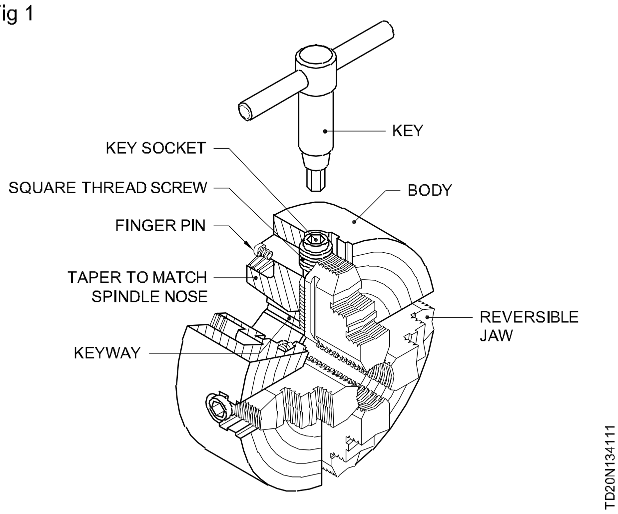
As it produces an angular shoulder to make it a square shoulder, another cutting tool should be used.

Lathe accessories - work holding devices : Four jaw chuck

Objectives : At the end of this lesson you shall be able to
- **explain the parts of a Four jaw chuck**
- **state the constructional features of a Four jaw chuck.**

Four-jaw chuck (Fig 1)

Fig 1



The four-jaw chuck is also known as an independent chuck, since each jaw can be adjusted independently; a work can be trued to within 0.001" or 0.02 mm accuracy, using this chuck.

This type of chuck is much more heavily constructed than the self-centering chuck, and has much greater holding power. Each jaw is moved independently by a square thread screw. The jaws are reversible for holding large diameter jobs. The independent four-jaw chuck has four jaws each working independently of the others in its own slot in the chuck body and actuated by its own separate square threaded screw. By suitable adjustment of the jaws, a workpiece can be set to run either true or eccentric with the machine centre.

Finished jobs when held in a four-jaw chuck can be trued with the help of a dial test indicator.

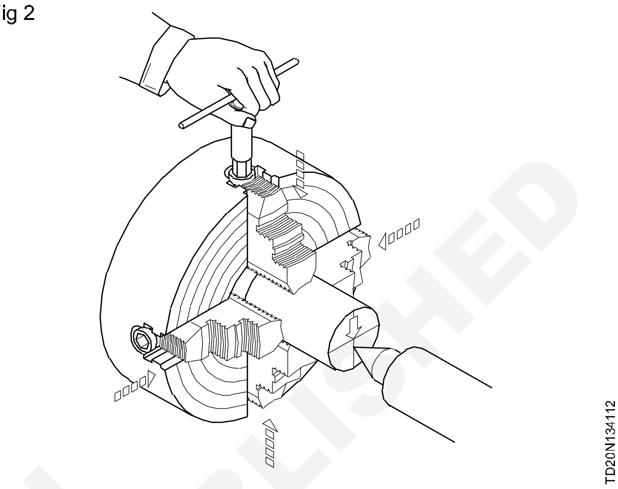
The checking of the workpiece should be carried out near the chuck and repeated as far from it as the workpiece permits, to ensure that the work is not held in the chuck at an angle to the axis of rotation.

The independent adjustment also provides the facility of deliberately setting the work off-centre to produce an eccentric workpiece. (Fig 2)

The parts of a four-jaw chuck are the:

- back plate
- body
- jaws
- square threaded screw shaft.

Fig 2



Back plate

The back plate is fastened to the back of the body by means of Allen screws. It is made out of cast iron/steel. Its bore is tapered to suit the taper of the spindle nose. It has a keyway which fits into the key provided on the spindle nose. There is a step in front on which the thread is cut. A threaded collar, which is mounted on the spindle, locks the chuck by means of the thread, and locates by means of the taper and key. Some chucks do not have back plates.

Body

The body is made out of cast iron/cast steel and the face is flame-hardened. It has four openings 90° apart to assemble the jaws and operate them. Four screw shafts are fixed on the periphery of the body by means of finger pins. The screw is rotated by means of a chuck key. The body, hollow in the cross-section, has equi-spaced circular rings provided on the face, which are marked by numerical numbers. Number 1 starts in the middle, and increases towards the periphery.

Jaws

The jaws are made out of high carbon steel, hardened and tempered, which slide on the openings of the body. These jaws are reversible for holding hollow work.

The back side of the jaw is square-threaded which helps in fixing the jaws with the operating screws.

Screw shaft

The screw shaft is made out of high carbon steel, hardened, tempered and ground. The top portion of the screw shaft is provided with a square slot to accommodate the chuck key. On the body portion, a left hand square thread is cut.

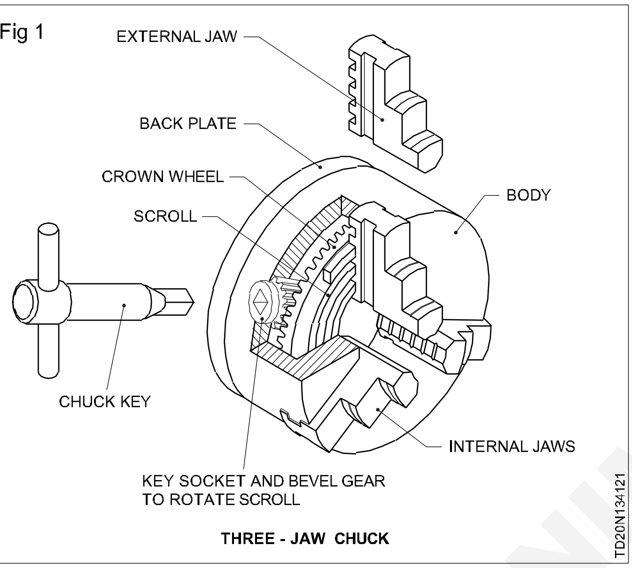
In the middle of the screw shaft, a narrow step is made and held by means of finger pins. The finger pins permit the screws to rotate but not to advance.

Lathe accessories - work holding devices : Three jaw chuck

Objectives : At the end of this lesson you shall be able to
- **explain the parts of a three jaw chuck**
- **state the constructional features of a three jaw chuck**
- **explain the difference between a three jaw chuck and a four jaw chuck**
- **state the merits and demerits of the four jaw chuck over a 3 jaw chuck.**

The three jaw chuck (Fig 1)

Fig 1



THREE - JAW CHUCK

The three jaw chuck is also known as self-centering chuck. The majority of the chucks have two sets of jaws for holding internal and external diameters. Only perfectly round work, or work with equally spaced flats, divisible by three, should be held in a three jaw chuck.

The construction of a three jaw chuck shows that the scroll not only clamps a component in place but also locates the component. This is fundamentally a bad practice, since any wear in the scroll and / or the jaws impairs the accuracy of location. Further, there is no means of adjustment possible to compensate for this wear.

The jaws of this type of chuck are not reversible, and separate internal and external jaws have to be used.

Parts of a three jaw chuck (Fig 1)

- Back plate
- Body
- Jaws
- Crown wheel
- Pinion

Back plate

The back plate is fastened at the back of the body by means of allen screws. It is made out of cast iron. Its bore is tapered to suit the taper of the spindle nose. It has a key-way which will fit into the key provided on the spindle nose. There is a step in the front on which the thread is cut. The threaded collar, which is mounted on the spindle, locks the chuck by means of the thread, and locates by means of the taper and the key.

Body

The body is made out of cast steel, and the face is hardened. The body has three openings - 120° apart to assemble the jaws and operate them. Three pinions are fixed on the periphery of the body to operate the jaws by means of a chuck key. The body is hollow in cross-section. The crown wheel is housed inside the body.

Jaws

The jaws are made out of high carbon steel, hardened and tempered, which slide on the openings of the body. Generally there are two sets of jaws, viz. external jaws and internal jaws. External jaws are used for holding solid works. Internal jaws are used for holding hollow works. The steps on the jaws increase the clamping range. The back side of the jaws are cut out of scroll thread. Each jaw is numbered in a sequential manner, which will help in fixing the jaws in the corresponding numbered slots.

Crown wheel

The crown wheel is made out of alloy steel, hardened and tempered. On one side of the crown wheel a scroll thread is cut to operate the jaws and the other side is tapered on which bevel gear teeth are cut to mesh the pinion. When the pinion is rotated by means of the chuck key, the crown wheel rotates, thus causing the jaws to move inward or outward depending upon the rotation.

Pinion

The pinion is made out of high carbon steel, hardened and tempered. It is fitted on the periphery of the body. On the top of the pinion, a square slot is provided to accommodate the chuck key. It has a tapered portion on which the bevel gear teeth are cut, which match with the crown wheel.

Merits of a Four jaw chuck

- A wide range of regular and irregular shapes can be held.
- Work can be set to run concentrically or eccentrically at will.
- Has considerable gripping power; hence, heavy cuts can be given.
- The jaws are reversible for internal and external work.

- Work can be readily performed on the end face of the job.
- There is no loss of accuracy as the chuck gets worn out.
- Workpieces must be individually set.
- The gripping power is so great that a fine work can be easily damaged during setting.

Comparison between a 3 jaw chuck and 4 jaw chuck

Three Jaw chuck	Four Jaw chuck
Only cylindrical or hexagonal work can be held.	A wide range of regular and irregular shaped jobs can be held.
Internal and external jaws are available.	Jaws are reversible for external and internal holding.
Setting up of work is easy.	Setting up of work is difficult.
Less gripping power.	More gripping power.
Depth of cut is comparatively less.	More depth of cut can be given.
Heavier jobs cannot be turned.	Heavier jobs can be turned.
Workpieces cannot be set for eccentric turning.	Workpieces can be set for eccentric turning.
Concentric circles are not provided on the face.	Concentric circles are provided.
Accuracy decreases as chuck gets worn out.	There is no loss of accuracy as the chuck gets worn out.

Merits of a Three jaw chuck

- Work can be set quickly and trued easily.
- A wide range of cylindrical and hexagonal work can be held.
- Internal and external jaws are available.

Demerits of a Three jaw chuck

- Accuracy decreases as chuck gets worn out.
- Run out cannot be corrected.
- Only round and hexagonal components can be held.
- When accurate setting or concentricity with an existing diameter is required, a self-centering chuck is not used.

Specification of a chuck

To specify a chuck, it is essential to provide details of the:

- type of chuck
- capacity of the chuck
- diameter of the body
- width of the body
- the method of mounting to the spindle nose.

Examples

Three jaw self-centering chuck

Gripping capacity 450 mm

Diameter of the body 500 mm

Width of the body 125 mm

Tapered or threaded method of mounting

Chucks other than three jaw and four jaw types and their uses

Objectives : At the end of this lesson you shall be able to
- **explain the chucks other than the three jaw and four jaw types**
- **state their constructional features**
- **state the uses of each of these chucks.**

Apart from the four jaw independent chucks and self centering chucks, other types of chucks are also used on a centre lathe. The choice depends upon the component, the nature of the operation, the number of components to be machined.

Some of the other types of chucks are:

- two jaw concentric chuck
- combination chuck
- collet chuck
- magnetic chuck
- hydraulic chuck or air operated chuck.

Two jaw concentric chuck (Fig 1)

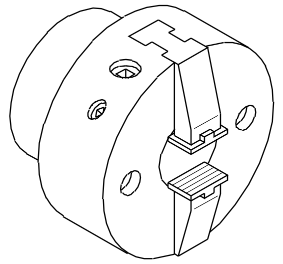
The constructional features of this chuck are similar to those of Three jaw and Four jaw chucks.

Each jaw is an adjustable jaw which can be operated independently. In addition to this feature, both jaws may be operated concentric to the centre. Irregular shaped works can be held. The jaws may be specially machined to hold a particular type of job.

Combination chuck

The combination chuck is normally a four jaw chuck in which the jaws may be adjusted either independently as done in a four jaw chuck, or together, as done in a three jaw universal chuck.

Fig 1



TWO JAW CONCENTRIC CHUCK

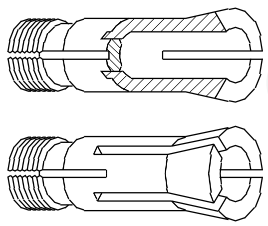
TD20N134131

This kind of chuck is used in places where duplicate workpieces are to be machined. One piece is accurately set as done in a four jaw chuck, and the subsequent jobs are held by operating the centering arrangement.

Collet chuck (Fig 2)

A collet is a hardened steel sleeve having slits cut partly along its length. It is held by a draw-bar which can be drawn in or out in the lathe spindle. The collet is guided in the collet sleeve, and held with the nose cap. It is possible to change the collet for different cross-sections depending on the cross-section of the raw material.

Fig 2



SPLIT COLLET

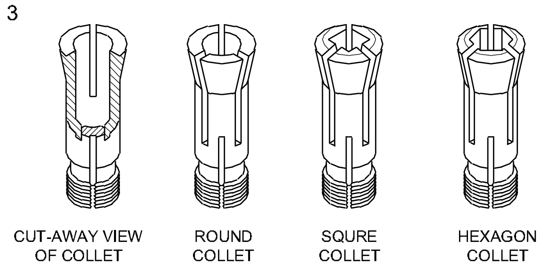
TD20N134132

There are three most commonly used types of collet chucks.

- Push-out chucks
- Draw-in chucks
- Dead length bar chucks

The operation of these chucks may be manual, pneumatic, hydraulic or electrical. They are mainly used to hold round, square, hexagonal or cast profile bars. (Fig 3)

Fig 3

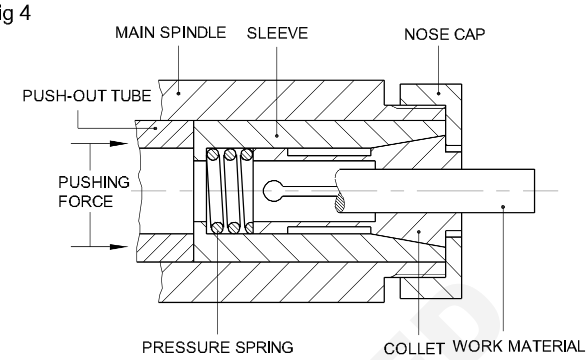


TD20N134133

Push-out chucks (Fig 4)

The collet closes on the workpiece in a forward direction and consequently an end-wise movement of the work results. The cutting pressure tends to reduce the grip of the collet on the workpiece.

Fig 4

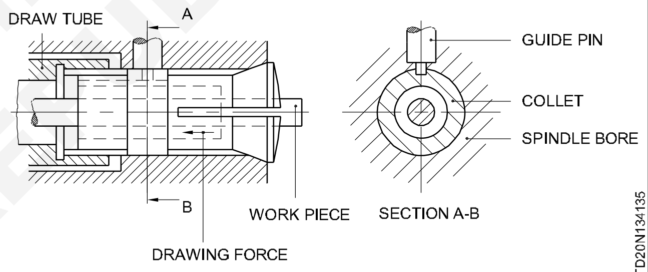


TD20N134134

Draw-in chuck (Fig 5)

The collet closes on the workpiece in a backward direction and movement of the work. Take special care to avoid errors of length due to this fault. The cutting pressure increases the grip of the collet on the workpiece.

Fig 5



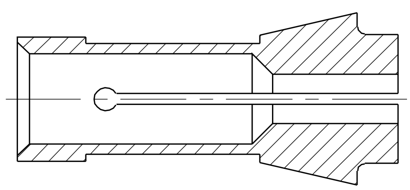
TD20N134135

Dead length bar chucks (Fig 6)

These chucks are widely used in modern machines as they provide an accurate end-wise location of the workpiece. The chuck does not move end-wise during gripping or closing operation. These chucks are made to hold round, hexagonal or square bars, and when they are not gripping, they maintain contact with the core thus preventing swarf and chips collecting between the collet and the core.

The disadvantage with these chucks is that each collet cannot be made to grip bars which vary by more than about 0.08 mm without adjustment.

Fig 6



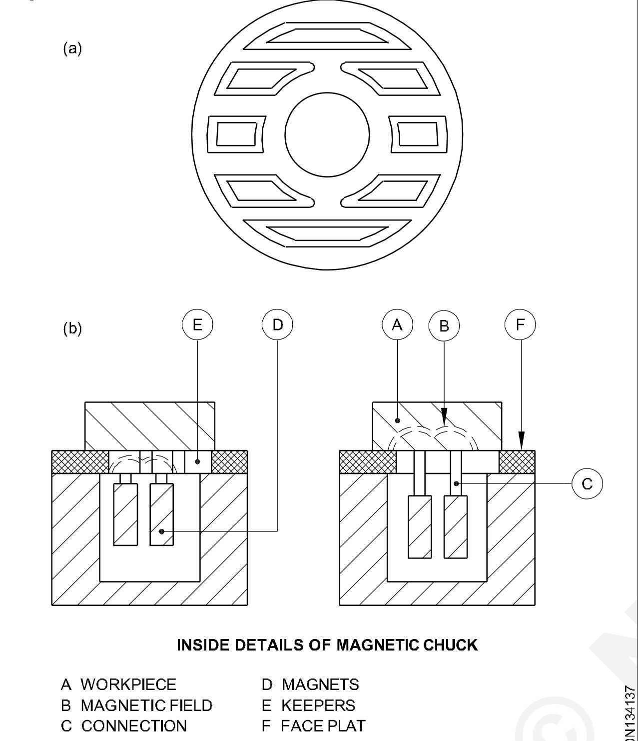
SPRING COLLET USED IN DEAD LENGTH BAR CHUCK

TD20N134136

Magnetic chuck (Figs 7a & 7b)

This chuck is designed to hold the job by means of magnetic force. The face of the chuck may be magnetized by inserting a key in the chuck and turning it to 180°. The amount of magnetic force may be controlled by reducing the angle of the key. The truing is done with a light magnetic force, and then the job is held firmly by using the full magnetic force.

Fig 7

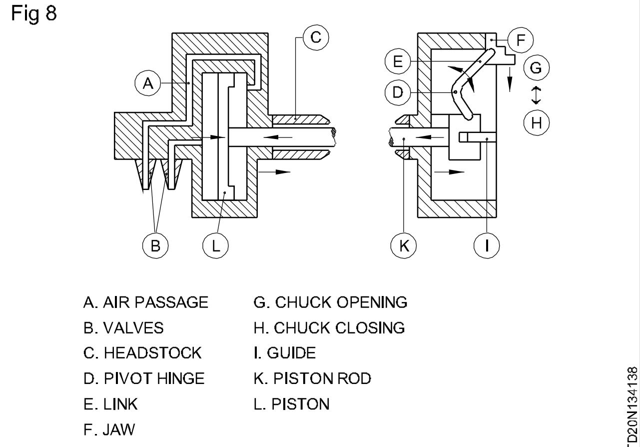


MAGNETIC CHUCK

Hydraulic chuck or air-operated chuck (Fig 8)

These chucks are mainly used for getting a very effective grip over the job. This mechanism consists of a hydraulic or an air cylinder which is mounted at the rear end of the headstock spindle, rotating along with it. In the case of a hydraulically operated chuck the fluid pressure is transmitted to the cylinder by operating the valves. This mechanism may be operated manually or by power. The movement of the piston is transmitted to the jaws by means of connecting rods and links which enable them to provide a grip on the job.

Fig 8



Uses of a two jaw concentric chuck

It is mainly employed to hold an irregularly shaped job. As the chuck is designed with two jaws, it can be used as a turning fixture.

Uses of a combination chuck

This chuck may be used both as a universal three jaw chuck and as a four jaw independent chuck. This chuck is very useful where duplicate workpieces are involved in the turning.

Uses of a collet chuck

It is mainly used for holding jobs within a comparatively small diameter. The main advantage of collets lies in their ability to centre work automatically and maintain accuracy for long periods. It also facilitates to hold the bar work.

Uses of a magnetic chuck

This type of a chuck is mainly used for holding thin jobs which cannot be held in an ordinary chuck. These are suitable for works where a light cut can be taken on the job.

Uses of hydraulic or air-operated chuck

These chucks are mainly used in mass production because of their speedy and effective gripping capacity.

Driving plate and face plate

Objectives : At the end of this lesson you shall be able to
- **explain between the features and the uses of different driving plates**
- **state the features and uses of different face-plates.**

When turning jobs in between centres, driving plates are used.

They are:

- catch plates
- driving plates
- safety driving plates.

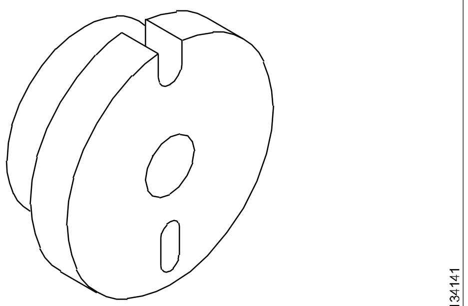
Catch plate

It is designed with a 'U' slot and an elongated slot to accommodate the bent tail of the lathe carrier. (Fig 1)

Driving plate with pin

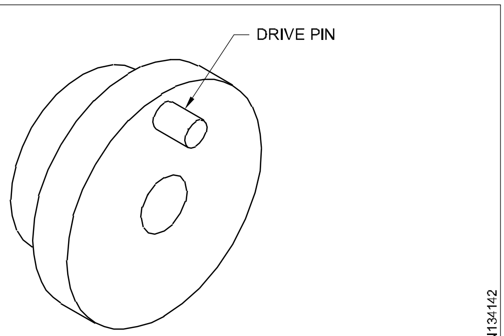
It is designed with a projected pin which locates the straight tail of the lathe carrier. (Fig 2)

Fig 1



CATCH PLATE

Fig 2

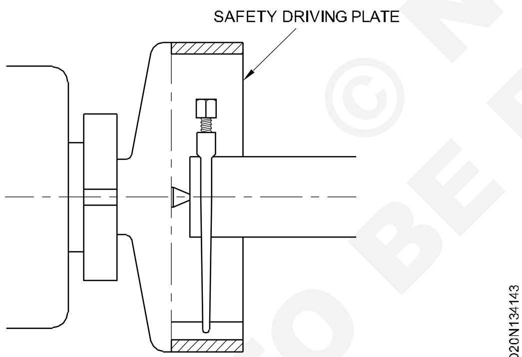


DRIVE PLATES

Safety driving plate

It is similar in construction to that of a driving plate but is equipped with a cover to protect the operator from any injuries. (Fig 3)

Fig 3



It is made of cast steel and is machined to have its face perfectly at right angles to the bore. It is provided with a stepped collar at the back. The bore is designed to suit the spindle nose to which the plate has to be mounted.

Uses

The driving plate with a straight tail carrier provides a positive drive for the workpiece.

Catch plates with bent tail carriers use a minimum length of the workpiece for clamping purposes.

A safety driving plate protects the operator from likely injuries.

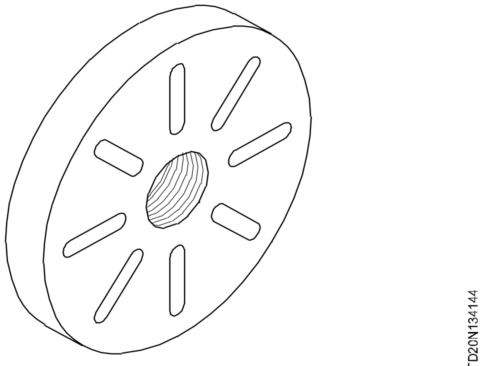
Face-plates

They are similar in construction to that of the lathe catch plates but are larger in diameter.

The different types of face-plates are:

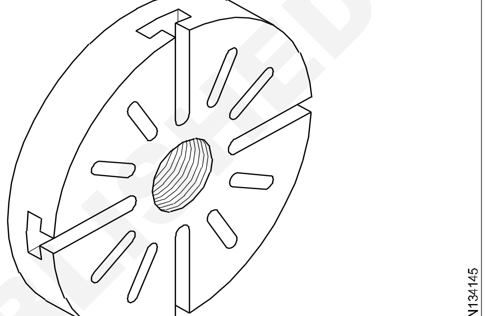
- face-plates with only elongated radial slots (Fig 4)

Fig 4



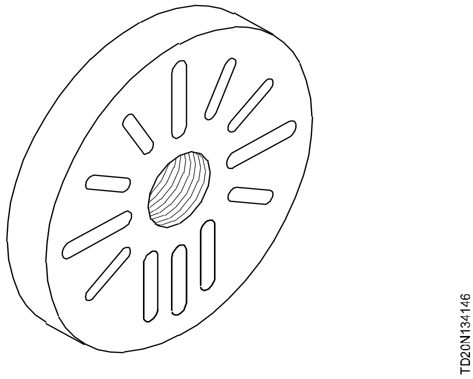
- face-plates with elongated slots and 'T' slots (Fig 5)

Fig 5



- face-plates with elongated radial slots and additional parallel slots. (Fig 6)

Fig 6



Face-plates are used along with the following accessories.

Clamps, 'T' bolts, angle plates, parallels, counterweights, stepped blocks, 'V' blocks etc.

Uses

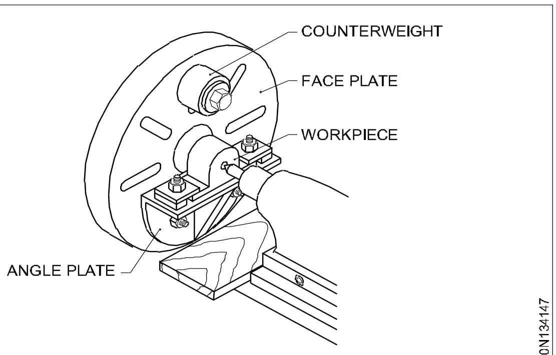
Large, flat, irregular shaped workpieces, castings, jigs and fixtures may be firmly clamped to a face-plate for various turning operations.

A work can be mounted on a face-plate while the face-plate is on the lathe spindle or on the workbench. If the workpiece is heavy or awkward to hold, the workpiece is mounted while the face-plate is on the workbench. Before mounting the face-plate and it is set up to the spindle, it is advantageous to locate the workpiece on the face-plate and centre the workpiece. Centre a punch mark or hole approximately on the face of the workpiece. This makes it easier to true the work after the face-plate is mounted on to the spindle.

The position of the bolts and clamps is very important, if a workpiece is to be clamped effectively.

If a number of duplicate pieces are to be machined, the face-plate itself can be set up as a fixture, using parallel strips and stop blocks.

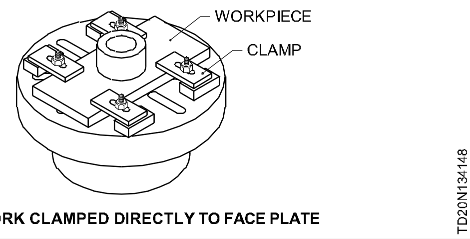
Fig 7



WORK HELD ON AN ANGLE PLATE

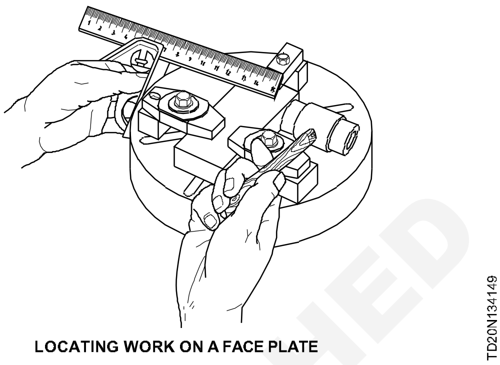
The application of the face-plate with the accessories in different set ups is shown in the sketches below. (Figs 7,8 and 9)

Fig 8



WORK CLAMPED DIRECTLY TO FACE PLATE

Fig 9



LOCATING WORK ON A FACE PLATE

Dismounting and mounting of chucks

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

- **explain how to dismount and mount chucks from a spindle nose.**

Depending upon the nature and delicacy of operators, different work-holding devices are to be mounted and dismounted on to the lathe spindle nose.

Dismounting a chuck

Switch off the motor.

Hold a ∅ÆÆ30x30mm iron piece projecting about 250mm more than the chuck to enable easy and safe lifting.

Set the spindle to the lowest speed.

Place a wooden block in the gap between the chuck body and the lathe bed. The thickness of the wooden block should be such that it can freely enter the above gap.

Loosen the chuck from the spindle nose. (Fig 1)

Slide and place the chuck on the wooden block. (Fig 2)

Clean and store the chuck.

Mounting a chuck

Aligning the chuck taper with the spindle nose taper and fastening with a screw type flange.

Fig 1

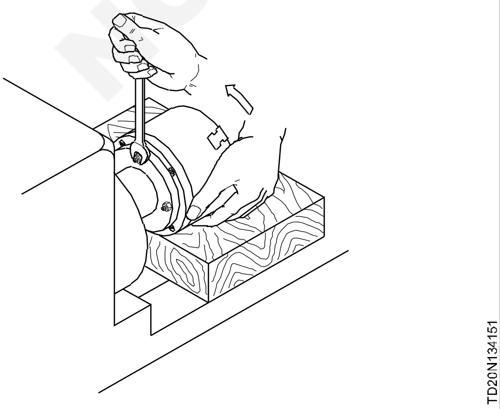
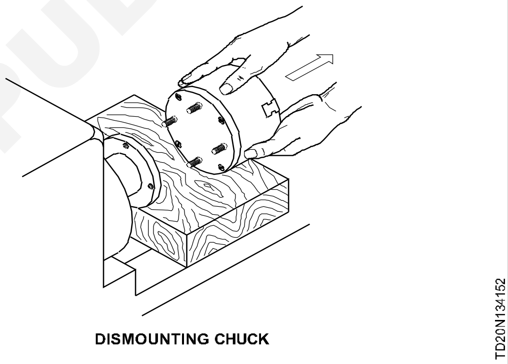


Fig 2

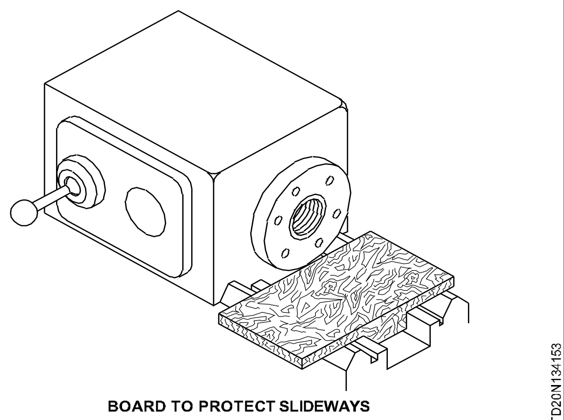


DISMOUNTING CHUCK

Aligning the taper of both the chuck and the spindle nose and fastening with nuts/bolts or with cams.

Place the wooden block which was used for dismounting the chuck on the bed near the spindle nose (Fig 3) before mounting the chuck.

Fig 3



BOARD TO PROTECT SLIDEWAYS

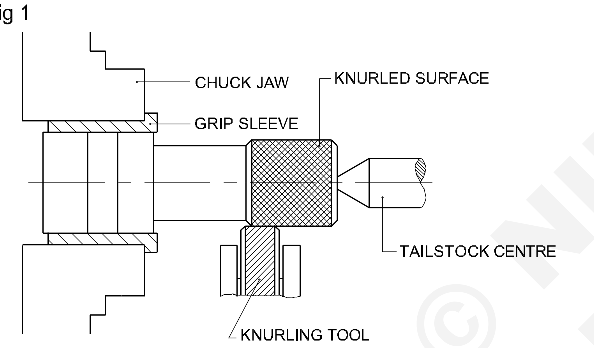
Knurling, meaning necessity types, grades of cutter and speed for knurling

Objectives: At the end of this lesson you shall be able to
- **define knurling operation**
- **state the purpose of knurling**
- **state the different types of knurls and knurling patterns**
- **explain the grades of knurls**
- **explain the various types of knurling tool-holders.**

Knurling (Fig 1)

It is the operation of producing straight lined, diamond shaped pattern or cross lined pattern on a cylindrical external surface by pressing a tool called knurling tool. knurling is not a cutting operation but it is a forming operation. Knurling is done at a slow spindle speed (1/3 times the turning speed). Soluble oil is to be used as coolant mostly and, sometimes straight cutting oil may be used to get better finish.

Fig 1



Purpose of knurling

The purpose of knurling is to provide.

- a good grip and make for positive handling.
- good appearance
- for raising the diameter to a small range for assembly to get a press fit.

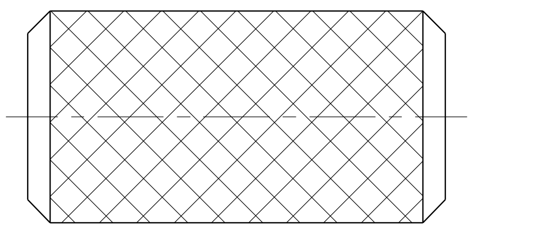
Types of knurls and knurling patterns

The following are the different types of knurling patterns.

Diamond knurling, Straight knurling, Cross knurling, Concave knurling and Convex knurling.

Diamond knurling (Fig 2)

Fig 2



It is a knurling of diamond shaped pattern. It is done by using a set of rolls. One roller has got right hand helical teeth and the other has left hand helical teeth.

Straight knurling (Fig 3)

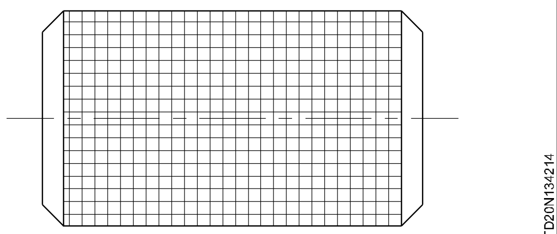
It is a knurling of straight lined pattern. This is done by using either a single roller or a double roller with straight teeth.

Fig 3

Cross knurling (Fig 4)

It is a knurling having a square shaped pattern. It is done by a set of rollers, one having straight teeth the other having teeth at right angles to the axis of knurl.

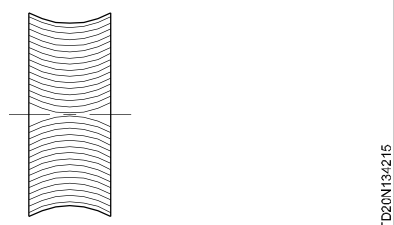
Fig 4



Concave knurling (Fig 5)

This is done by a convex knurl on a concave surface. This is done only by plunging the tool. The tool should not be moved longitudinally. The length of the knurling is limited to the width of the roller.

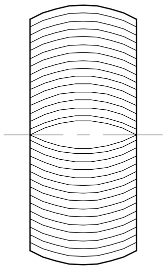
Fig 5



Convex knurling (Fig 6)

This is done by using a concave knurl of a convex surface. This is also done by plunging the tool.

Fig 6



Grades of knurling (Fig 7)

Knurling can be done in three grades.

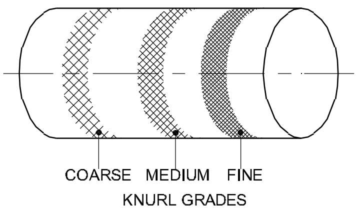
Coarse knurling, Medium knurling and Fine knurling

Coarse knurling is done by using coarse pitched knurls of 1.75mm pitch. (14 TPI)

Medium knurling is done by using medium pitched knurls of 1.25 mm pitch. (21 TPI)

Fine knurling is done by using fine pitched knurls of 0.75mm pitch. (33 TPI)

Fig 7



Types of knurling tool-holders

The different types of knurling tool-holders are:

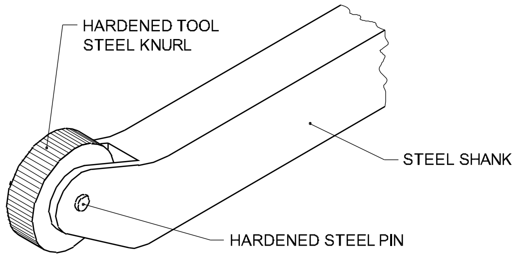
- single roller knurling tool-holders (parallel knurling tool-holders)
- knuckle joint type knurling tool-holders
- revolving types of knurlling tool-holders (universal knurling tool-holders).

A knurling tool-holder has a heat-treated steel shank and hardened tool steel knurls. The knurls rotate freely on hardened steel pins.

Single roller knurling tool-holder (Fig 8)

It has only one single roller which produces a straight lined pattern.

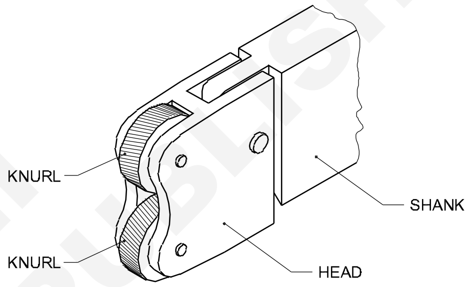
Fig 8



Knuckle joint type knurling tool-holders (Fig 9)

This tool holder has a set of two rollers of the same knurling pitch. The rollers may be of straight teeth or helical teeth. it is self-centering.

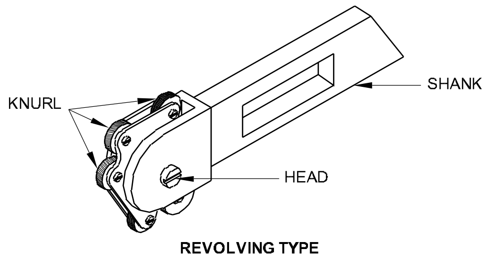
Fig 9



Revolving head knurling tool (Fig 10)

This tool-holder is also called a universal knurling tool-holder. It is fitted with 3 pairs of rollers having coarse, medium and fine pitches. These are mounted on a revolving head with pivots on a hardened steel pin. It is also self-centering.

Fig 10



Differences between different types types of knurling tool-holders are given in Table -1

Table -1

Single roller	Knuckle joint	Revolving type
Only one roller is used.	A pair of rollers are used.	Three pairs of rollers are used.
Only one pattern of knurling can be produced with this type of knurling tool-holder.	Cross or diamond knurling pattern can be produced.	Knurling patterns of different pitches can be produced.
It is not self-centering.	It is self-centering.	It is self-centering.

Knurling - Speed and Feed

The tables shown be used as a guide for determining the amount of end-feed or in-feed per revolution of the work. The rate of the feed for diamond pattern knurling is slower than that for straight or diagonal knurling.

Straight or Diagonal

End -FEED KNURLING

Approximate

FEED per REVOLUTION

T.P.I	Alum Brass	Mild Steel	Alloy Steel
12	.008”	.006”	.004”
16 - 20	.010”	.008”	.005”
25 - 35	.013”	.010”	.007”
40 - 80	.017”	.012”	.009”

Straight or Diagonal

IN - FEED KNURLING

Approximate

REVOLUTION

T.P.I	Alum Brass	Mild Steel	Alloy Steel
12	12	15	25
16 - 20	10	13	22
25 - 35	8	11	20
40 - 80	6	9	18

Vernier bevel protractor

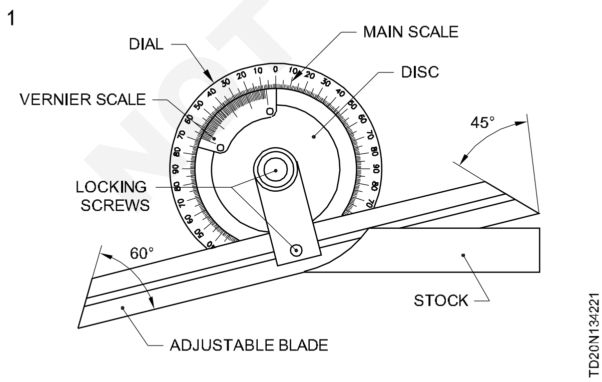
Objectives: At the end of this lesson you shall be able to
- **identify the parts of a universal bvel protractor**
- **state the functions of each part**
- **list out the uses of a vernier bevel protractor.**

The vernier bevel protractor is a precision instrument meant for measuring angles to an accuracy of 5 minutes. (5)

Parts

The following are the parts of vernier bevel protractor. (Fig 1)

Fig 1



Stock

This is one of the contacting surfaces during the measurement of an angle. Preferably it should be kept in contact with the datum surface from which the angle is measured.

Dial

The dial is an integrated part of the stock. It is circular in shape, and the edge is graduaged in degrees.

Disc

It is pivoted to the dial and can be rotated through 360°. The vernier scale of the instrument is attached to the disc. The disc is locked to the dial when reading the measurement.

Blade

This is the other surface of the instrument that contacts the work during measurement. It is fixed to the dial with the help of the clamping lever. A parallel groove is provided in the centre of the blade to enable it to be longitudinally positioned whenever necessary.

Locking screws

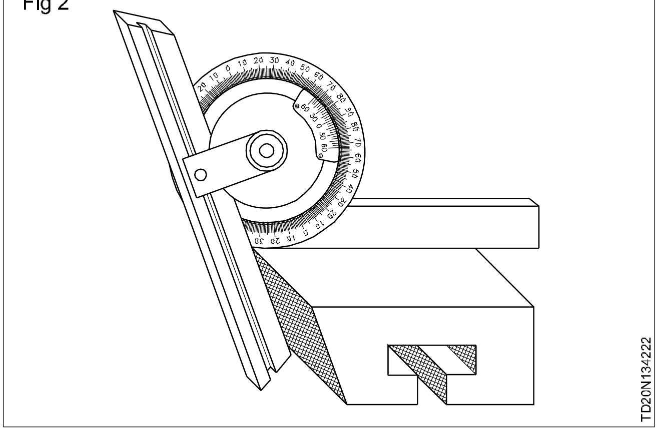
Two knurled locking screws are provided, one to lock the dial to the disc, and the other to lock the blade to the dial.

All parts are made of good quality alloy steel, properly heat-treated and highly finished. A magnifying glass is sometimes fitted for clear reading of the graduations.

Uses of a vernier bevel protractor

Apart from being used for measuring angles (Fig 2) a vernier bevel protractor is also used for setting work-holding devices on machine tools. Work-tables etc.

Fig 2



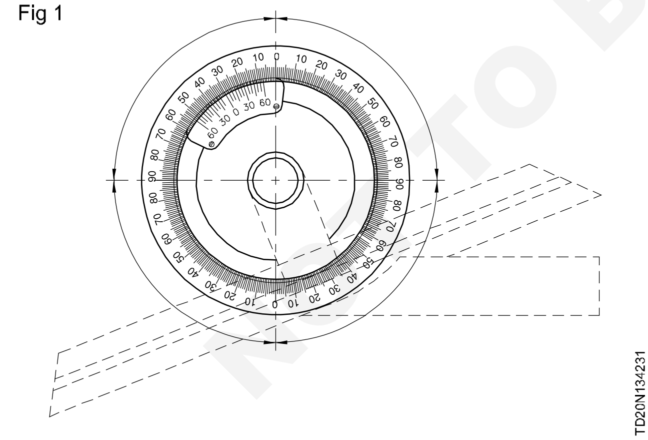
Graduations on universal bevel protractor

Objectives: At the end of this lesson you shall be able to
- **state the main scale graduations on the dial**
- **state the vernier scale graduations on the disc**
- **explain the least count of the vernier bevel protractor.**

The main scale graduations

For purposes of taking angular measurements, the full circumference of the dial is graduated in degrees. The 360° are equally divided and marked in four quadrants, from 0 degree to 90 degrees, 90 degress to 0 degree, 0 to 90 degrees and 90 degrees to 0 degrees. Every tenth division is marked longer and numbered. Each division represents 1 degree. The graduations on the dial are known as the main scale divisions. On the disc, 23 divisions spacing of the main scale is equally divided into 12 equal parts on the vernier. Each 3rd line is marked longer and numbered as 0, 15, 30, 45, 60. This constitutes the vernier scale. Similar graduations are marked to the left of 0 also. (Fig 1)

Fig 1



One vernier scale division (VSD) (Fig 2)

$$\frac{23^\circ}{12} = 1\frac{11^\circ}{12} = 1^\circ 55'$$

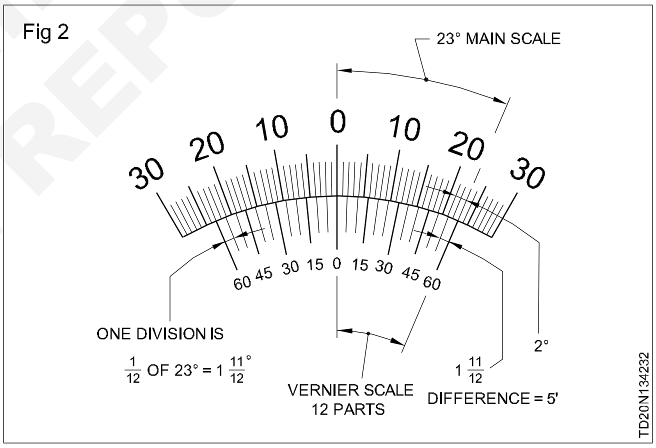
The least count of the vernier bevel protractor

When the zero of the vernier scale coincides with the zero of the main scale, the first division of the vernier scale will be very close to the 2nd main scale division. (Fig 2)

2 MSD - 1 VSD

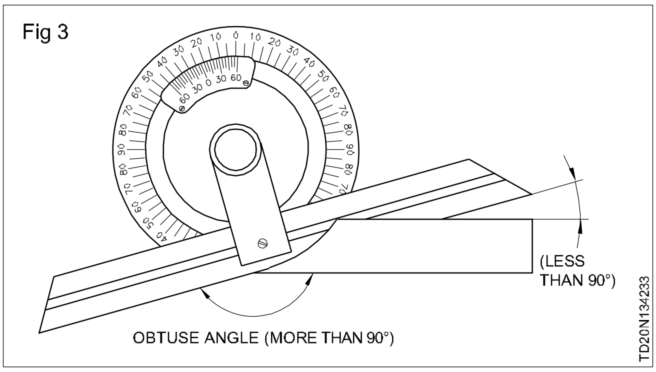
$$\text{i.e. the least count} = 2^\circ - \frac{23}{12} = \frac{1^\circ}{12} = 5'.$$

Fig 2



For any setting of the blade and stock, the reading of the acute angle and the supplementary obtuse angle is possible, and the two sets of the vernier scale graduations on the disc assist to achieve this. (Fig 3)

Fig 3



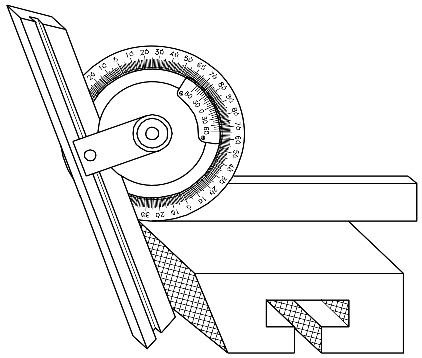
Reading of universal bevel protractor

Objectives: At the end of this lesson you shall be able to
- **explain acute angle setting**
- **explain obtuse angle setting.**

For reading acute angle set up (Fig 1)

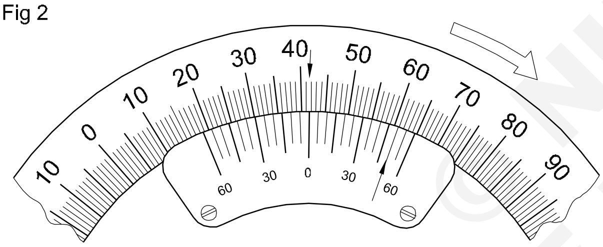
First read the number of whole degrees between zero of the main scale and zero of the vernier scale.

Fig 1



Note the line on the vernier scale that exactly coincides with any one of the main scale divisions and determine its value in minutes. (Fig 2)

Fig 2



To take the vernier scale reading, multiply the coinciding divisions with the least count

Example: 10 x 5' = 50'

Total up both the readings to get the measurements = 41° 50'.

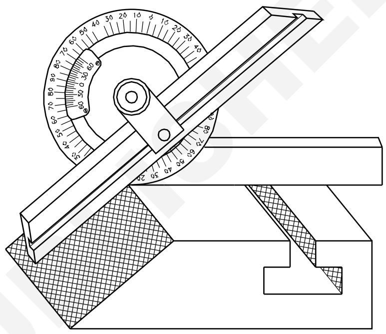
If you read the main scale in an anticlockwise direction, read the vernier scale also in an anticlockwise direction from zero.

If you read the main scale in a clockwise direction, read the vernier scale also in a clockwise direction from zero.

For obtuse angle set up (Fig 3)

The vernier scale reading is taken on the left side as indicated by the arrow. (Fig 4) The reading value is subtracted from 180° to get the obtuse angle value.

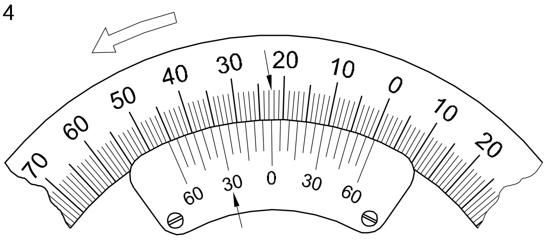
Fig 3



Reading 22 30'

Measurement 180° - 22°30' = 157°30'

Fig 4



Cutting Tool Materials

Objectives : At the end of this lesson you shall be able to
- **list the different cutting tool materials**
- **state the properties and uses of various types of cutting tool materials.**

Review Of Cutting Tool Materials: Various types of materials are used for making tools and tips. Each one has advantages and disadvantages.

High Carbon Steel (0.9% to 1.5% carbon)

This is useful for making cutting tools for light finishing cuts and for machining soft materials. It is quite tough,but the cutting edge softens and wears quickly, due to heat generated whilst cutting (at 250°C). 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 6000C. 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.

Satellite: This is a rather brittle, non-ferrous, cast alloy comprising of cobalt, chromium, tungsten and carbon (1.8 to 2.5%). 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 tantalum or niobium. It is the hardest cutting tool material normally used. It is capable of withstanding temperatures even above 1000°C. Several grades of cemented carbide tools are available, each tool 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. 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 be 3 to 4 times longer than a conventional carbide insert. Also, 40% higher cutting speeds can be used.

Ceramics: The latest development in metal cutting tools are the use of aluminum oxide, generally referred to as ceramics. Ceramic tools are made of aluminum 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 of that for cemented carbides and up to about 40 times of that for high speed steel tools. Heat conductivity of ceramics is very low. The ceramic 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 16500 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 0.05mm. To summarize, 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 with great accuracy on small diameters
- Turning small diameters, if the machine is not capable of running at a high rpm
- Screw cutting
- Intermittent cutting.

Cemented carbide tools may be used when:

- A fast and higher rate of metal removal is needed
- Cutting hard and non-ferrous materials
- High speed thread cutting is involved.

Comparative cutting speeds

Recommended speeds will vary according to the following factors: the kind and hardness of material being cut, the rate of feed, the depth of cut, the finish desired, the rigidity of the machine, the rigidity of the work set up, the type of cutting tool and the type of cutting fluid used. The lathe must be capable of running at high speeds, as very 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.

Properties of cutting tool materials

The cutting tool material must have the following properties

- Hot hardness
- Wear resistance

- Toughness
- Shock resistance
- Chemical stability
- Low friction
- Favourable cost

Brazing and grinding of carbide tipped tools

Objectives : At the end of this lesson you shall be able to
- **state what is brazing**
- **list the type of brazing**
- **brief the flux and filler in brazing**
- **explain the process of brazing**
- **Enumerate the grinding process of brazed carbide tipped tools.**

Brazing is a metal-joining process in which two or more metal items are joined together by melting and flowing a filler metal into the joint, with the filler metal having a lower melting point than the adjoining metal.

There are many heating methods available to accomplish brazing operations. The most important factor in choosing a heating method is achieving efficient transfer of heat throughout the joint and doing so within the heat capacity of the individual base metals used. The most commonly used heating methods are

– Torch brazing
– Furnace brazing
– Induction brazing
– Dip brazing
– Resistance brazing
– Vacuum brazing

Fluxing

Flux is a chemical compound applied to the joint surfaces before brazing. A coating of flux on the joint area guards the surfaces from the air, preventing oxide formation. It also dissolves and absorbs any oxides that form throughout heating or that was not completely removed in the cleaning process.

Filler

A variety of alloys are used as filler metals for brazing depending on the intended use or application method. In general, braze alloys are composed of three or more metals to form an alloy with the desired properties. The filler metal for a particular application is chosen based on its ability to wet the base metals,

Braze alloy is generally available as rod, ribbon, powder, paste, cream, wire and etc

The common types of filler metals used are

– Aluminum-silicon
– Copper
– Copper-silver
– Copper-zinc (brass)
– Copper-tin (bronze)
– Gold-silver
– Nickel alloy
– Silver

The basic process of brazing

Cleaning

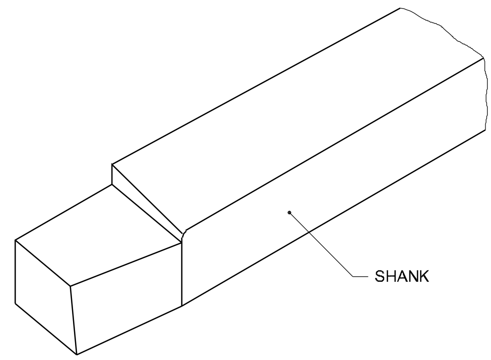
Cleaning the metal parts is to get rid of oil and grease. In most cases cleaning can be done by dipping the part into a suitable decreasing solvent, by vapor or by alkaline cleaning

If the metal surfaces are coated with oxide or scale, it can be removed by chemically or mechanically

For chemical cleaning use an acid pickle treatment

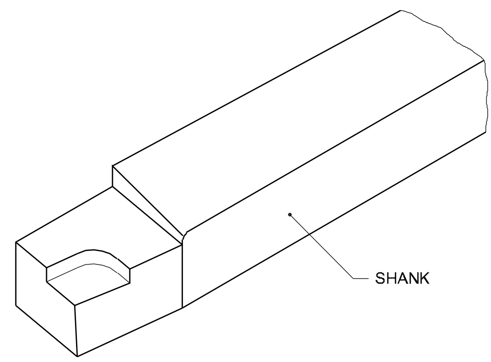
For mechanical cleaning use grinder or file and etc (Fig1)

Fig 1



Make arrangement for seating the carbide tip (Fig 2)

Fig 2



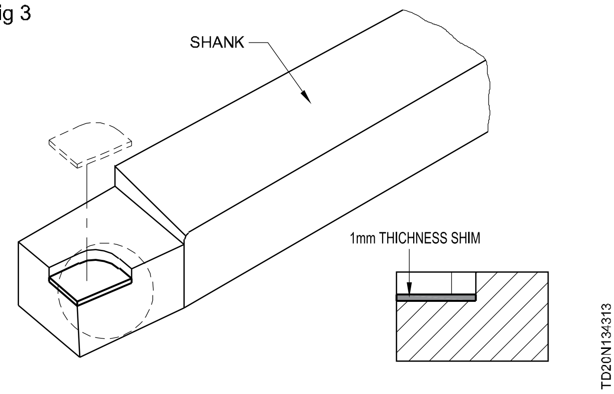
Once the parts are thoroughly cleaned, it is good practice to flux and braze as soon as possible. This avoids the chance of recontaminations

Fluxing

Coat the flux to the properly cleaned surface with a light layer of black flux. Black flux has more boron than white flux and works longer at higher temperature.

Cut the filler material (sheet) to the shape of carbide tip and place it in position on the tool shank as shown in Fig 3

Fig 3



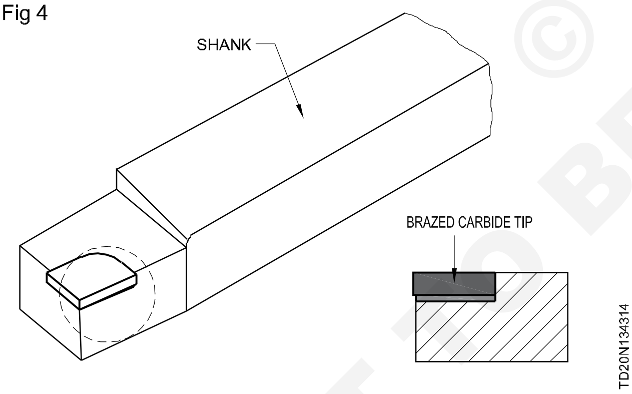
Apply a layer of flux on the filler sheet.

Place the clean carbide tip on the top, of the flux layer and coat the outside with a thick layer of flux Fig 4

Brazing

Hold the part in place with a ceramic rod while brazing.

Fig 4



Heat the whole part as evenly as possible until the whole joint is at the proper temperature. This is usually a dark cherry red. Remember that the part has to be hot enough all the way through.

Once the parts are hot enough you will see the alloy flow out of the joint. You may also feel the part move just a bit as the alloy turns liquid. Once the part is the right color (dark cherry red to cherry red) and the alloy is coming out just a bit (and it may feel mushy), then wiggle the part just a little bit to allow any flux or fumes to escape.

The torch flame is important both as a source of heat and as a way to protect the material. The flame needs to be a reducing flame, which means it, is a little oxygen starved.

Once the torch is removed the heating stops immediately and the plate begins to cool

The point to be made here is that the brazing process is a manual process but it is extremely precise and the time at full brazing temperature is very short, Considerably less than a second. This is important because part of successful brazing is to hold the materials at temperature long enough to form an intermediate compound in a layer between the two materials and the brazing alloy.

Slowly cool the carbide to minimize cracking of the carbide and clean the flux. The flux cleans of with worm water.

Grinding of brazed carbide tipped tools on tool grinder (Fig 5 &6)

Fig 5

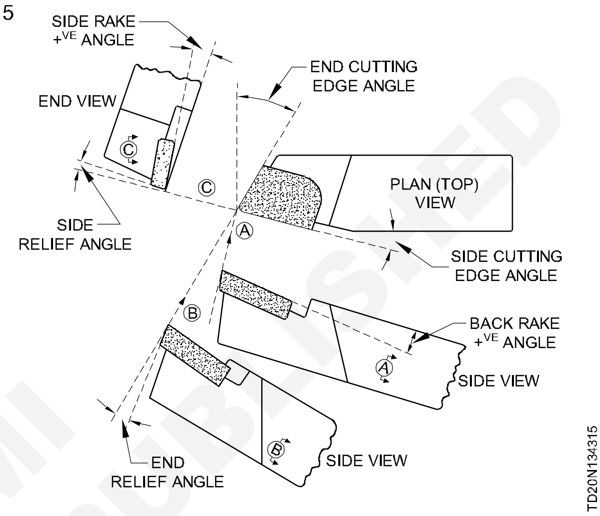
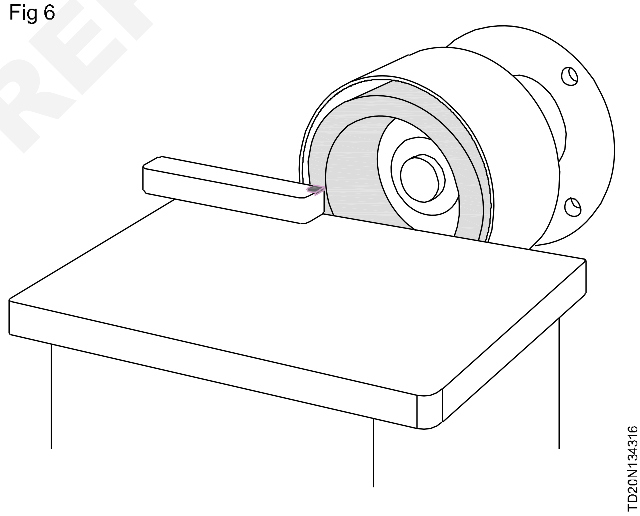


Fig 6



- Grind the side cutting edge and side relief angle on the tool shank using aluminum oxide grinding wheel
- Grind end cutting angle and end relief angle on the tool shank using aluminum oxide grinding wheel
- Grinding side cutting angle and side clearance angle on carbide tip using silicon grinding wheel
- Grinding end cutting angle and end clearance angle on carbide tip using silicon grinding wheel
- Grind nose radius and its clearance angle on carbide tip using diamond grinding wheel
- Lap the cutting edges using oil stone

Types of taper

Objectives : At the end of this lesson you shall be able to
- **state the uses of tapers**
- **identify the elements of a taper**
- **state the different standard tapers and their uses.**

A taper is uniform increase or decrease in diameter along the length of a cylinder.

Uses of taper

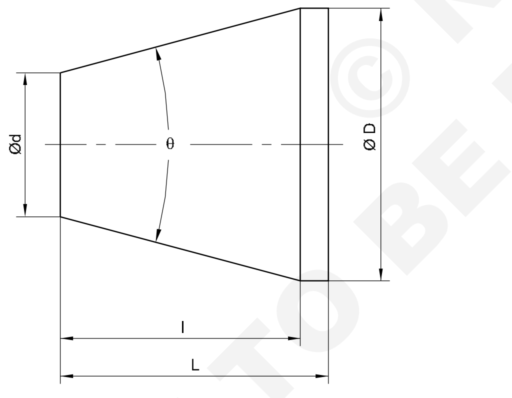
Tapers are used for the following.

- Assist to transmit drive in the assembled parts.
- Used for easy assembly and disassembly of parts.
- Gives self-alignment in the assembled parts.

Elements of taper (Fig 1)

Big diameter in mm	D
Small diameter in mm	d
Length of the taper in mm	ℓ
Included taper angle in degrees	θ
Total length of job in mm	L

Fig 1

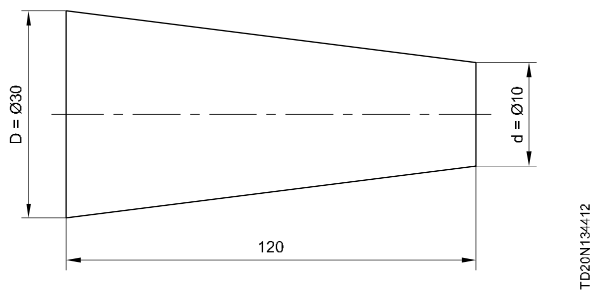


Expression of taper and its conversion

The methods by which tapers can be expressed are:

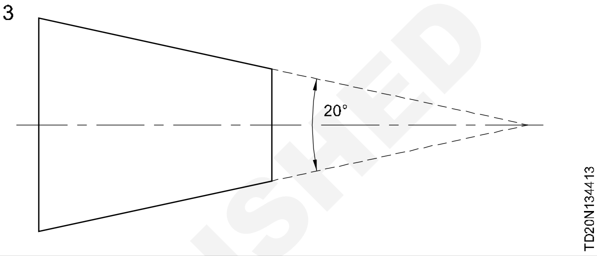
- giving the big dia. small dia. and the length of taper (Fig 2)

Fig 2



- giving the included angle of the taper in degrees. (Fig 3)
- giving Taper per foot (TPF)

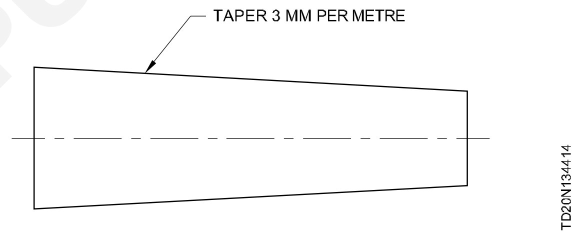
Fig 3



Example

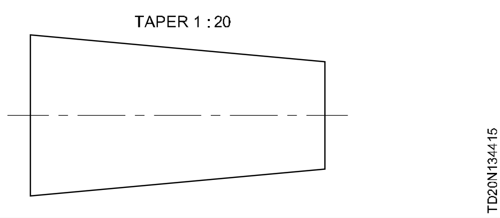
5/8" taper per foot means in a length of 12" taper (1 foot) the difference in diameter is 5/8" or mm per metre (Fig 4)

Fig 4



- giving taper in ratio - Ratio 1:20 means, for a taper length of 20 units the difference in diameter is 1 unit (Fig 5)
- mentioning by standard taper.

Fig 5



Example

MT3 (Morse taper number 3)

The relationship between the elements of a taper-

j = included angle of a taper

a = Half included angle of a taper

$$\text{Tan}\ \alpha = \frac{D - d}{2\ell}$$

$$\tan\alpha = \frac{TPF}{24} \text{ or } \frac{TPM}{2000}$$

$$\tan\alpha = \frac{Ratio}{2}$$

Classification of tapers

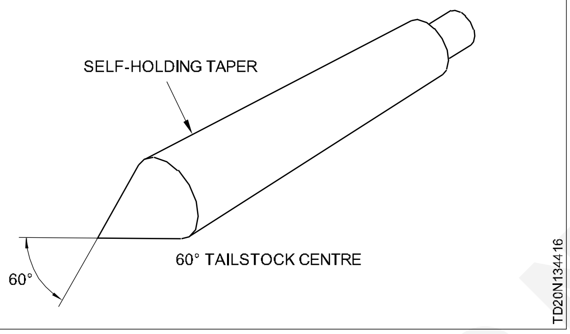
Tapers are classified as

- self-holding tapers
- quick releasing tapers.

Self-holding taper (Fig 6)

A self-holding taper has the property of holding the two parts together and be able to assemble together without any additional locking device such as keys. Just insert the internal taper into the external taper with a slight 'bang' and they get locked together. These tapers have a smaller taper angle that is limited to a maximum of 3°.

Fig 6



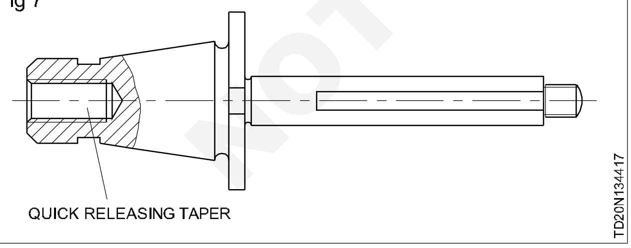
Example

Taper shank of drills, reamers and sleeves.

Quick releasing taper (Fig 7)

Quick releasing tapers in contrast to the self-holding tapers do not hold the parts together by themselves. They require additional locking devices for holding. (They have a larger included angle the value of which is not less than 18°. The purpose of quick releasing tapers is only to provide perfect alignment of the tool mating parts.)

Fig 7



Example

Arbor of milling machines.

Different standard tapers and their uses

The common standard tapers in use are:

- Morse taper (MT)
- Brown and Sharpe taper (BS)
- Jarno taper (JT)
- metric taper
- pin taper.

Morse taper

The Morse taper is the most commonly used standard taper in the industry. It is a self-holding taper. This taper is usually used in spindles of lathes and drilling machines, shanks of drills, reamers, centres, etc. The Morse taper is denoted by the letters MT. It is available from MT0 to MT7. The numbers MT0 to MT5 are commonly used on taper shanks of twists of drills, reamers and lathe centres. The included angle of the Morse taper is approximately 3° and the taper per foot is 5/8".

Brown and Sharpe taper

Both quick releasing and self-holding tapers are available in Brown and Sharpe tapers. The taper used in the arbors of a milling machine is a quick-releasing Brown and Sharpe taper, having a taper of 3 1/2" T.P.F.

Brown and Sharpe self-holding tapers are available from BSI to BS18. The taper per foot is 1/2" except BS10 which has a taper of 0.5161" taper per foot.

Jarno taper

Jarno tapers are self-holding and are used on external tapers of the lathe spindle nose where the chuck or face plate is mounted. It is available from Nos. 1 to 20. The amount of taper per foot is 0.6". The dimensions of this taper will be as follows.

$$\text{Big diameter of taper} = \frac{\text{Number}}{8}$$

$$\text{Small diameter of taper} = \frac{\text{Number}}{10}$$

$$\text{Length of taper} = \frac{\text{Number}}{2}$$

Jarno taper is mostly used in die-making machines.

Metric taper

Metric taper is available as both self-holding and quick-releasing tapers. A self-holding metric taper has an included angle of 2° 51' 51".

Quick releasing metric tapers are used as the external tapers of lathe spindle noses. Metric tapers are expressed by numbers which represent the big diameter of the taper in millimetres.

Standard pin taper

Standard pin tapers are used in taper pins. It is a self-holding taper. It is available both in Metric and British systems. The amount of taper is 1:50 in the metric system and 1:48 (1/4" TPF) in the British system.

Taper turning methods

Objectives: At the end of this lesson you shall be able to
- **state the methods of turning tapers on lathes**
- **state how each method is performed**
- **state the advantages and disadvantages of each method.**

Methods of turning taper on a lathe

The different methods of taper turning on a lathe are:

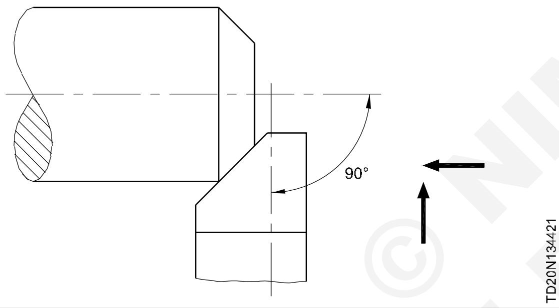
- form tool method
- compound rest method
- tailstock offset method
- taper turning attachment method
- taper turning by combining feeds.

Form tool method (Fig 1)

This method is used in mass production for producing small lengths of taper. The form tool should be set at right angles to the axis of the work and feed.

The carriage should be locked while turning taper by this method.

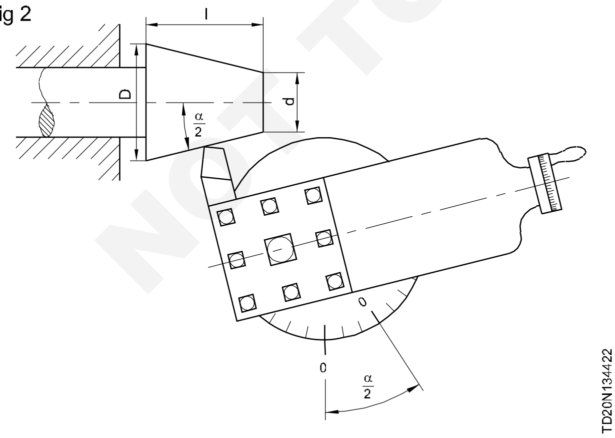
Fig 1



Compound rest method (Fig 2)

In this method, the compound rest is swivelled to half the included angle of the taper, and the taper is turned by feeding the top slide.

Fig 2



The angle '$\frac{\alpha}{2}$' to which the compound rest is set is found by the formula

$$\tan\frac{\alpha}{2} = \frac{D-d}{2\ell}$$

where

D = big diameter of taper

d = small diameter of taper

ℓ = length of taper

$\frac{\alpha}{2}$ = 1/2 included angle in degrees.

Advantages

- Both internal and external tapers can be produced.
- Steep tapers can be produced.
- Easy setting of the compound rest.

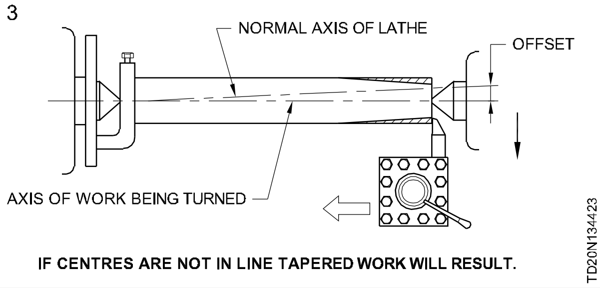
Disadvantages

- Only hand feed can be given.
- Threads on the taper portion cannot be produced.
- The taper length is limited to the movement of the top slide.

Tailstock offset method (Fig 3)

In this method the job is held at an angle, and the tool moves parallel to the lathe axis. The body of the tailstock is shifted on its base to an amount corresponding to the angle of the taper.

Fig 3



These tapers 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

$$\text{offset} = \frac{(D-d)L}{2\ell}$$

where

D = big diameter of taper

d = small diameter of taper

ℓ = taper length

L = total length of the job.

Advantages

- Power feed can be given.
- Good surface finish can be obtained.
- Maximum length of 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 the work is held between centres only.
- Damages the centre drilled holes of the work.
- The alignment of the lathe centres will be disturbed.

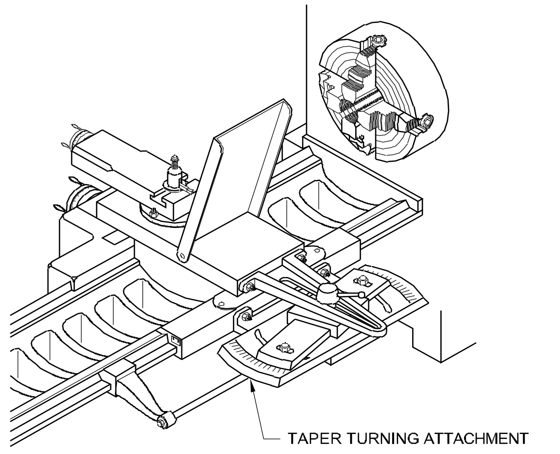
Taper turning attachment method (Fig 4)

A special 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 to the required angle by the attachment.

Advantages

- Both internal and external tapers can be produced.
- Threads on both internal and external taper portions can be cut.

Fig 4



- Power feed can be given.
- Lengthy taper can be produced.
- Good surface finish is obtained.
- The alignment of 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

- Use is limited to turning of slow taper angles only.

Calculation of the compound slide swivel angle

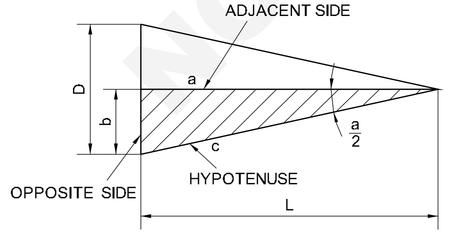
Objectives : At the end of this lesson you shall be able to

- **derive a formula to determine the swivel angle**
- **solve problems involving taper calculation**
- **refer to tables and determine the value of the angle for the arrived result**
- **determine the depth of cut to reduce the taper length**

Derivation of the formula

For convenience a tapered job whose small diameter is zero is taken (Fig 1) to illustrate as to how the formula can be derived.

Fig 1



The taper is divided into two angled triangles by the centre line. By referring to the shaded right angled triangle in figure 1, the side (b) shown against the half included angle of taper a/2, is termed as the opposite side. The side (a) is termed as the adjacent side and side (c) is termed as the hypotenuse.

There is a relationship between the sides of the triangle and the a/2. They can be expressed as ratios. The ratio of the sides (b) and (a) is a constant value for a given angle a/2. This ratio b/a does not change for a given value of a/2. This means that if 'b' increase or decrease there will be a proportionate increase or decrease of side 'a' making the ratio b/a constant. This ratio between the opposite side to the adjacent side of an angle in a right angled triangle is referred to as the tangent value of the angle.

The equation for the tangent 2/2 is therefore, Tan 2/2= b/a. Since this value is the same for a particular angle, the tangent values for all angles are put together into tables under the heading 'Natural Tangents'. Therefore, they need no longer be calculated individually, but can be taken from the tables.

Referring to Fig 2, which has a small diameter also, the shaded triangle D-d refers to 'b' of the formula and l refers to 'a' of the formula.

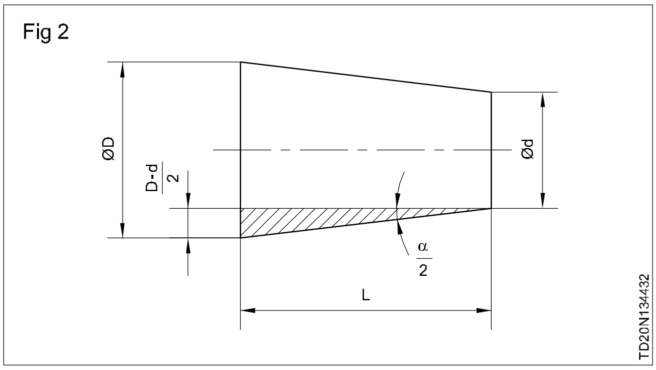


Fig 2

D = 30 mm d = 22 mm & l =40 mm

Now the formula becomes

$$\text{Tan } \alpha/2 = \frac{\frac{D-d}{2}}{l} = \frac{D-d}{2xl} = \frac{D-d}{21}.$$

For example, referring to Fig 3 we have

$$\text{Tangent } \alpha/2 = \frac{D-d}{2l} = \frac{30-22}{80}$$

$$= \frac{8}{80} = \frac{1}{10} = 0.1.$$

Referring to the logarithm tables of Natural Tangents we find that the angle whose tangent value is O.1, is 5°. 45', and this is the top slide swivelling angle to turn the tapered job of Fig 3.

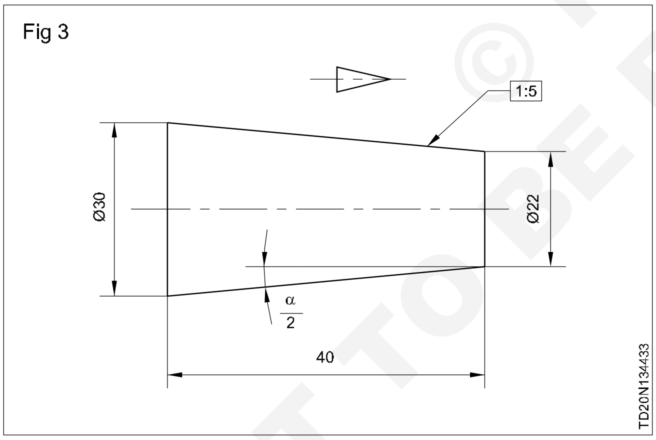


Fig 3

Taper expressed as a ratio to determine the swivel angle

The general formula is $\text{Tan } \frac{\alpha}{2} = \frac{D-d}{2l}$

This can be rewritten as

$$\text{Tan } \frac{\alpha}{2} = \frac{D-d}{l} x \frac{1}{2}$$

This $\frac{D-d}{l}$ is the taper ratio

Hence the formula becomes

$$\text{Tan of half the included angle} = \frac{\text{Taper ratio}}{2}.$$

The taper ratio is given as 1:5

$$\text{Tan of half included angle of taper} = \frac{\text{Taper ratio}}{2}.$$

$$\text{Tan } \frac{\alpha}{2} = \frac{1/5}{2} = \frac{1}{10} = 0.1$$

$$\frac{\alpha}{2} = 5° \, 45'.$$

The compound slide swivel angle is 5° 45'.

Taper per foot is given to determine the compound slide swivelling angle.

Example

(Given 5/8" TPF)

This means that the different in diameter (D-d) is 5/8" for taper length of 1 foot or 12".

$$\text{Tan } \alpha/2 = \frac{D-d}{2l}$$

There D - d =5/8" and l = 12"

$$\text{Tan } \alpha/2 = \frac{\frac{5"}{8}}{2 \times 12} = \frac{5}{8x24} = 0.0260$$

$\alpha/2 = 1°26'$

$$\text{The formula is Tan of half included} = \frac{\text{Taper per foot}}{24}$$

Remember that it is half included angle of the taper to which the tops slide is to be swivelled

To determine the depth of cut to be given to get a definite in length of the taper, the taper angle remaining same. (Fig 4)

Referring to Fig 4, 9 is the radius at the bigger end, (also the difference in diameter divided by 2, since the small diameter of the taper is zero) 5 is the length of the taper, 4 is the change in the taper length, 1 is the depth of cut to be given to get the change in taper length.

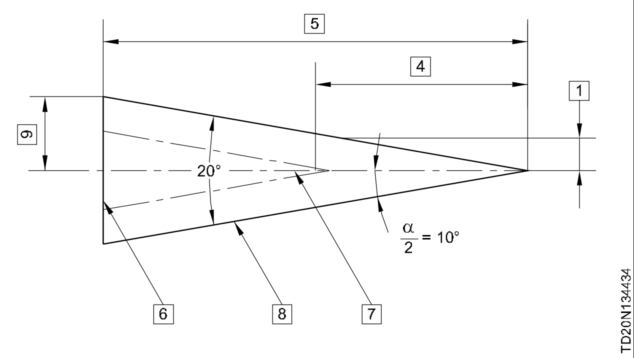
6 Opposite side to $\frac{\alpha}{2}$

7 Adjacent side

8 Hypotenuse

Then l = 4 x tan a/2.

Fig 4



Example: The taper length 5 of Fig 4 with an included angle of 20° is to be shortened by 2 mm. What should be the depth of cut?

$$I = 4 \times \tan \frac{\alpha}{2}$$

$$1 = 2\ mm \times \frac{\tan 20°}{2}$$

=2 mm x tan 10°

=2 x 0.1763

= 0.3526 mm.

Hence a depth of cut of 0.35 mm is to be given in order to reduce the taper length by 2 mm, the taper included angle remaining the same 20°.

Calculation of taper turning by offsetting tailstock

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

- **describe the amount of offset by experessing the diameter**
- **describe the amount of offset by expressing TPF.**

Calculation of the amount of offset

If the taper is expressed by giving the big diameter (D) the small diameter (d) the length of taper (I), then

$$\text{offset} = \frac{(D - d) \times L}{2I}$$

Where L = total length of job

Example

The big diameter of a tapered job (D) = 30 mm.

The small diameter of the tapered job (d) = 26 mm.

The length of taper portion (I) = 100 mm.

Total length of job (L) = 200 mm

$$\text{offset} = \frac{(D - d) \times L}{2I}$$

$$= \frac{(30 - 26) \times 200}{2 \times 100}$$

$$= \frac{4 \times 200}{2 \times 100}$$

= 4 mm

If the taper is expressed in TPF then the amount of offset

$$= \frac{TPF \times L}{2}$$

where TPF is given in inches

L = total length of job

If taper is expressed as a ratio then the amount of offset

$$= \frac{ratio \times L}{2}$$

If taper is expressed by included angle i.e. 2

Offset = L x tan

where L = total length

= 1/2 included angle in degrees.

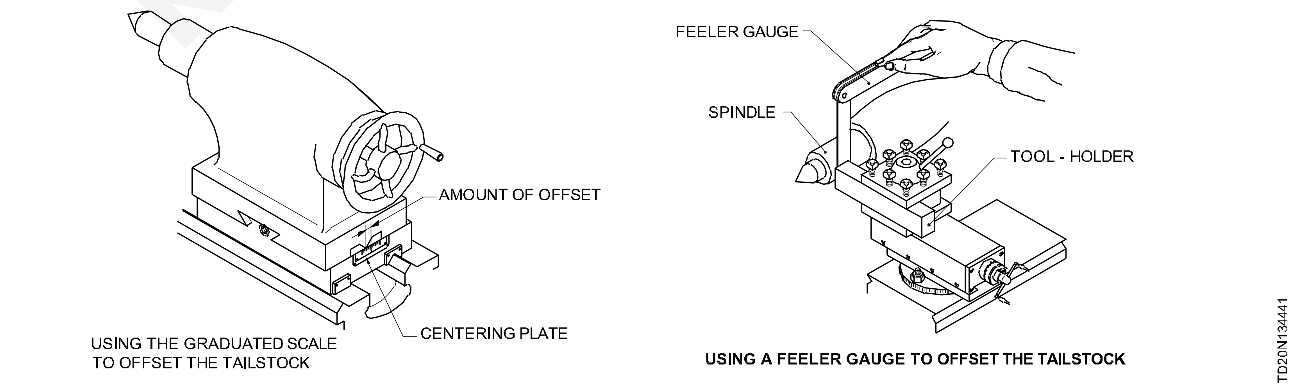
Different methods of offsetting the tailstock (Fig 1)

Setting offset with the help of the inside measuring jaws of a vernier caliper to the required mm, if direct graduation is not provided on the base of the tailstock.

Using a dial test indicator.

Using a cross-slide graduated collar and feeler gauge.

Fig 1

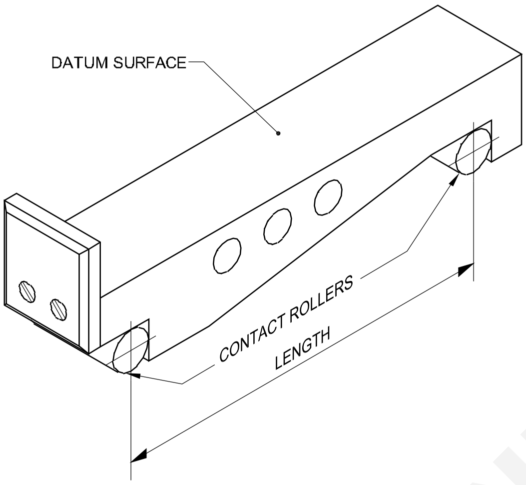


Sine bar - Description and uses

Objectives: At the end of this lesson you shall be able to
- **state the principle of a sine bar**
- **state the sizes of sine bars**
- **state the features of sine bars**
- **state the different uses of sine bars.**

A sine bar is a precision measuring instrument for checking and setting of angles. (Fig 1)

Fig 1

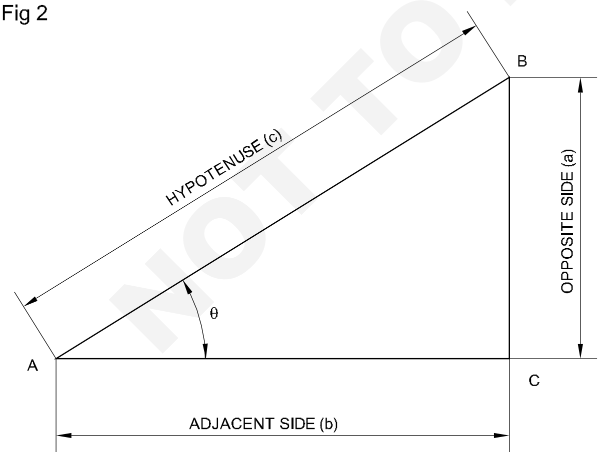


The principle of a sine bar

The principle of a sine bar is based on the trigonometrical function.

In a right angled triangle the function known as Sine of the angles is the relationship existing between the opposite side to the angle and the hypotenuse. (Fig 2)

Fig 2

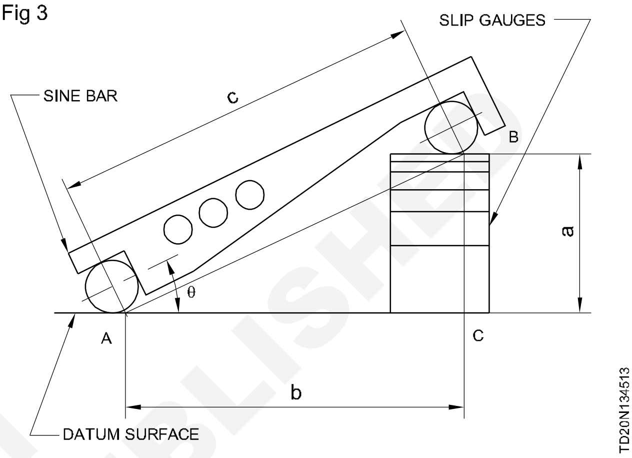


It may be noted that for setting the sine bar to different angles, slip gauges are used.

A surface plate or marking table provides the datum surface for the set up.

The sine bar, the slip gauges and the datum surface upon which they are set to form a right angled triangle. (Fig 3)

Fig 3



The sine bar forms the hypotenuse (c) and the slip gauge stack forms the side opposite (a).

Sin of the angle $\theta = \dfrac{\text{Opposite side}}{\text{hypotenuse}}$

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

Features

This is a rectangular bar made of stabilized chromium steel.

The surfaces are accurately finished by grinding and lapping.

Two precision rollers of the same diameter are mounted on either end of the bar. The centre line of the rollers is parallel to the top face of the sine bar.

There are holes drilled across the bar. This helps in reducing the weight, and also it facilitates clamping of sine bar on angle plate.

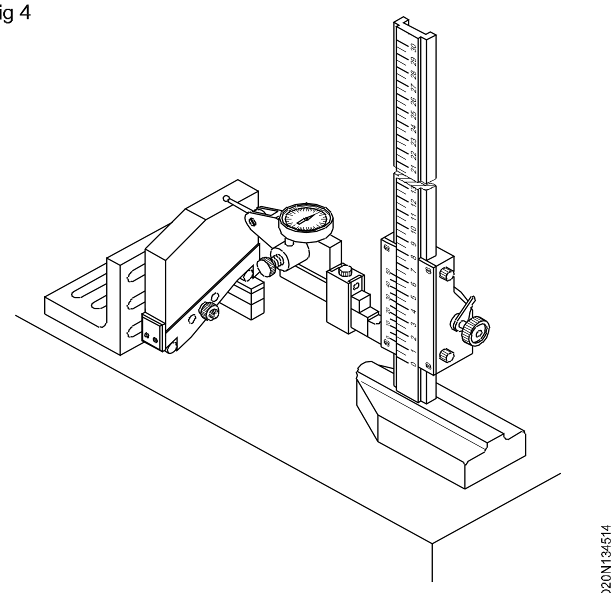
The length of the sine bar is the distance between the centres of the rollers. The commonly available sizes are 100 mm, 200 mm, 250 mm and 500 mm. The size of a sine bar is specified by its length.

Uses

Sine bars are used when a high degree of accuracy to less than one minute is needed for

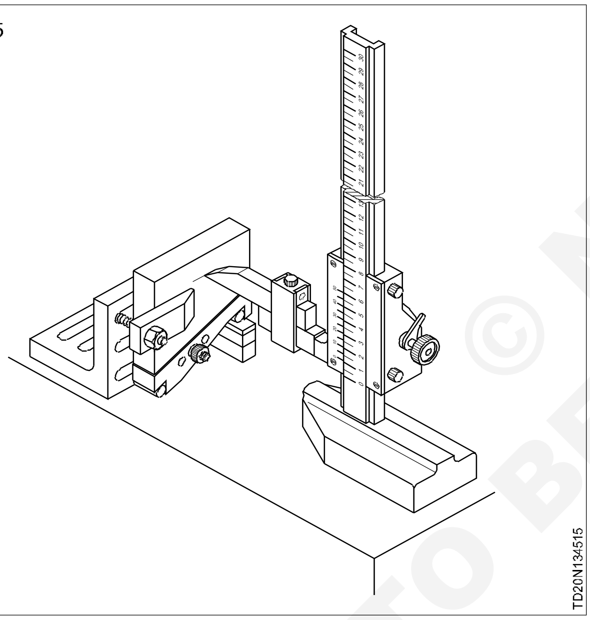
– measuring angles (Fig 4)

Fig 4



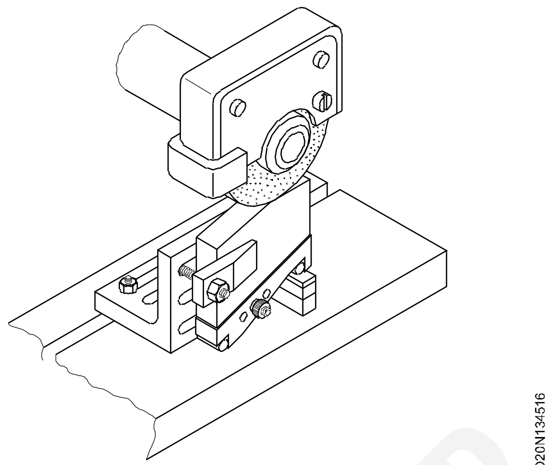
– marking out (Fig 5)

Fig 5



– setting up for machining. (Fig 6)

Fig 6



Determining taper using sine bar and slip gauges

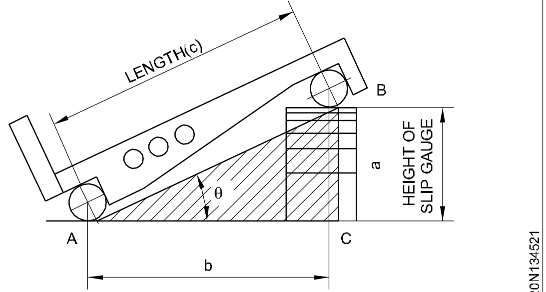
Objectives: At the end of this lesson you shall be able to

- **state correctness of a known angle**
- **explain the height of slip gauges to a known angle.**

Sine bars provide a simple means of checking angles to a high degree of accuracy of not less than one minute upto 450.

The use of a sine bar is based on trigonometric function. The sine bar forms the hypotenuse of the triangle and the slip gauges form the opposite side. (Fig 1)

Fig 1



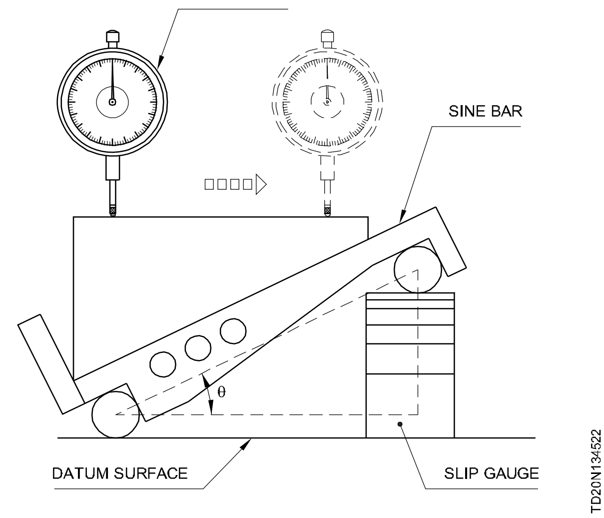
Checking the correctness of a known angle

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 the roller. (Fig 1)

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 in the figure and the dial is set to zero.

Fig 2



Move the dial 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 zero on both ends. The actual angle can then be calculated and the deviation, if any, will be the error.

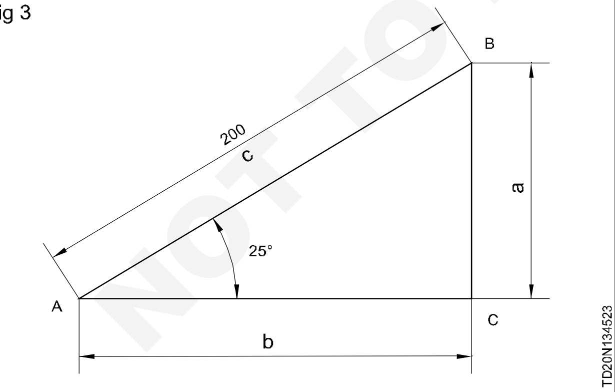
Method of calculating the slip gauge height

Example (Fig 3)

Exercise 1

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

Fig 3



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

$\theta = 25^\circ$

a = C Sine q

= 200 x 0.4226

a = 84.52 mm

The height of the slip gauge required is 84.52 mm.

The value of sinq can be obtained from mathematical tables. (Natural trigonometrical functions)

Tables are also available with readily worked out sine bar constants for standard sine bar lengths.

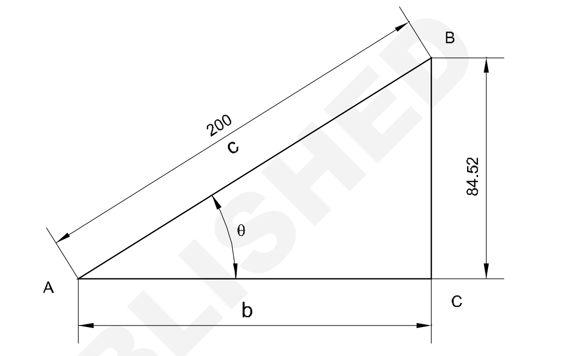
Calculating the angle for tapered components

Example 2

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

What will be the angle of the component? (Fig 4)

Fig 4



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

$$= \frac{84.52}{200}$$

Sine q = 0.4226

The angle whose sine value is 0.4226 is 25°. Hence the angle of tapered component is 25°.

Classroom Assignment

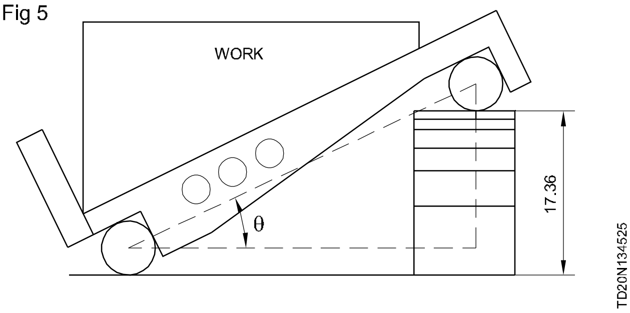
1 What will be the angle of the workpiece if the slip gauge pack height is 17.36 mm and the size of the sine bar used is 100 mm? (Fig 5)

Answer ________________________

2 Calculate the height of the slip gauge pack to raise a 100 mm sine bar to an angle of $3^\circ\ 35^1$.

Answer ________________________

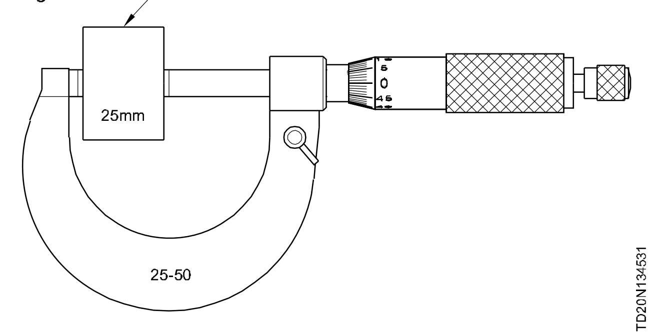
Fig 5



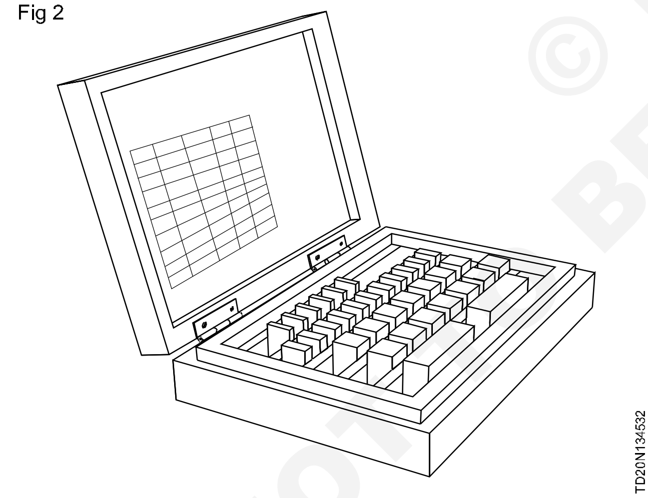
Slip gauges, types uses and selection

Objectives: At the end of this lesson you shall be able to
- **state the features of slip gauges**
- **state the different grades of slip gauges and their uses**
- **state the number of slips in standard sets**
- **state the precautions to be followed while using slip gauges.**

Slip gauges or gauge blocks are used as standards for precision length measurement. (Fig 1) These are made in sets and consist of a number of hardened blocks, made of high grade steel with low thermal expansion. They are hardened throughout, and heat treated further for stabilization. The two opposite measuring faces of each block are lapped flat and parallel to a definite size within extremely close tolerances.



These slip gauges are available in various sets with different numbers. (Fig 2) (Ref.Table 1)



A particular size can be built up by wringing individual slip gauges together. (Figs 3 & 4)

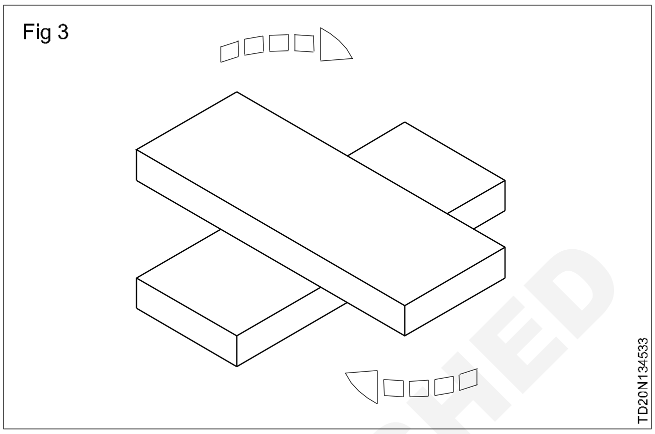
Wringing is the act of joining the slip gauges together while building up to sizes.

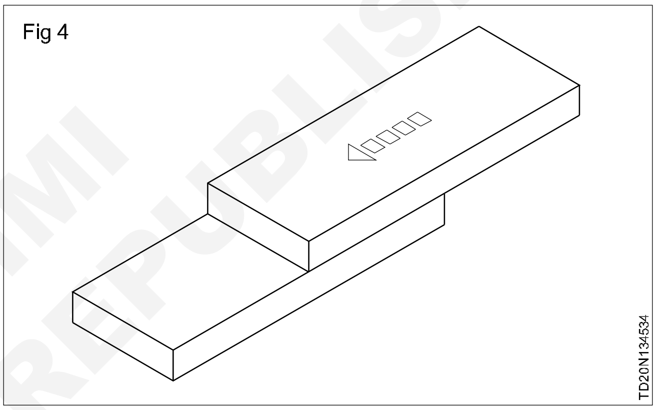
steel or tungsten carbide. These are used for protecting the exposed faces of the slip gauge pack from damage while in use.

Grades

The following four grades of slip gauge are recommented as per IS 2984 -1981.

Grade '00' , Grade '0' Grade '1' & Grade '2'





Grade '00' shall normally be used for calibration purposes. It shall not be used in combination.

Grades 0,1 and 2 are intended for general use.

Grade '0' is used only for calibration of Inspection Grade '1', Grade '1' is used in tool room and standards room. It is also used for calibration of Grade '2'.

Grade '2' is used on machines for setting purposes and on surface plate for inspection purposes in shop floor. It is also used for zero setting of precision measuring instruments in shop floor. **For further details refer IS 2984 - 1981**

Some sets of slip gauges also contain protector slips of made to standard thickness from higher wear-resistant steel or tungsten carbide. These are used for protecting the exposed faces of the slip gauge pack from damage while in use.

Grades

The following four grades of slip gauge are recommented as per IS 2984 -1981.

Grade '00' , Grade '0' Grade '1' & Grade '2'

Grade '00' shall normally be used for calibration purposes. It shall not be used in combination.

Grades 0,1 and 2 are intended for general use.

Grade ‘0’ is used only for calibration of Inspection Grade ‘1’, Grade ‘1’ is used in tool room and standards room. It is also used for calibration of Grade ‘2’.

Grade ‘2’ is used on machines for setting purposes and on surface plate for inspection purposes in shop floor. It is also used for zero setting of precision measuring instruments in shop floor. **For further details refer IS 2984 - 1981**

B.I.S. recommendations

Three grades of slip gauges are recommended as per IS - 2984. They are:

Grade ‘00’

Grade ‘0’

Grade 1

Grade 2.

Care and maintenance points to be remembered while using slip gauges

(Avoid handling the slip gauges with bare hands since this affects the size of them due to heating).

Use a minimum number of blocks as far as possible while building up a particular dimension.

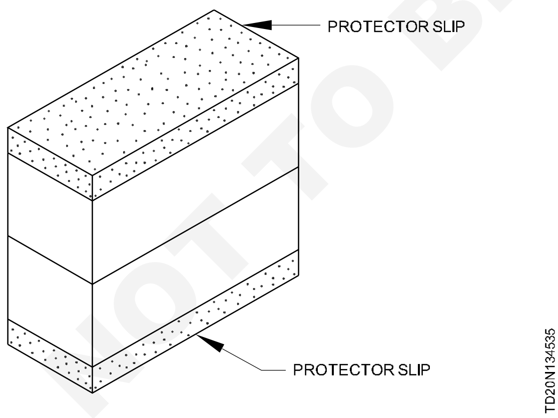
While building the slip gauges, start wringing with the largest slip gauges and finish with the smallest.

While holding the slip gauges do not touch the lapped surfaces.

If available use protector slips on exposed faces.(Fig 5)

After use, clean the slips with carbon tetrachloride and apply petroleum jelly for protection against rust.

Fig 5



Before use, remove petroleum jelly with carbon tetrachloride. Use chamois leather to wipe the lapped surfaces.

Table 1

Set of 112 pieces (M 112)

Range (mm)	Steps (mm)	No.of pieces
Special piece 1.0005	-	1
1st series 1.001 to 1.009	0.001	9
2nd series 1.01 to 1.49	0.01	49
3rd series 0.5 to 24.5	0.5	49
4th series 25.0 to 100.0	25.0	4
Total pieces		**112**

Set of 103 pieces (M 103)

Range (mm)	Steps (mm)	No.of pieces
Special piece 1.005	-	1
1^{st} series 1.01 to 1.49	0.01	49
2^{nd} series 0.5 to 24.5	0.5	49
3^{rd} series 25 to 100	25.0	4
Total pieces		**103**

Set of 78 pieces (M 78)

Range (mm)	Steps (mm)	No.of pieces
1.0025	-	1
1.005	-	1
1.0075	-	1
1.01 to 1.49	0.01	49
0.5 to 9.5	0.5	19
10.0 to 50.0	10.0	5
75.0 & 100.0	-	2
Total pieces		**78**

Set of 47 pieces (M 47)

Range (mm)	Steps (mm)	No.of pieces
1st series 1.005	-	1
2nd series 1.01 to 1.09	0.01	9
3rd series 1.1 to 1.9	0.1	9
4th series 1.0 to 24.0	1.0	24
5th series 25.0 to 100.0	25.0	4
Total pieces		**47**

Set of 87 pieces (M 87)

Range (mm)	Steps (mm)	No.of pieces
1st series 1.001 to 1.009	0.001	9
2nd series 1.01 to 1.49	0.01	49
3rd series 0.5 to 9.5	0.5	19
4th series 10.0 to 100.0	10.0	10
Total pieces		**87**

Set of 46 pieces (M 46)

Range (mm)	Steps (mm)	No.of pieces
1st series 1.001 to 1.009	0.001	9
2nd series 1.01 to 1.09	0.01	9
3rd series 1.1 to 1.9	0.1	9
4th series 1.0 to 9.0	1.0	9
5th series 10.0 to 100	10.0	10
Total pieces		**46**

Even though there are a number of sets of slip gauges available, the popularly recommended are: M 112, M 87, M 46, M 38 and M 9

Set of 38 pieces (M 38)

Range (mm)	Steps (mm)	No.of pieces
1.005	-	1
1st series 1.01 to 1.09	0.01	9
2nd series 1.1 to 1.9	0.1	9
3rd series 1 to 9.0	1.0	9
4th series 10.0 to 100	10.0	10
Total pieces		**38**

Set of 86 pieces (M 86)

Range (mm)	Steps (mm)	No.of pieces
1st series 1.001 to 1.009	0.001	9
2nd series 1.01 to 1.49	0.01	49
3rd series 0.5 to 9.5	0.5	19
4th series 10.0 to 90.0	10.0	9
Total pieces		**86**

Set of 9 pieces (M 9)

Range (mm)	Steps (mm)	No.of pieces
1.001 to 1.009	0.001	9

Selection and determination of slip gauges for different sizes

Objective: At the end of this lesson you shall be able to
- **state slip gauges for different sizes.**

For determining a particular size, in most cases a number of slip gauges are to be selected and stacked one over the other by wringing of the slip gauges.

While selecting slip gauges for a particular size using the available set of slip gauges, first consider the last digit of the size to be built up. Then consider the last or the last two digits of the subsequent value and continue to select the pieces until the required size is available.

Example

Building up a size of 44.8725 mm with the help of 112 piece set. (Table 1)

Set of 112 pieces

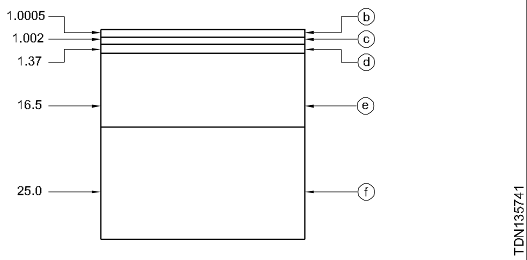
Range (mm)	Steps (mm)	No.of pieces
1.0005	-	1
1.001 to 1.009	0.001	9
1.01 to 1.49	0.01	49
0.5 to 24.5	0.5	49
25.0 to 100.0	25.0	4
Total pieces		**112**

Table 1

Procedure	Slip pack	Calculation
a First write the required dimension.		44.8725
b Select the slip gauge having the 4th decimal place.	1.0005 Substract	1.0005 43.872
c Select 1st series slip that has the same last figure.	1.002 Substract	1.002 42.87
d Select the 2nd series slip that has the same last figure and that will leave .0 or 0.5 as the last figure.	1.37 Substract	1.37 41.5
e Select a 3rd series slip that will leave the nearest 4th series slip (41.5-25=16.5).	16.5 Substract	16.5 25.00

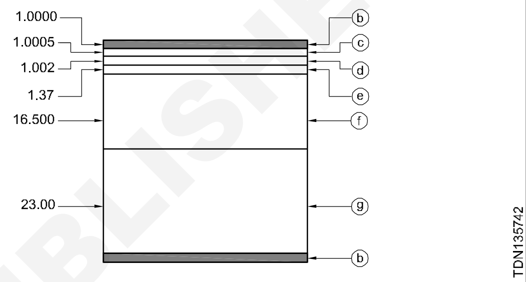
f Select a slip that eliminates the final figure. Add	25.0 Substract 44.8725	25.00 0

Fig 1



Procedure	Slip gauge pack	Calculation
a First write the . required dimension		44.8725
b Two numbers of protectorslips of 1mm each	2.000 Substract	2.0000 42.8725
c Select the slip gauge having the 4th decimal place	1.0005 Substract	1.0005 41.8720
d Select 1st series slip that has the same last figure	1.002 Substract	1.0020 40.8700
e Select the 2nd series slip that has the same last figure and that will leave .0 or 0.5as the last figure	1.3700 Substract	1.3700 39.5000
f Select a 3rd series slip that will leave the nearest 4th series slip	16.5000 Substract	16.5000 23.0000
g Select a slip that eliminates the final figure	23.0000 Substract	23.0000

Fig 2



Care and maintenance of slip gauges & sine bar

Objective: At the end of this lesson you shall be able to
- **state the preventive measures to be taken for protecting slip gauges and sine bar**

Precision measuring instruments play an important role in maintaining the quality of the products. Measuring Instruments are also very expensive. It is important that the instruments are well looked after and maintained by the person who uses it.

Protection against corrosion

High atmospheric humidity and sweat from hands can cause corrosion to instruments. Avoid this.

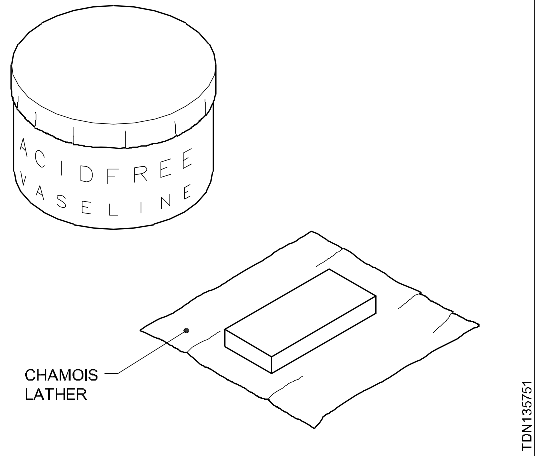
Acid-free vaseline (petroleum jelly) applied lightly on the instruments can give protection against corrosion. (Fig 1)

Be sure that the instruments are thoroughly cleaned and free from water or moisture before applying vaseine.

Use chamois leather for giving a light coating of vaseline.

Always clean the slip gauges with carbon tetrachloride and apply petroleum jelly after use.

Fig 1



Milling machines

Objectives: At the end of this lesson you shall be able to
- **state the introduction of milling machines**
- **classify the types of milling machines**
- **state the specification of milling machine.**

Introduction

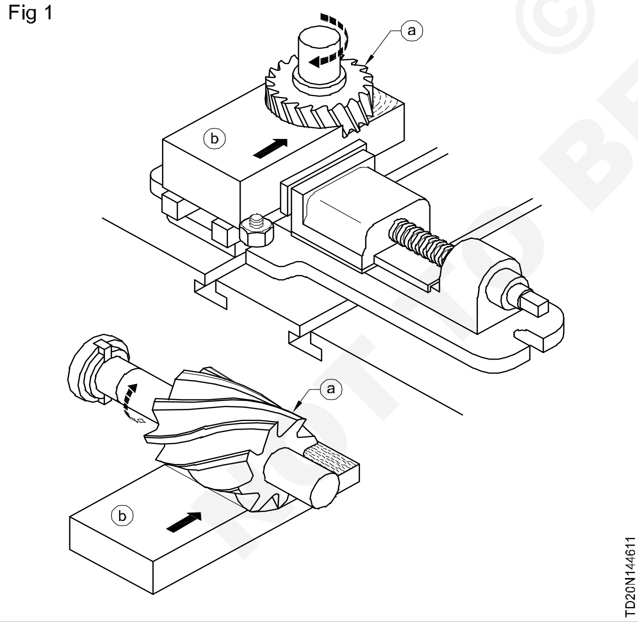
A milling machine is a machine tool that removes metal as the work is fed against a rotating multipoint cutter. The cutter rotates at a high speed and because of the multiple cutting edges it removes metal at a very fast. The machine can also holds one or more number of cutters at a time. This is why a milling machine finds wide application in production work. This is superior to other machines as regards accuracy and better surface finish, and is designed for machining a variety of tool room work.

Principle of milling (cutting)

In milling, the cutter has a rotary movement, the speed of which depends upon the cutting speed required. Driving the milling arbor at various rotational speeds makes it possible to achieve approximately the same cutting speeds [peripheral speed] with cutters of different diameters.

While the milling cutter (a) rotates at a high speed, and because of the multiple points, it removes metal at a very fast rate, in comparision with other machine tools. (Fig1)

Fig 1

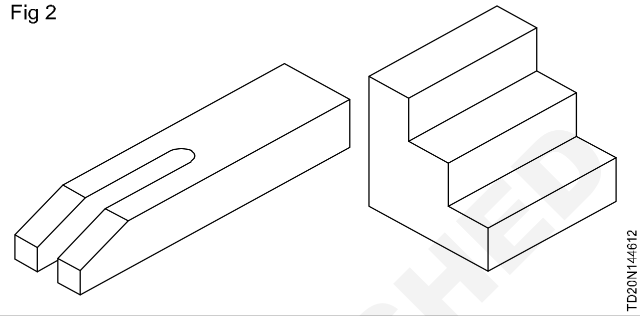


Job (b) can be machined manually or automatically.

By milling we can produce flat (horizontal, vertical, angular) and formed surfaces. (Fig 2)

A milling machine finds wide application in production work as the machine can hold one or more number of cutters at a time, and is good in accuracy, surface finish etc.

Fig 2



Classification

The classification according to the general design of the milling machine is:

- column and knee type
- fixed bed type
- planer type
- special type

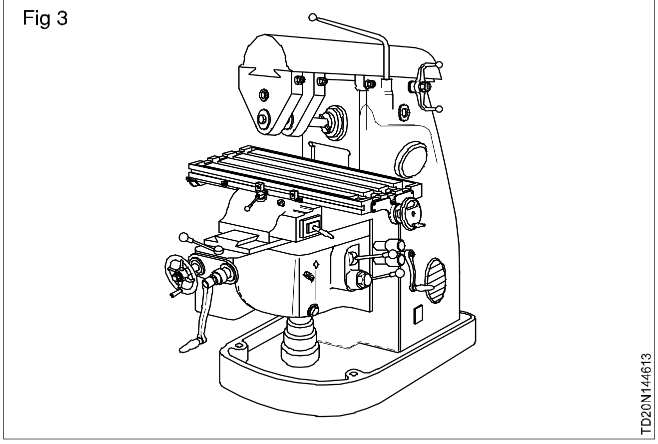
But out of these types the one that is used most in general workshop is the column and knee type machine.

In the column and knee type category the following machines are covered.

- Plain/horizontal milling machine (Fig 3)
- Vertical milling machine (Fig 4)
- Universal milling machine (Fig 5)

1 Plain milling machine (Fig 3)

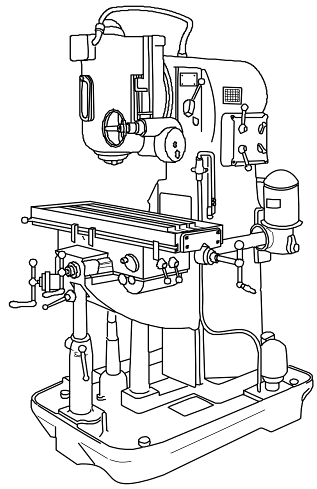
Fig 3



- More rigid and sturdy than other machines and accommodate heavy work.

- Table may be fed either by hand or by power against a rotating cutter and in the three directions namely longitudinately.
- Milling cutters mounted on the horizontal arbor.
- Spindle rotates horizontally and parallel to the mchine table.

Fig 4



TD20N144614

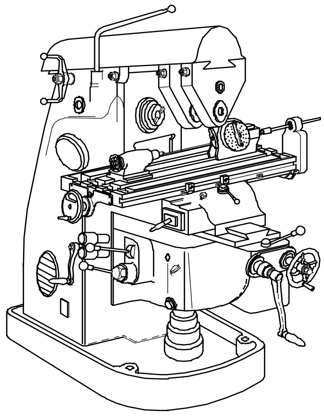
2 Vertical milling machine (Fig 4)

- Distinguished from the horizontal milling machine by position of the spindle vertical or perpendicular to the work table.
- The spindle rotates about the vertical axis.
- Spindle moved up and down by spindle feed and also may be filted or swivelled.
- Most suitable for boring, pocket milling, profile milling and for making keyways.

3 Universal milling machine (Fig 5)

- Table apart from having movements in 3 directions and also can be swivelled about its horizontal axis.
- Maximum swivelling can be made is 45° both in clockwise and anticlockwise direction.

Fig 5



TD20N144615

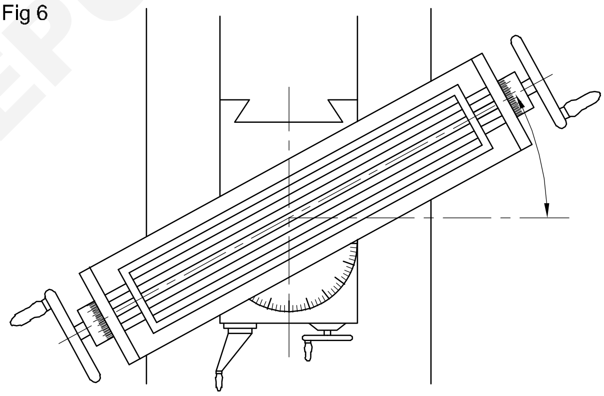
- As angular feeding is possible suitable for milling helical gears, helical grooves etc,.

The universal millling machine is similar in construction to the plain machine. But its table, apart from having movements in 3 directions, can also be swivelled about the horizontal axis. The maximum swivelling is 45° both the clockwise and anticlockwise directions. (Fig 6)

The swivelling of the table permits angular feeding. Because of this, the universal milling machine is suitable for milling helical gears, helical grooves, etc,.

This machine is supported with different attachments like vertical head, slotting head, rack-milling attachment.

Fig 6



TD20N144616

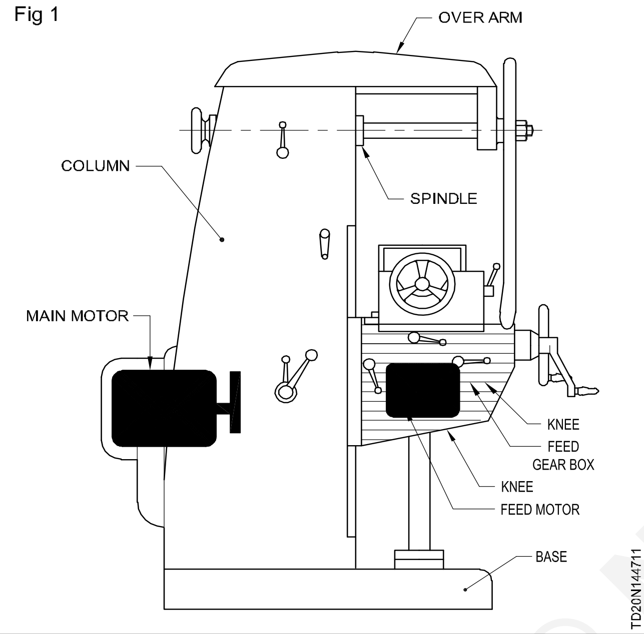
Construction and specification

Objective: At the end of this lesson you shall be able to
- **state the main parts of a milling machine and its functions.**

Part of a milling machine

The principle parts of a milling machine (Fig 1) are as follows.

Fig 1



Base

The base of the machine serves as a foundation member for all the other parts which rest upon it. It is made of cast iron. It carries a column.

Column

The column is the main supporting frame mounted vertically on the base. The column houses all the driving mechanism for the spindle and table feed. The main motor is usually incorporated in the column. The top of the column is finished to hold an overarm that extends outward at the front of the machine. The lower part of the column is a study box base, which incorporates the cutting fluid tank.

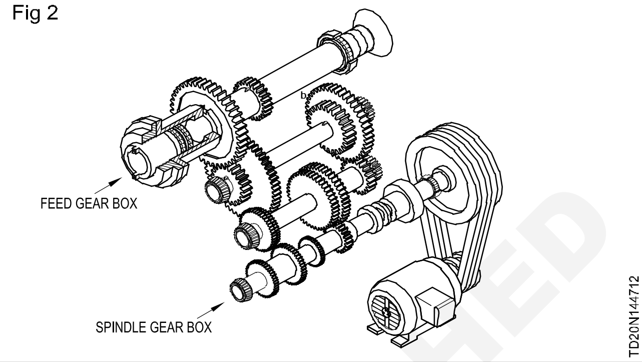
Knee

The knee is of rigid casting that slides up and down along a precision-machined guideway. The knee houses the speed mechanism of the table and the different controls to operate it. The feed motor and gearbox are usually incorporated in the knee.

Gearbox

The gearbox for the spindle drive comprises shafts with bearings and gears, Fig 2 and controls for the setting of the spindle speed.

Fig 2



Spindle

The spindle is housed in the upper part of the column and receives power from the motor and transmits it to the arbor.

The front end of the spindle projects from the column face and is provided with a tapered hole into which various cutting tools and arbors may be mounted. The accuracy in machine depends primarily on the rigidity of the spindle. The speed of the spindle can be selected by the speed gearbox, and the feeds can be selected through the feed gearbox. The spindle is arranged horizontal milling machine and vertically in the vertical milling machine. (Fig 3 & 4)

Fig 3

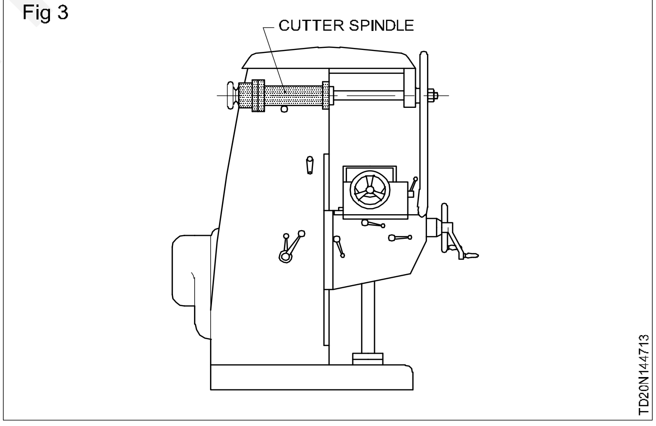
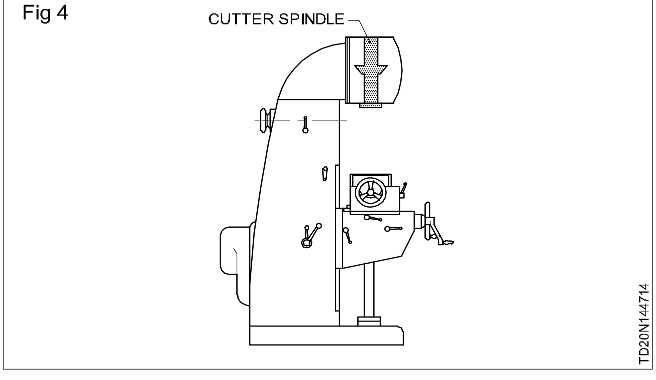


Fig 4



Saddle (Fig 5)

The saddle is placed on the top of the knee which slides come guideway, set exactly at 90° to the column face.A cross-feed screw near the top of the knee engages a nut on the bottom of the saddle to move it horizontally for applying the cross-feed.

Fig 5

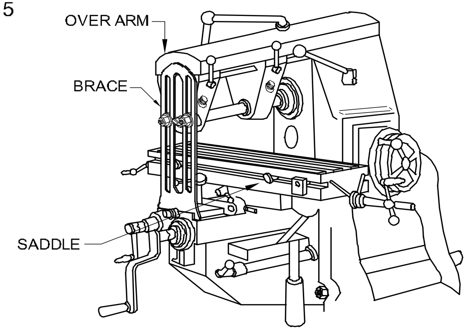
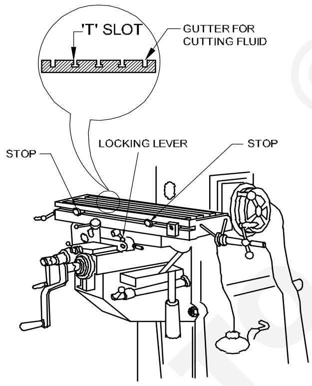


Table (Fig 6)

The table rests on the guideway on the saddle and travels longitudinally. 'T' slots are provided on the table to mount the workpieces directly or to mount the work-holding devices. The longitudinal feed-stops are located on the front of the table. This disengage the machine feed at a set position. The table is also fitted with a hand wheel for hand feed in the longitudinal direction, and a lever for locking the table. There is a gutter around the edges of the table to collect the cutting fluid.

Fig 6



Overarm and brace (Fig 5)

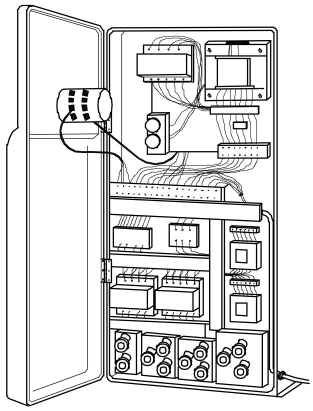
The overarm is mounted on the top of the column above the spindle, and is intended as a support when milling with an arbor. The arm is adjustable so that the bearing support may be provided nearest to the cutter.

The brace provides extra stability to the machine. It must be loosened before the table elevation setting is altered.

Electrical equipment (Fig 7)

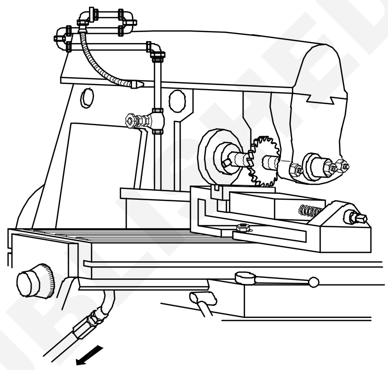
The electrical equipment for the different controls of the machine is usually grouped in the column. The equipment consists mainly of fuses, motor breakers and contactors. Motor breakers switch off the power in the event of an overload.

Fig 7



Cutting fluid equipment (Fig 8)

Fig 8



The cutting fluid equipment consists of a pump, piping and hoses, a nozzle and shut-off valve, and a tank in the machine base.

The specification/size of a milling machine is specified as follows

1 The size of the table (length and width)

2 The maximum lengths of longitudinal, cross and vertical travel of the table.

3 Number of spindle speeds, number of feeds.

4 Spindle nose taper.

5 Power required.

6 Net weight of the machine.

7 The floor space required.

8 Type of the machine.

Example of specification (Milling)

Spindle

1 Spindle position - Horizontal

2 Spindle taper - ISO 50

3 Spindle motor power - (7.5 KW or more)

4 Spindle speed range - 35 to 1800 r.p.m.

Table

1 Table size - 2055 mm x 355 mm or more

2 Clamping area 1600 mm x 355 mm

3 Max. permissible load on table 500 kg or more

4 The table top should hacve longitudinal T-slots. The dimension of T-slots is to be specified by supplier. Vendor to specify.

5 Longitudinal traverse 1200 mm or more.

6 Vertical traverse 425mm or more.

7 Cross traverse 320 mm or more 2.2.8 swivelling of Table 45 degree.

Feed Rates

1 Rapid traverse rate longitudinal 4000 mm/min or more.

2 Rapid traverse rate cross 800 mm/min or more.

3 Feed rates longitudinal 8-1600 mm/min (approx).

4 Feed rates cross 1.6 - 320 mm/min.

In addition to the above net weight, floor space required, work holding devices, cutter holding devices attachment and accessories required may furnished.

Driving and feed mechanism of milling machine

Objectives: At the end of this lesson you shall be able to
- **state type of mechanism**
- **explain the automatic feed**
- **describe the manual feed.**

Milling machine mechanism is divided into two types:

Spindle drive mechanism

Table feed mechanism

The spindle drive mechanism is incorporated in the column. All modern machines are driven by individual motors housed with in the column. The spindle receives power from a combination of gears and clutch assembly. By altering gears spindle receives multiple speed.

Table feed mechanism contained with in the knee of the machine table. It has three different feed movements.

Longitudinal feed movement

Cross feed movement

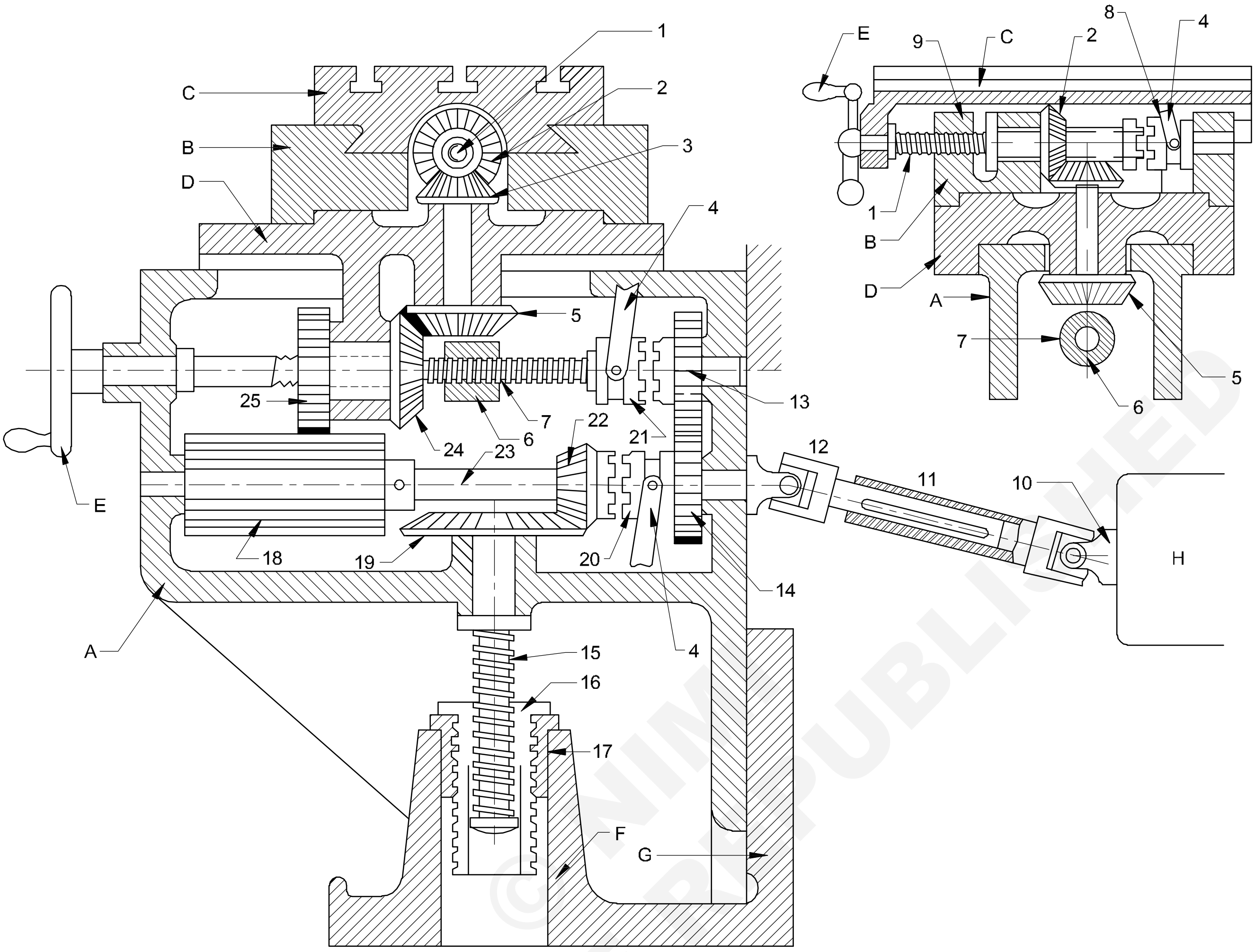
Vertical feed movement

These three movements can get manual feed or automatic feed. By swiveling the three handle wheels we can give manual feed. By engaging the clutch operating levers we can get the automatic feed as follows:

Fig 1 illustrates the power feed mechanism contained within the knee A of the machine to enable the table C to have three different feed movements, i.e longitudinal, cross and vertical. The power is transmitted from the feed gearbox H consisting of change gears to shaft 23 in the knee A of the machine by a telescopic - shaft 11. Both ends of the shaft 11 are provided with universal joint 10 and 12. Telescopic shaft and universal joints are necessary to allow vertical movement of the knee A, gear 14 attached to the jaw clutch 20, is keyed to the shaft 23 and receivers gear 13 which is free to rotate on the extreme end of the cross feed screw 7. Bevel gear 22 is free to rotate on shaft 23 and is in mesh with gear 19 fastened to the evaluating screw 15,16 serves as a nut for 15, and as a screw in nut 17,15 and 16 therefore, serve as a telescopic screw combination and a vertical movement of the knee is thus possible. As soon as the clutch 20 is engaged with the clutch attached to the bevel gear 22 by means of a lever 4,22 rotates and this being in mesh with gear 19 causes the elevating screw 15 to rotate in 16 giving a vertical movement of the knee. like-wise, when the clutch21, which is a keyed to the cross feed screw 7, is engaged with the clutch attached to gear 13, power comes to the screw 7 through gears 4 and 13. This causes the screw 7 to rotate in nut 6 of the clamp bed giving a cross feed movement of the clamp bed D and saddle B.

Gear 18 is fastened to shaft 23, and meshes with gear 25 which is fastened to the bevel gear 24. Again 24 meshes with gear 5 attached to a vertical shaft which carries at its upper end another bevel gear 3. Gear 3 meshes with gear 2 which is fastened to the table feed screw 1. Therefore, longitudinal feed movement of the table is possible through gears 18,25,24,5,3, and 2.

Fig 1



MILLING MACHINE POWER FEED MECHANISM

A.	KNEE
B.	SADDLE
C.	TABLE
D.	CLAMP BED
E.	FEED HAND WHEEL
F.	BED
G.	COLUMN
H.	FEED GEAR BOX

1.	LONGITUDIONAL FEED SCREW
2,3,5,19,22,24.	BEVEL GEARS
4.	CLUTCH OPERATING LEVER
6.	NUT
7.	CROSS FEED SCREW
8,20,21.	POWERFEED CLUTCH
9.	SADDLE NUT
10,12.	UNIVERSAL JOINT
11.	TELESCOPIC FEED SHAFT
13,14,18,25.	GEARS
15,16,17.	ELEVATING SCREW
23.	FEED SHAFT

TD20N144811

Different types of milling cutter and their uses - cutter nomenclature

Objectives: At the end of this lesson you shall be able to
- **state the two main categories of milling cutters**
- **state the different types of plain milling cutters**
- **state the uses of plain milling cutters.**

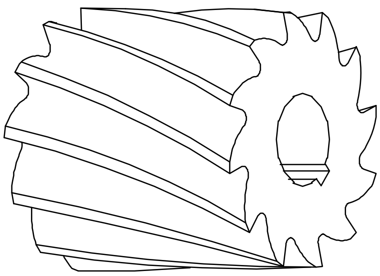
Milling cutters

Milling cutters generally fall into two categories, solid cutters and inserted tool cutters.

Solid cutters (Fig 1)

These cutters are those in which the teeth have been cut into the body of the cutter. The teeth may be straight (parallel) or helical (at an angle) to the axis of the cutter. Solid type cutters are generally made of high speed steel.

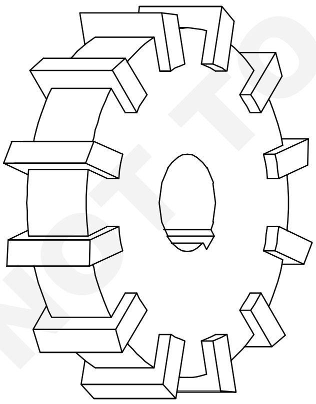
Fig 1



Inserted tool cutter (Fig 2)

These cutters have removable and replaceable teeth which are fastened or locked into the body of the cutter. The inserted tool construction is generally used on large cutters as the blades can be quickly replaced when they become dull.

Fig 2



Plain milling cutters

Plain milling cutters are cylindrical, having teeth on the periphery only. They are used to produce flat surfaces, by feeding the table longitudinally. The cutter teeth may be straight or helical according to the size of the cutter. Wider plain cutters are used for slab milling which are known as slab milling cutters.

Types of plain milling cutters

Light duty plain milling cutters

These are less than 19mm wide usually have straight teeth. (Fig 3) Those over 19mm wide have a helix angle of about 25°. (Fig 4)

Fig 3

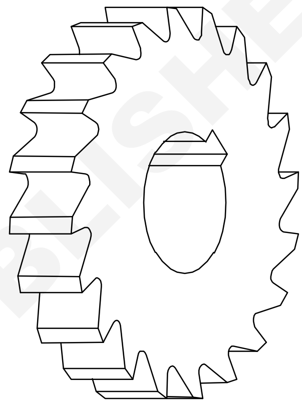
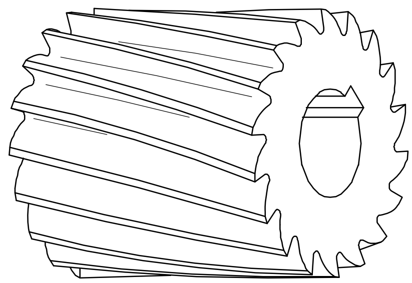


Fig 4

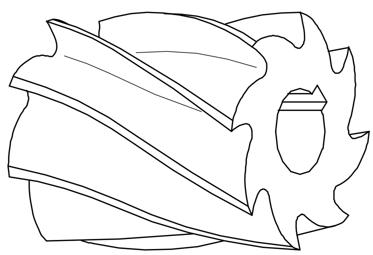


This type of cutter is used only for light milling operations since it has too many teeth to permit the chip clearance required for heavier cuts.

Heavy duty plain milling cutters (Fig 5)

These cutters have fewer teeth than the light duty ones, which provides for better chip clearance. The helix angle upto 45°.

Fig 5

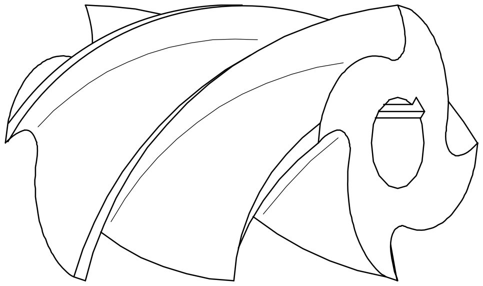


The greater helix angle on the teeth produces a smoother surface due to shearing action and reduces chatter. Less power is required for the cutter than what the straight tooth and small helix angle cutters require.

Helical plain milling cutters (Fig 6)

These cutters are high helix cutters with the helix angles from 45° to over 60°. They are particularly suited to this milling of wide and intermittent surfaces in contour and profile milling. These cutters are used for milling soft steels, brass, etc.

Fig 6



Plain milling cutters are also made in shank type. These are sometimes nicked on their periphery on a helical pattern for chip breaking and smooth operation.

Specification

The size of the plain milling cutter is specified by the outside diameter, length and the bore size.

Example

∅ 50 x 100 x 27 bore, 45°

Direction of helix of the cutter

The teeth (cutting edge) of a cutter may be either straight or follow a helix.

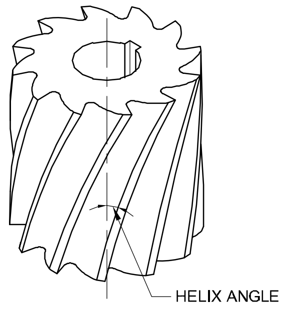
If the cutter axis is hold vertically and the helix is towards the right side it is called a right hand helix cutter. (Fig 7) and if the helix is towards the left side, it is called a left hand helix cutter (Fig 8)

The helix angle generates a force directed along the cutter axis during cutting and a reaction to this force in the workpiece.

When a cutter has a helix and a cut of the same hand, this force will pull the cutter away from the spindle.(Fig9) when the helix and cut are of opposite hands, the force will press the cutter into the spindle. (Fig 10) As a consequence, cutters having a helix and a cut of the same hand can only be safely used when they are positively attached to the spindle. The frictional hold of a taper is inadequate in this situation.

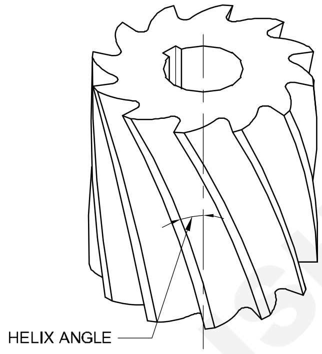
When mounting a cutter on the arbor of a milling machine, it is particularly important that the hands of the cut and the helix are checked.

Fig 7



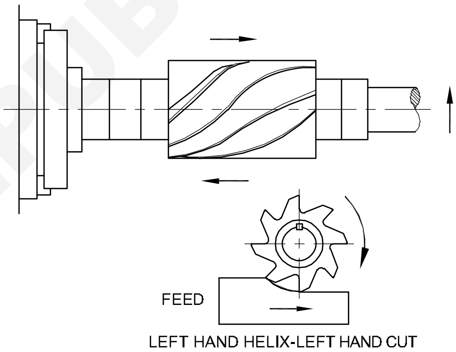
RIGHT HAND HELIX

Fig 8



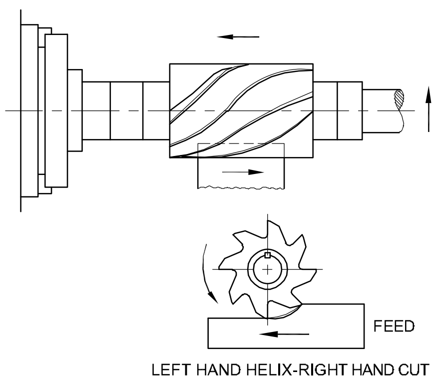
LEFT HAND HELIX

Fig 9



CUTTER PULLS AWAY FROM SPINDLE

Fig 10



CUTTER PUSHED TOWARDS SPINDLE

Side and face cutters

Objective: At the end of this lesson you shall be able to
- **state the different types of side and face milling cutters and their uses.**

These cutters differ from plain milling cutters due to the fact that they have teeth on the periphery and face.

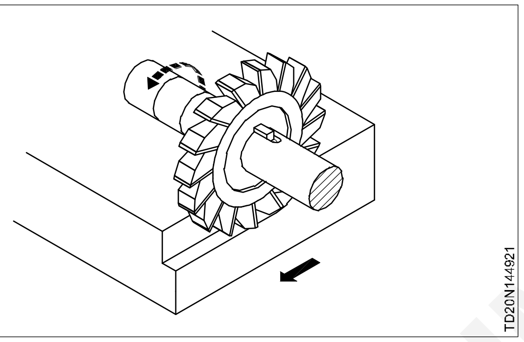
These cutters are mainly used for step milling, slot milling and straddle milling. These cutters are available from 50 to 100 mm in diameter and the width of the cutters ranges from 5 to 32 mm.

Types of side and face milling cutters

Half side milling cutter (Fig 1)

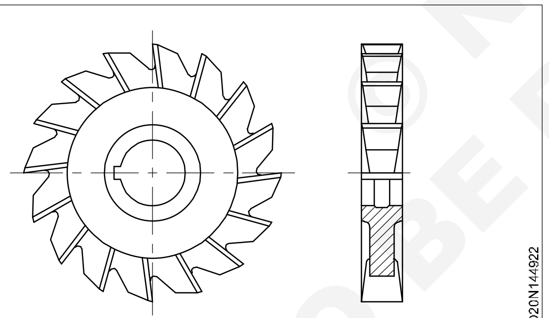
Cutters with teeth on one side only are called half side milling cutters and are used for heavy straddle milling, and for machining one side only.

Fig 1



Plain side and face milling cutter (Fig 2)

Fig 2



Inside and face milling cutters with teeth on both the sides are known as plain side and face milling cutters and are used for slot cutting (Fig 3) and face milling. These cutters are also used for straddle milling. (Fig 4).

Fig 3

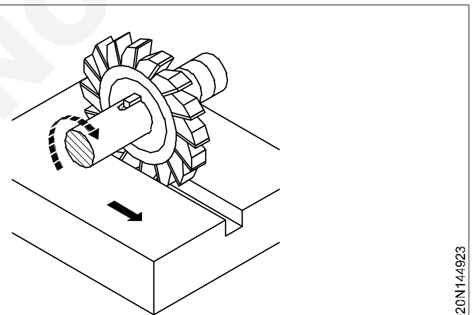
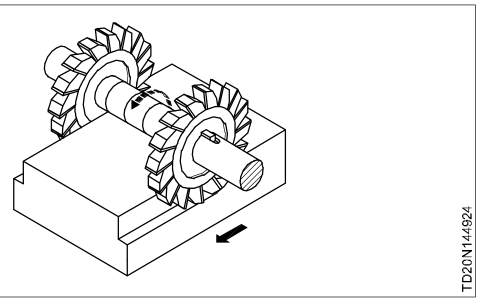


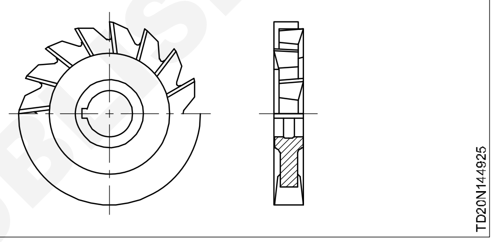
Fig 4



Staggered teeth side milling cutter (Fig 5)

These cutters have alternate teeth with opposite helix angles. Due to this design, the chip space increases to a great extent. These cutters are used for milling deep and narrow slots or keyways.

Fig 5

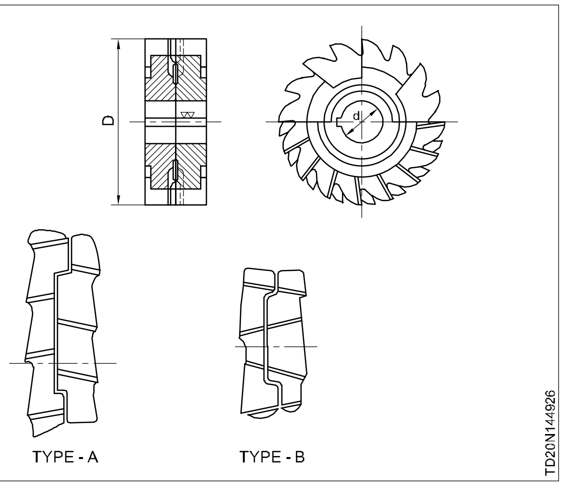


Interlocking side milling cutter (Fig 6)

This cutter is formed out of two half side milling cutters or two staggered teeth side milling cutters. They are made to interlock to form one unit. The teeth of the two cutters may be plain or of alternate helix. The cutters are used for milling wider slots of accurate width. The width of the cutter can be varied by inserting spacers between the two halves of the cutter.

The width of the cutter ranges from 10 to 32 mm with the diameters ranging from 50 to 200 mm. The width of the cutter may be adjusted to the max/min of 4 mm. The interlocking cutters can be adjusted to compensate for thr wear, and get sharpened as well.

Fig 6



End mill cutters

Objectives: At the end of this lesson you shall be able to
- **list the different types of end mill cutters and their uses**
- **state the application of slot mill cutters**
- **state the different types of methods of holding end mill cutters**
- **explain the influence of down-milling and up-milling in end mill cutters.**

End mill cutters

End mill cutters have the cutting teeth on the end as well as on the periphery, and are filted to the spindle by a suitable adapter.

The end mill is used for milling small faces, slots, (Fig 1) for milling profiles (Fig 2) and milling recesses. (Fig 3) Some end mills have indexable inserts which can be replaced when worn out. (Fig 4)

Fig 1

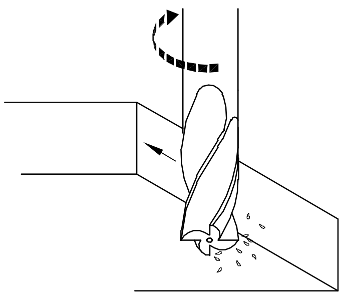


Fig 2

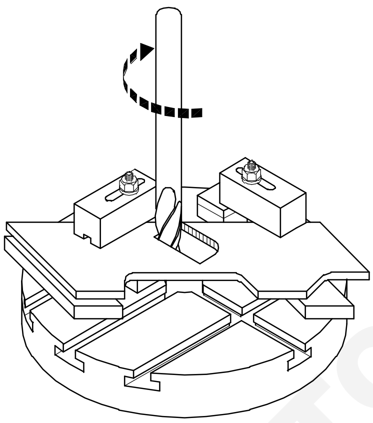


Fig 3

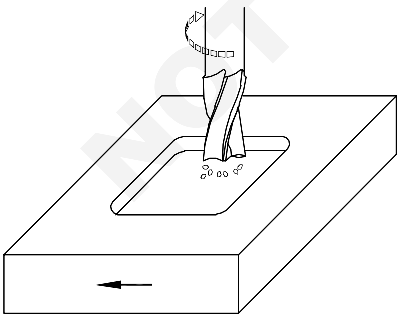
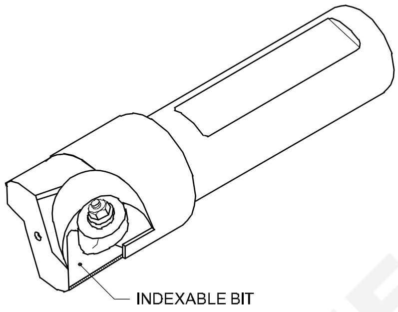
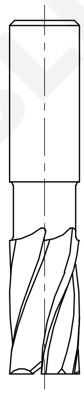


Fig 4



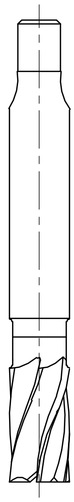
End mill cutters are solid type of cutters in which the shank and the cutters are integral. (Fig 5)

Fig 5



End mill cutters have straight shank (Fig 5) or taper shank (Fig 6).

Fig 6



Slot drills

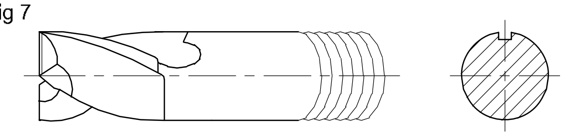


Fig 7

The two-flute type (Fig 7) is called a slot drill. The slot drills have flutes which meet at the cutting end, forming two cutting tips across the bottom. These tips are of different lengths, one extending beyond the central axis of the cutter. This permits the slot drill to be used in amilling machine for drilling a hole to start a slot that does not extend to the edge of the metal. It is used for plunge milling like keyways etc.

Angular and slitting saw milling cutter

Objectives : At the end of this lesson you shall be able to
- **state the different types of angular milling cutters and their uses**
- **state the specification of angular milling cutters**
- **explain slitting saw cutters and their uses**
- **state the specification of a slitting saw cutter.**

Angular milling cutters

These cutters have teeth on the periphery, and the cutting edges are placed on a conical surface.

Angular cutters have teeth that are neither parallel nor perpendicular to the cutting axis.

Angular milling cutters are made with a hole for use in horizontal milling, or with a shank for use in both horizontal and vertical milling. (Fig 1)

They may be divided into two groups.

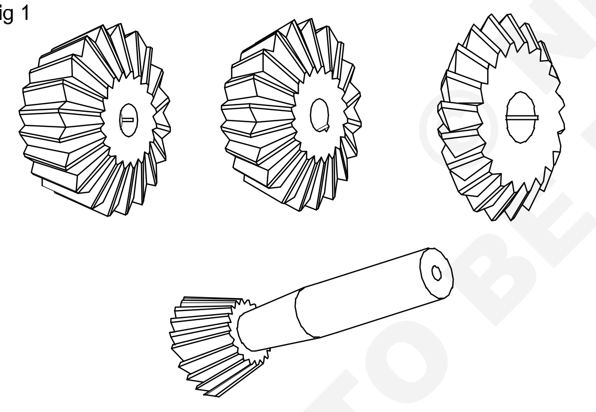


Fig 1

Single angle milling cutters

These cutters have teeth on the angular surface, and may or may not have teeth on the flat side. The included angle between the flat face and the angular face designates the cutters, such as 45° or 60° angular cutter. (Fig 2) They may be of the shell or shank type.

Specification

A shell end single angle cutter of diameter D = 80, angle μ= 50° of 'tool type' H and for right hand cutting shall be specified and designated as

Shell end single angle milling cutter 80 x 50°H IS:6256.

A dovetail milling cutter type A having diameter D = 20 mm, angle μ = 60° of tool type 'N' for right hand cutting shall be specified as

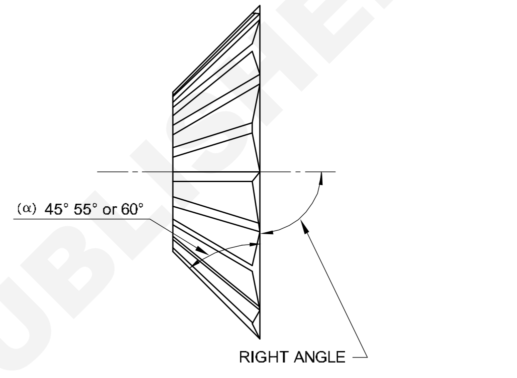


Fig 2

Dovetail milling cutter A20 x 60°N BIS 6255.

In type 'A', the small end is having less diameter and in type 'B' the small end is having more diameter than in type A.

Single angle cutters are used to dovetail guideways etc. (Fig 3)

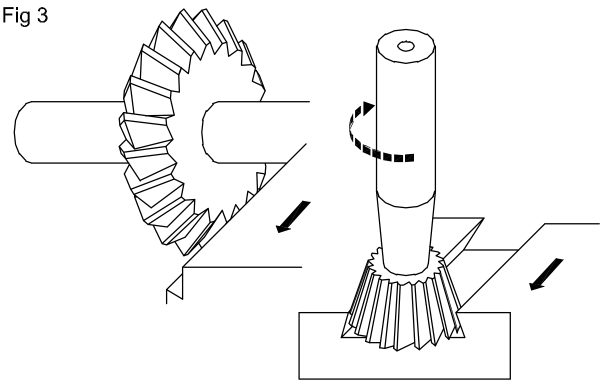


Fig 3

Equal angle milling cutters

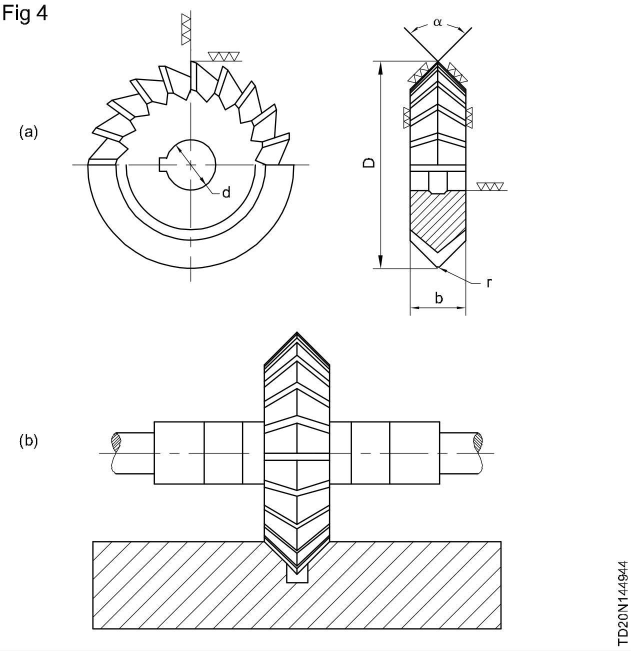
These cutters have two intersecting angular surfaces with cutting teeth on both. When these cutters have equal angles on both sides of the line at right angle to the axis (symmetrical), they are designated as per the size of the included angle such as 45°, 60° or 90°. Double angle cutters have two cutting edges. (Fig 4a)

Specification

An equal angle milling cutter of diameter D = 56 mm for angle 60° of 'tool type' N shall be specified as Equal angle milling cutter 56 x 60° N IS 6326.

It is used to machine Vee slots. (Fig 4b)

Fig 4

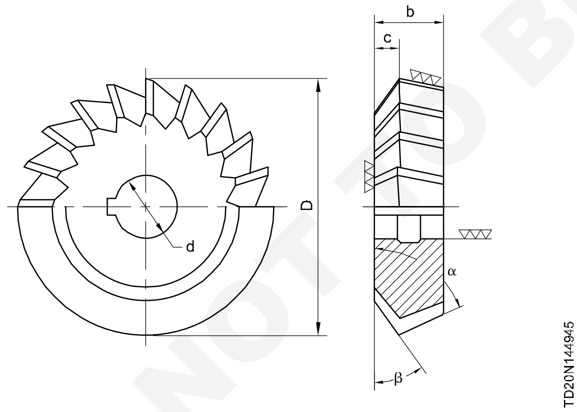


Double unequal angle cutter

When the angles formed are not the same (unsymmetrical), the cutters are designated by specifying the angle on both sides of the plane or line. (Fig 5)

These cutters are generally used for milling the flutes on taps or reamers. The cutters are marked with the type of taps or reamers for which they should be used.

Fig 5



Specification

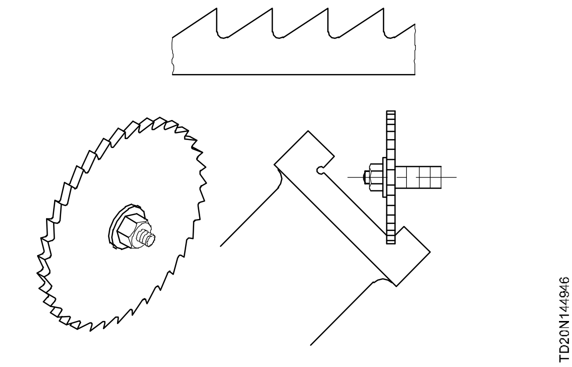
A double angle milling cutter of diameter D = 50 mm, b = 12 mm and angle 75° of 'tool type H and for right hand cutting shall be specification as

Double angle milling cutter 50 x 12 x 75°H IS 6325.

Slitting saw

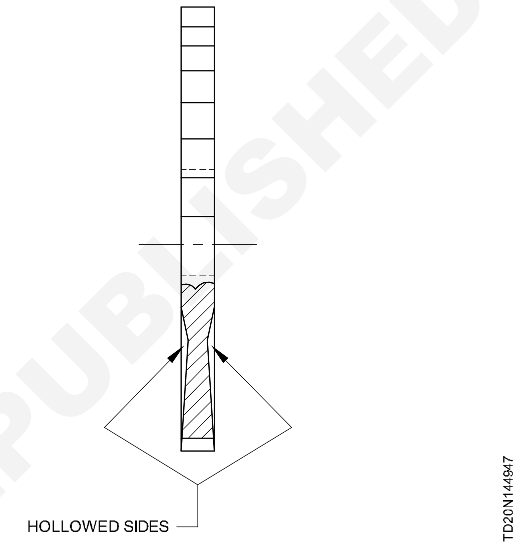
It is basically a thin plain milling cutter. It has a large number of teeth. (Fig 6)

Fig 6



In order to prevent the sides of the saw from rubbing or binding when in use, the sides are relieved or dished. (Fig 7)

Fig 7



Slitting saws are made in widths of 3 to 6 mm. Because of the thin cross-section, they should be operated at approximately one quarter to one eighth of the feed per tooth used for the other cutters. For non-ferrous metals, these speeds can be increased. Unless a special driving flange is used for slitting saws, it is not advisable to key the saw to the milling arbor.

The arbor nut should be pulled up as tightly as possible by hand only. Since slitting saws are so easily broken, some operators find it desirable to adopt climb or down-mill method when sawing. However to overcome the play between the lead screw and nut, the backlash eliminator should be engaged.

A slitting saw is specified by its outside diameter, bore diameter and thickness.

Example : 150 x 6 x 27 mm bore

Form milling cutters

Objectives : At the end of this lesson you shall be able to
- **state the name of different types form milling cutters and their uses**
- **state the specification of form milling cutters.**

Form milling cutters have irregular profile on the cutting edges in order to generate an irregular outline of the work. They are normally solid but, sometimes, may have inserted teeth. Different types of standard form cutters are described below.

Convex milling cutter

These cutters have their teeth curved outwards on the circumferential surface to form the contour of a semi circle. Concave semi-circular surfaces are produced with this cutter. (Fig 1) The diameter of the cutter ranges from 50 to 125 mm and the radius of the semi circle ranges from 1.6 to 20.0 mm. (Fig 2)

Fig 1

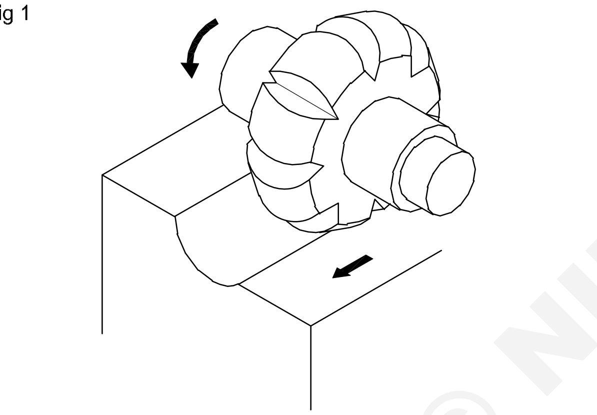
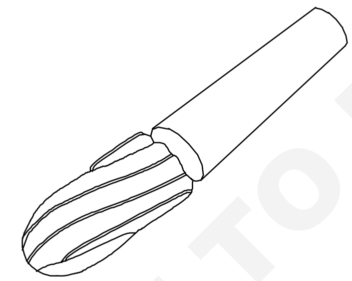


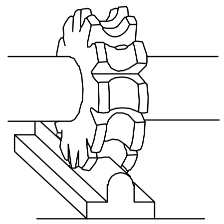
Fig 2



Concave milling cutter

These cutters have their teeth curved inwards on the circumferential surface to form the contour of a semi circle. Convex semicircular surfaces are produced with this cutter. (Fig 3).

Fig 3



The diameter of the cutter ranges from 56 to 110 mm and the radius of the semi circle ranges from 1.5 to 20.0 mm.

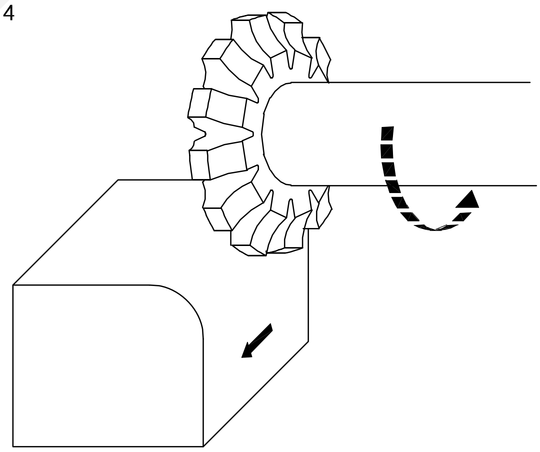
Corner rounding milling cutter

The corner rounding milling cutters have their teeth curved inwards on the circumferential surface to form the contour of a quarter circle. A convex surface is produced with this cutter. This cutter is used for cutting a radius on the corner or edges. It may be of either the shank or arbor type.

Corner rounding cutters are available with their teeth having placed on one side or both the sides. (Fig 4)

The cutters are specified by the type, diameter width, radius of the form and bore size.

Fig 4



'T' slots cutters

Objectives : At the end of this lesson you shall be able to
- **state the purpose of 'T' slot milling cutters**
- **state the specification of the 'T' slot milling cutters.**

'T' slot cutters

These cutters are profile-sharpened side milling cutters with a straight or taper shank. (Fig 1) They have staggered teeth and are either solid or tipped in construction. Due to the staggered teeth the chips are cleared without clogging.

The 'T' slot cutter is used to cut 'T' shaped slots in machine tool work tables. (Fig 2) Before cutting the 'T' slot, a narrow vertical groove is machined with an end mill or a slot milling cutter.

Fig 1

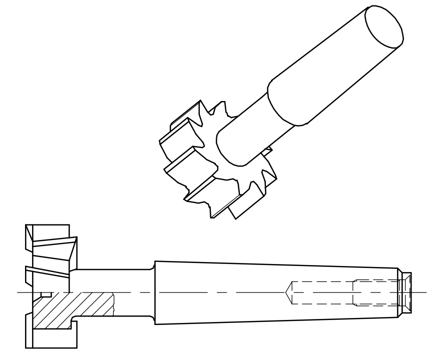
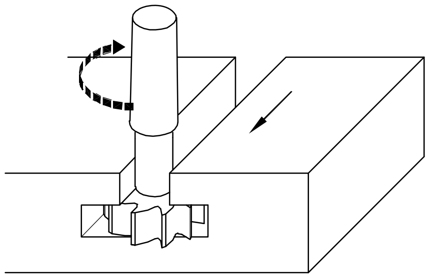


Fig 2

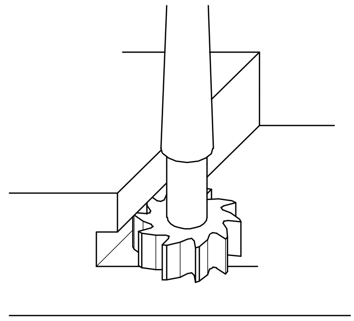


This cutter can also be used to mill undercuts in wider milled channels. (Fig 3)

Specification

A 'T' slot milling cutter with plain paralled shank for milling a 'T' slot of nominal size 12, tool type N, for right hand cutting is designated as plain parallel shank 'T' slot cutter 12 BIS:2668. When the cutter is required with a tool type

Fig 3



other than N, an appropriate tool type H or S shall be added to the designation after the size.

Tool type

N - for mild steel, soft cast iron and medium hard non-ferrous metals.

H - for specially hard and tough metals.

S - for soft and ductile material.

A 'T' slot milling cutter with Morse taper shank with tapped hole for milling a 'T' slot of nominal size 18, tool type N, for right hand cutting is designated as taper shank 'T' slot cutter 18 BIS:2668. When the cutter is required with a tool type other than N, an appropriate tool type H or S shall be added in the designation immediately after the size.

Example : 16 N BIS 2668

Special milling cutters

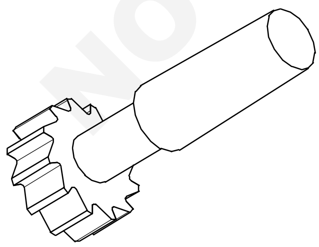
Objectives : At the end of this lesson you shall be able to
- **state the woodruff keyway cutter, thread cutter, gear cutter, tap and reamer cutter and state their uses**
- **state the sprocket cutter, spline cutter, fly cutter and bolted cutter and state their uses.**

Many types and sizes of cutters are available. The selection of an appropriate cutter for a particular type of operation is very important.

Woodruff key cutters

These are similar in appearance to 'T' slot cutters. These cutters have cutting edges on the periphery only. There are no side teeth. (Fig 1)

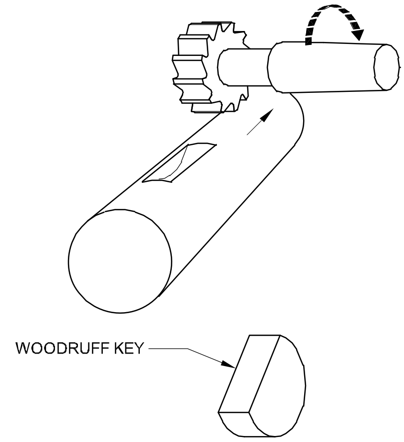
Fig 1



They are used for milling semi-cylindrical keyways in shafts for woodruff key seats. (Fig 2)

This cutter is provided with a shank which may be parallel or with Morse taper. These cutters are either solid or tapered in construction. They can either be of the arbor or shank type.

Fig 2



This cutter may have straight or staggered teeth.

Specification

A woodruff key slot milling cutter, type A of diameter d = 16.5 mm and width b = 5 mm and tool type 'N' for right hand cutting shall be specified as woodruff slot milling cutter A 16.5 x 5 N BIS2669.

When the cutter is required for left hand cutting, the letter 'L' shall be added before the size in the designation.

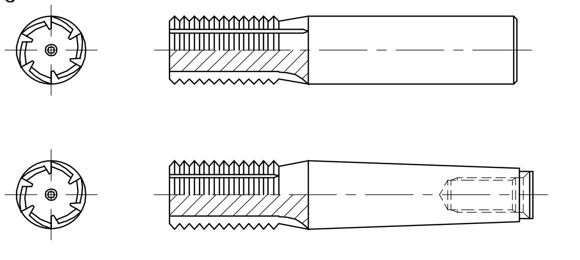
Type 'A' is for straight teeth and type B for staggered teeth.

Thread milling cutter

For milling the threads of specific form and size, thread milling cutters are used. Generally acme and worm-threads are produced on the workpiece by thread milling cutters.

Both parallel shank and taper shank thread milling cutters are available. (Fig 3)

Fig 3



Gear cutter (Fig 4)

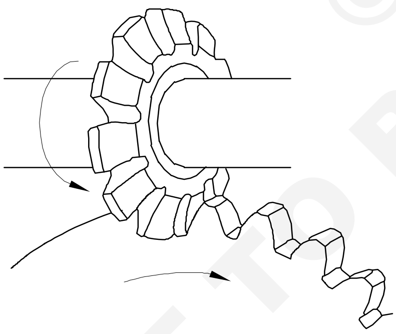
These cutters are used to machine gear teeth by milling.

These cutters having formed cutting edges reproduce the shape of the cutter teeth on the gear blank. According to the gear teeth profile the shape of the cutter teeth may be involute or cycloid. These cutters are available in a very wide range of sizes covering the various sizes of gear tooth.

Example

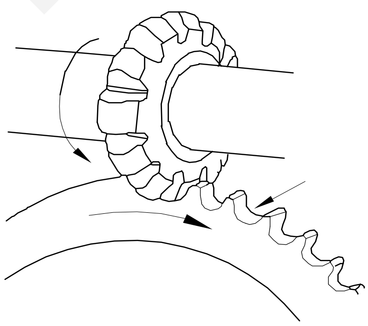
Involute cutter, 3mm module, 27 mm bore, cutter No.5

Fig 4



Sprocket cutters (Fig 5): These cutters are designed to cut the teeth of sprocket wheels which are used in chain drives, such as those found on bicycles and on machinery in general.

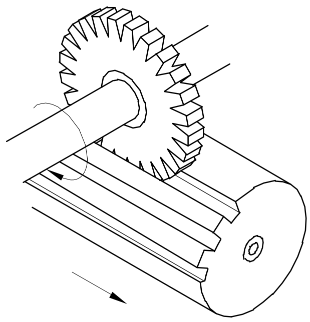
Fig 5



Spline cutters (Fig 6)

These cutters are used to cut splines. They are marked with the type and size of the spline that they should be used for.

Fig 6

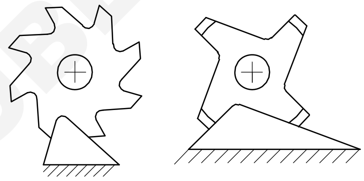


Tap and reamer cutter (Fig 7)

These cutters are used for producing grooves or flutes in taps and reamers. These are the special type of double angle cutters.

The point end of the tool is rounded and the tooth profile corresponds to the type of groove that it is to produce.

Fig 7



Fly cutters (Fig 8)

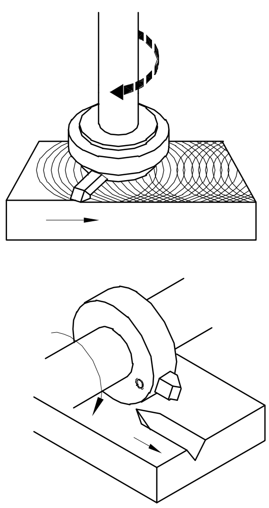
Fly cutters are single point tools having only one cutting edge. These tools are held in various types of holders.

These cutters are used to machine shapes which cannot be produced using standard milling cutters.

These are also used to mill flat surfaces which are truly flat to a very high standard of accuracy.

This cutter is used in tool room and in emergency when standard cutters are not available.

Fig 8

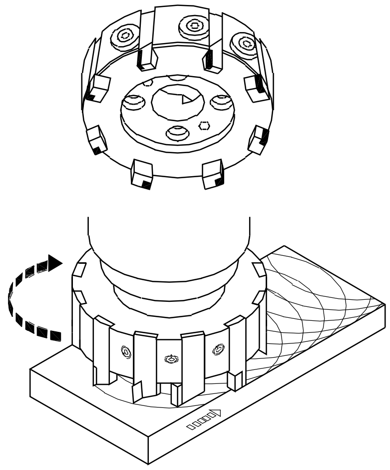


Bolted cutters (Fig 9)**:** The face milling cutters having no shank but one of a larger diameter and they are bolted directly on to the nose of the spindle.

This cutter is used for face milling on large areas of the workpiece. For utmost rigidity, this system is used.

The cutter has a body with slots and fixing devices for the cutting teeth which are fixed into the body. The cutting teeth, which may be made of high speed steel or carbide, can easily be replaced when worn out.

Fig 9



Milling cutter materials

Objectives: At the end of this lesson you shall be able to
- **list the various type of milling cutter materials**
- **state material composition and their properties**
- **brief the effect the coated materials on milling cutters.**

Milling tool materials

The most common materials in milling tools include:

Carbon steel: This is the most inexpensive material of the bunch and includes 0.6-1.5% carbon with small amounts of manganese and silicon. You'll typically find this material used for low-speed operations in twist drills, forming tools, milling cutters, and turning.

High-speed steel (HSS): This material combines chromium, tungsten, and molybdenum to give HSS improved hardness, toughness, and wear resistance over carbon steel. HSS tools are generally more expensive than others, but they're built to last and provide a high material removal rate for both ferrous and nonferrous materials.

Solid carbide: This material is more resistant to wear than HSS but also prone to chipping instead of wearing out evenly over time. Because of this, solid carbide used mainly in finishing applications in newer milling machines or those with less spindle wear. Typically carbide tools are made by sintering carbide with another metal, like tungsten, titanium, or tantalum, giving these tools high heat resistance amd making them ideal for high-quality surface finishes.

Ceramics: Ceramics are corrosion-resistant and made from aluminium oxide and silicon nitride, Their heat and wear resistamce means they can function in high heat cutting environments where other tools would not. These tools are typically ideal for cast iron, hard steels, and super alloys.

Coating: Coating on milling cutters increase hardness, enhance tool longevity, and allow for faster cutting speeds. The most popular coatings include

Titanium Nitride (TiN): The standard finish used on alloy steel, aluminium, and plastic.

Titanium carbonitride (TiCN): Provides a better wear resistance than TiN.

Super-life Titanium Nitride (A1-TiN): The best coating for high feed/speed and high temperature appliations.

Job holding devices

Objectives: At the end of this lesson you shall be able to
- **explain vices, clamps, v-block, parallel block**
- **list the type of slotting tool and its tool angle.**

Work holding device

Work pieces that are to be machined should be held with support firmly such that there is no slackening of the work while machining. Otherwise the jobs will be damaged, dimensional accuracies will vary, and sometimes the cutting tool/ cutter itself will break. Due to this reason, the operator may be injured.

The different types of work holding/supporting devices are as follows.

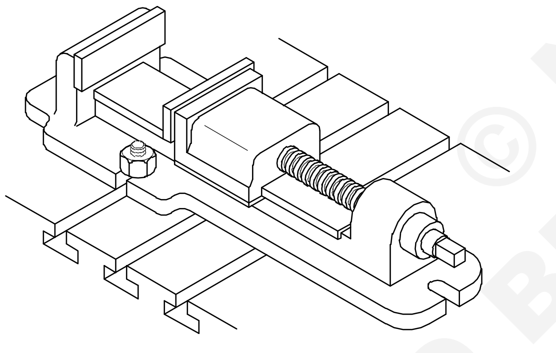
Machine vices and clamps

There are two main types of machine vices used on slotter.

Plain machine vice (Fig 1)

It is very robust in construction and so it is used on machines where it is used only for squaring and stock removal.

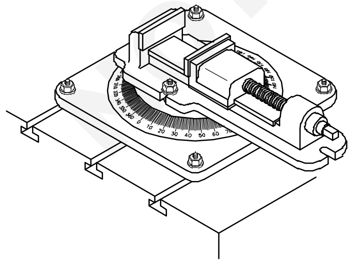
Fig 1



Swivel base vice (Fig 2)

It is the most commonly used on a slotter. The base can be swivelled to the desired angle to machine angular jobs.

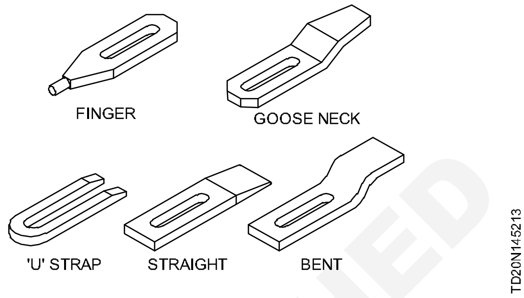
Fig 2



Strap clamps (Fig 3)

They are used for clamping the workpiece directly on the machine table. Straps come in different shapes and they are used in conjunction with T bolts/nuts.

Fig 3



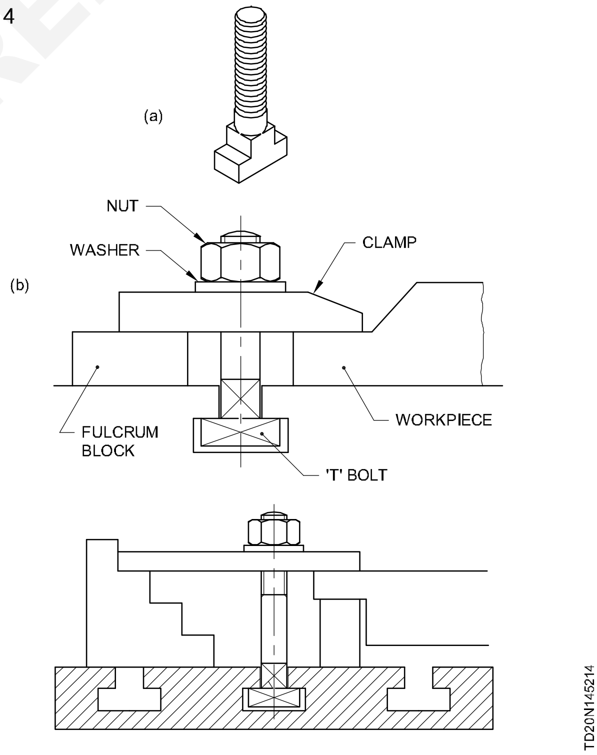
Tee bolts (Fig 4a)

These bolts having Tee shaped heads are suitable for the table slots of a machine. The body of the bolt is provided with sufficient length of threads so that the same bolt can be used for holding jobs of various thicknesses.

Different clamping arrangements are shown in Fig 4b to 12.

A normal clamping unit (Fig 4b) consists of a Tee bolt, nut washer, clamp and fulcrum block - plain/stepped.

Fig 4



Toe dogs and poppets

These are used for machining the top surfaces of huge jobs having no holes on the side, or small repetitive jobs as shown in Figs 5 & 6 (parallel plate A, workpiece B protective strip C, poppet D, toe dog E, stop F).

Fig 5

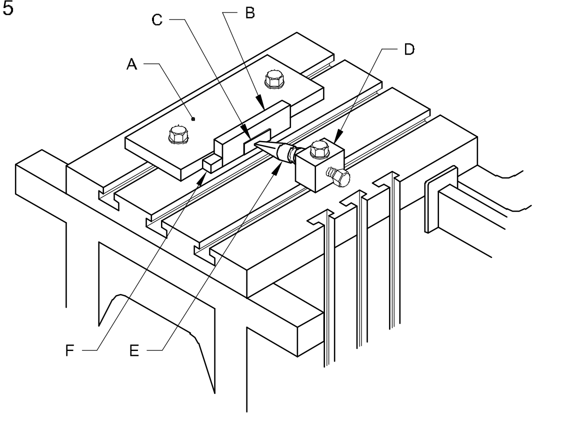
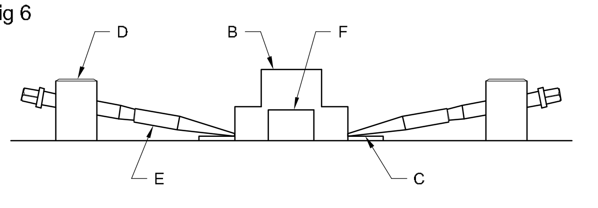


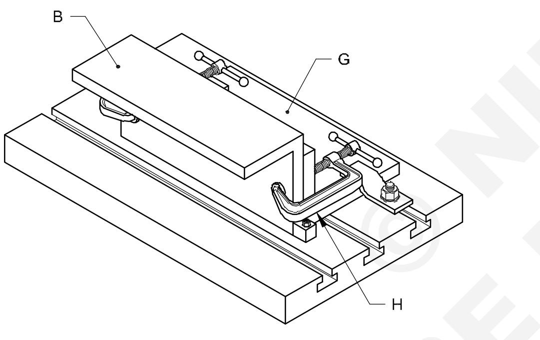
Fig 6



Angle plate (Fig 7)

'C' clamp (H)

Fig 7



Vee block (Fig 8)

These are 'V shaped blocks and are used for supporting round jobs in clamping.

Parallel strips (Fig 9)

These are perfectly ground steel strips used for elevating/ supporting the workpiece. They should be used only on machined surfaces.

Fig 8

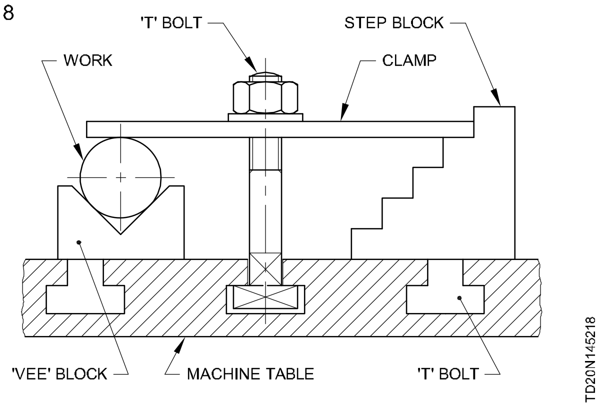
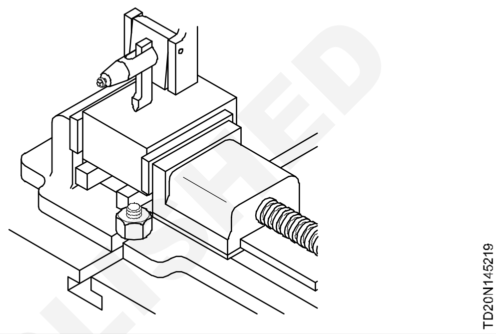


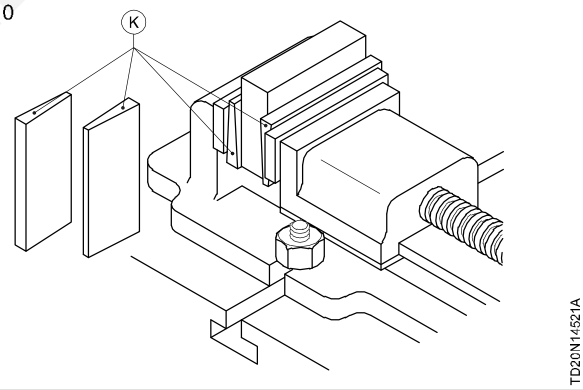
Fig 9



Degree parallels (K in Fig 10)

These are the steel strips with ground angular surface: Because of this, angular surfaces can be generated without swivelling the tool head.

Fig 10



Milling cutter holding devices

Objectives : At the end of this lesson you shall be able to
- **state the different types of arbors**
- **state the uses of arbors**
- **name the parts of an arbor**
- **specify the arbor**
- **brief the different methods of holding end mills.**

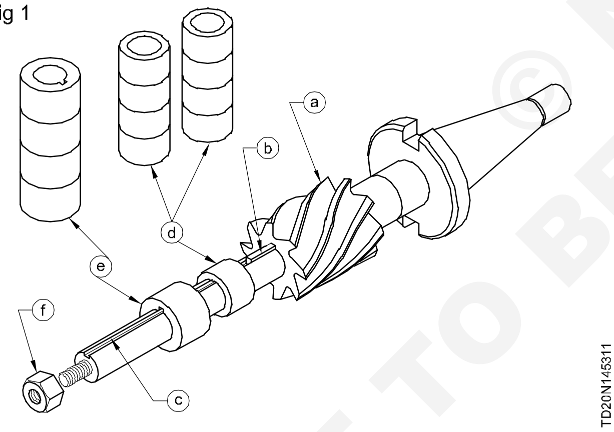
Types of arbors and their uses

An arbor is considered as an extension of the machine spindle on which milling cutters are mounted. Arbors are provided with quick-release taper shanks for proper alignment with the spindle. There are two types of arbors, normally used for holding the cutters. They are (1) long arbor and (2) short or stub arbor.

Long arbor (Fig 1)

Long arbors are used for holding cutters in both horizontal and universal milling machines. The milling cutter (a) is driven by a key (b) which fits into the keyway (c) on the arbor and cutter. This prevents the cutter from turning on the arbor. The spacer (d) and bearing bushings (e) hold the cutter in position on the arbor after the nut (f) has been tightened.

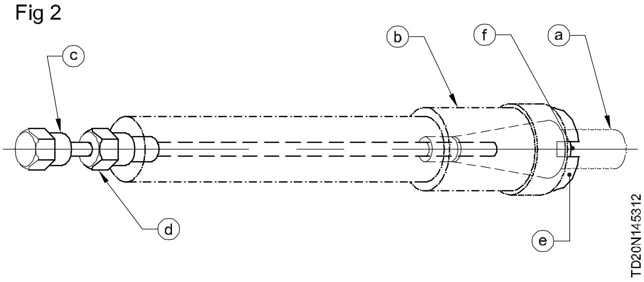
Fig 1



The tapered end of the arbor (a) is held securely with the machine spindle (b) by a draw-in bar (c) and lock-nut (d). (Fig 2) The flange (e) has two notches (f) to engage with the spindle tennon for transmitting the power.

The outer end of the arbor assembly is supported by the bushing and the arbor support.

Fig 2



Long arbors with I.S.O.taper shanks are available in different diameters. The normal diameters used commonly are Ø16, Ø22,Ø27, Ø32, Ø40 and the taper is ISO40/50.

The arbor is designated by the taper number, diameter and length.

Example ISO40 x Ø 22 x 500 mm.

ISO40 = Type of taper

Ø 22 = diameter of the arbor in 'mm'

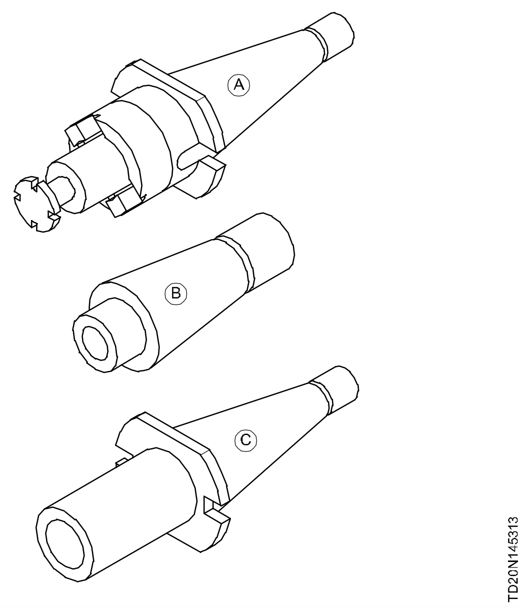
500mm = length of the arbor

Stub arbor

Stub arbors are used to mount various tapers of cutters in the spindle of horizontal and vertical milling machines.

The arbors are held with the machine spindle by a taper and a draw-in bar. The arbors are of three types (A), (B) & (C) as shown in Fig 3.

Fig 3

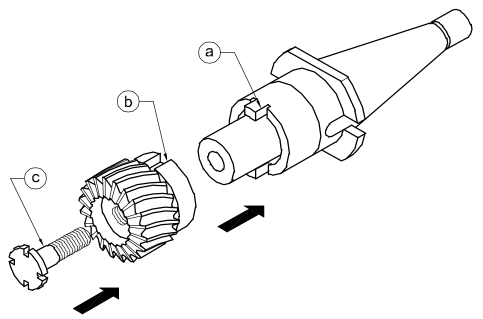


Types

Type A (Fig 4) is used to mount the shell end mills and similar cutters.

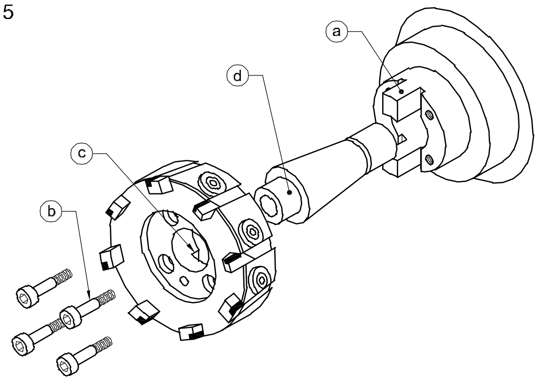
The cutter is pushed on the arbor so that the arbor key (a) fits with the slot (b) on the cutter. The cutter is tightened on the arbor using the screw (c).

Fig 4



Type B (Fig 5) is used to mount large face milling cutters. It is made with a centralizing spigot (a) to ensure that the cutter is centralized with the cutter spindle.

Fig 5



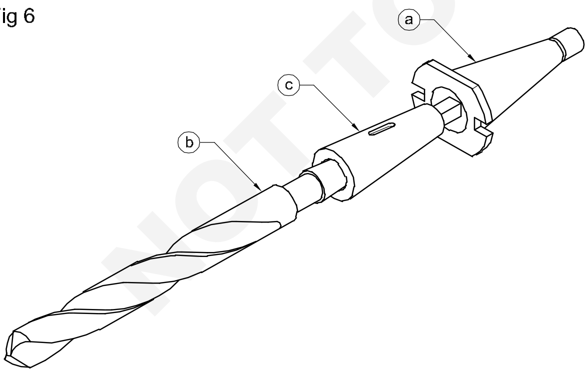
The cutter is held on the arbor by four screws (b). It has a slot (c) which fits over the spindle (d) to provide the drive.

Type C (Fig 6) is a Morse taper adapter arbor (a). It is used to hold drills, reamers, chucks (b), etc. which have taper shanks and also Morse taper sleeves (c) which are used to adapt a Morse taper to a larger taper.

According to B.I.S. specifications stub arbors with Morse taper shanks are available from 13 to 27 mm in diameters.

The stub arbor is designated by the taper number, diameter and length.

Fig 6



Collet chuck (Fig 7)

The chuck is supplied with a set of spring collets (Fig 8) in various sizes to suit the shanks of standard parallel shank cutters.

The chuck body Fig 9(a) is mounted in the machine spindle nose in the same way as the horizontal milling arbor.

Fig 7

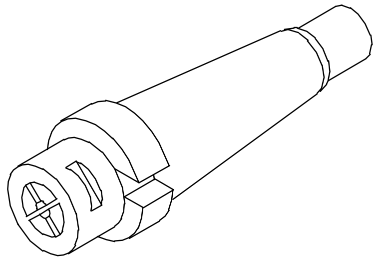


Fig 8

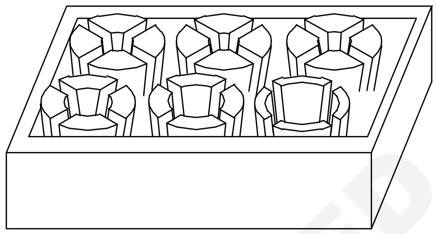
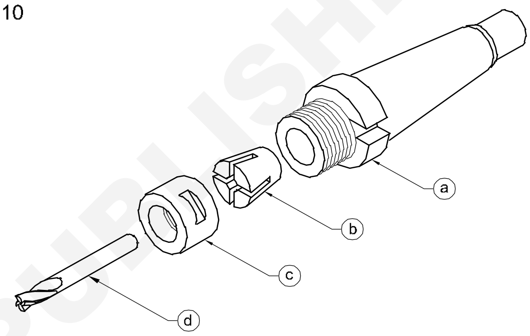


Fig 10



The collet (b) (of the same size as that of the cutter shank) is pushed in to the chuck body. The nut (c) is screwed on until it just grips the collet.

The cutter (d) is inserted into the collet and then the nut is tightened using the special spanner supplied with the chuck.

Adapters

These are used to reduce the internal taper in the work spindle, so that it fits on required arbor or cutter. (Fig 10) This type is used for holding the cutters with internal thread. Another type of adapters (shown in Fig 11) with Morse taper and flat tongs is used for holding taper shank end mills with tongs.

Fig 10

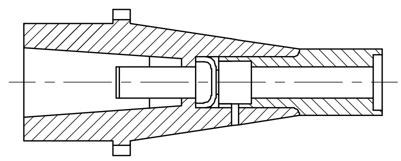
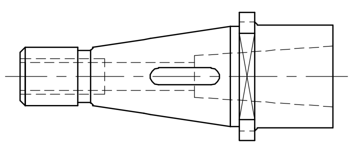


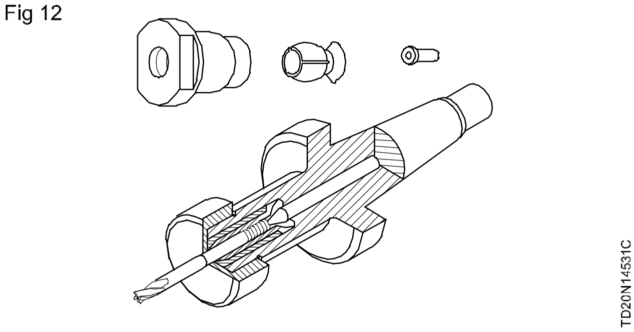
Fig 11



Self-locking chucks (Heavy duty chucks)

The cutter is provided with fine pitch thread at the end of the cutter shank. The cutter is mounted by turning with the thread provided into the chuck body and clamping. (Fig 12)

Fig 12



The common methods used to mount end mills in vertical milling machine are shown in Figs 13 to 16.

Fig 13

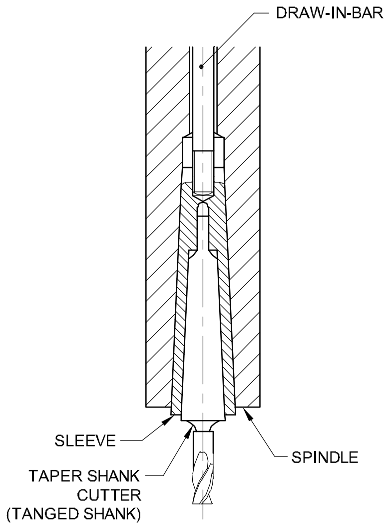


Fig 14

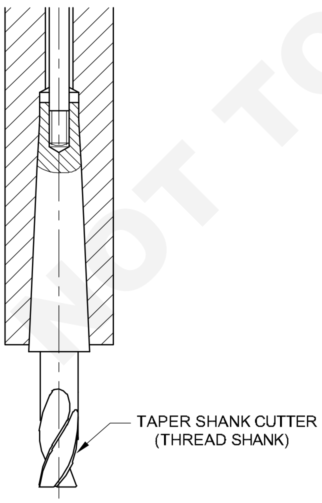


Fig 15

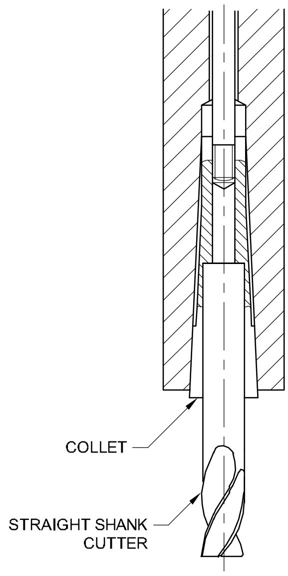
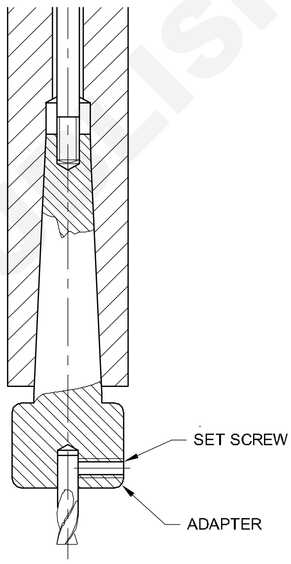


Fig 16



Milling process

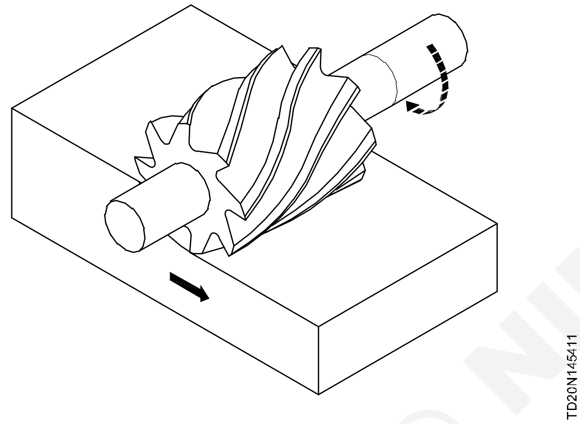
Objective: At the end of this lesson you shall be able to
- **state the different milling processes.**

Although the majority of operations performed on a knee and column type machine are either plain milling or side milling, several other operations or combinations of operations may be performed too.

Plain or slab milling (Fig 1)

It is the production of a horizontal flat surface parallel to the axis of the milling machine. (Fig 1) The workpiece is held in a vice or fixture, or fastened directly on to the table.

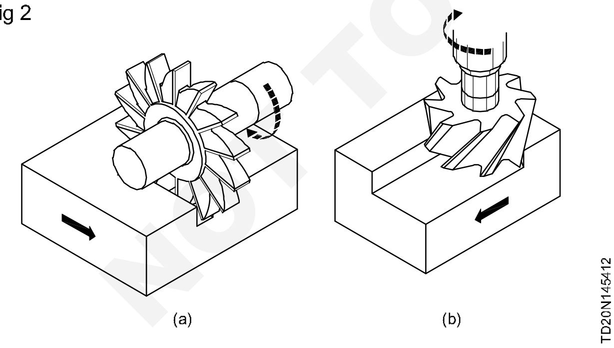
Fig 1



Side milling (Figs 2 a & b)

It is the process of machining a vertical flat surface perpendicular to the axis of the milling machine arbor. This operation is performed by the combined action of the peripheral and side teeth on a side and facing cutter.

Fig 2



Face milling

It is done to produce a flat surface parallel to the column of the machine. This is done by means of a face milling cutter mounted on the vertical milling machine spindle. (Fig 3). Face milling may also be done in a horizontal milling to produce horizontal flat surfaces. (Fig 4)

Fig 3

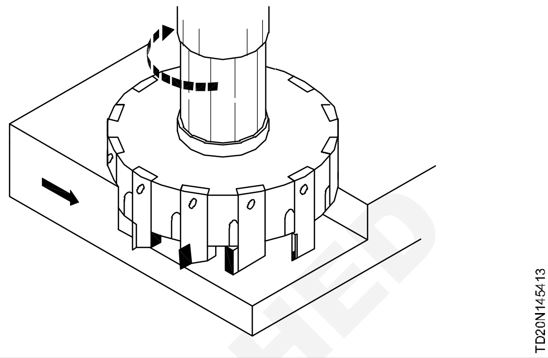
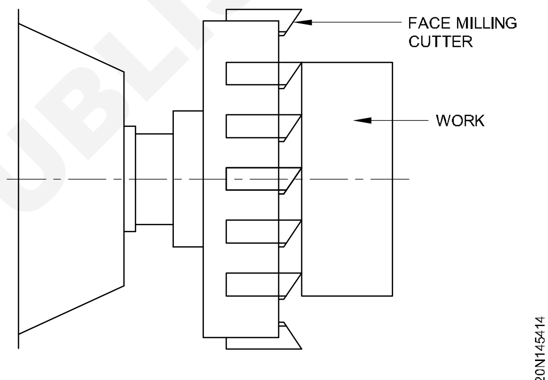


Fig 4

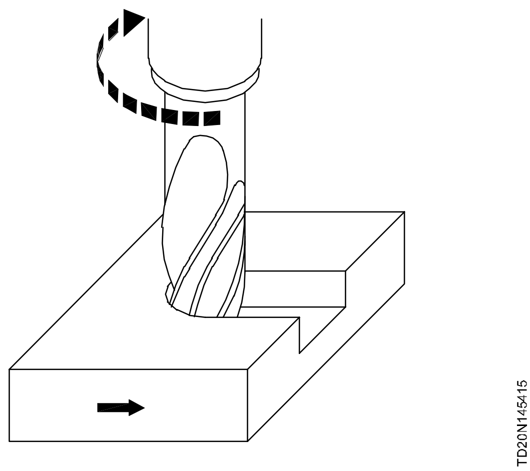


FACE MILLING OPERATION

End milling (Fig 5)

It is an operation similar to face milling but done using a much smaller cutter. Cutting is done on the end of the cutter as well as on the periphery. This operation is used for facing small surfaces, milling slots or grooves, producing internal recesses and for truing the edges of a workpiece.

Fig 5



Up milling and down milling

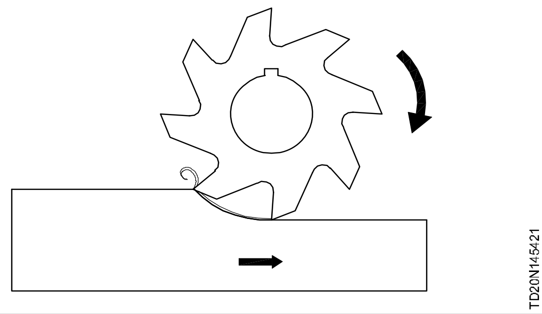
Objectives : At the end of this lesson you shall be able to
- **distinguish between 'up' milling and 'down' milling**
- **state the advantages and disadvantages of 'up' milling and 'down' milling.**

The two operating procedures are distinguished by the directions in which the teeth of the cutter and the feed of the workpiece move.

Up-cut milling

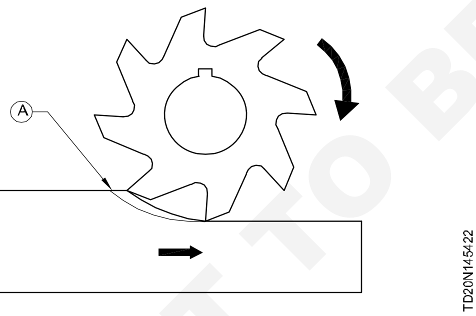
The most commonly used method of feeding is to bring the work against the direction of rotation of the cutter. (Fig 1) This is otherwise known as convention milling. This is the most commonly used method.

Fig 1



In up-milling the removal of chip by each cutting edge starts at the thinnest part of the chip (a) and progresses to the thickest part. (Fig 2) The cutting edge slides in the material before it starts to cut. This scraping causes a good deal of heat and wear on the cutter. As the cutter teeth emerge from the material, the accumulated cutting forces are suddenly released.

Fig 2



The cutter and workpiece suddenly spring apart, the machine chatters, and the material surface is flawed by ripples. The spindle for the milling is tensioned in the direction of the feed. The forces involved are taken up by the flank of the thread in contact.

Advantages

Up-cut milling is feasible on any milling machine. The hard skin of a casting is machined off from underneath.

The cutter first penetrates the less hard layer within.

Chances of cutter breakage are less.

Disadvantages

The workpiece tends to get lifted.

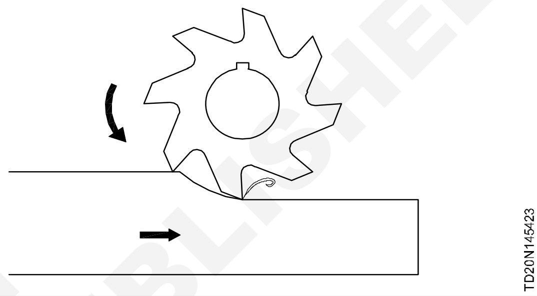
The scraping process makes tool service intervals short.

It is at a maximum force while the metal chip is being milled off.

Down-cut milling

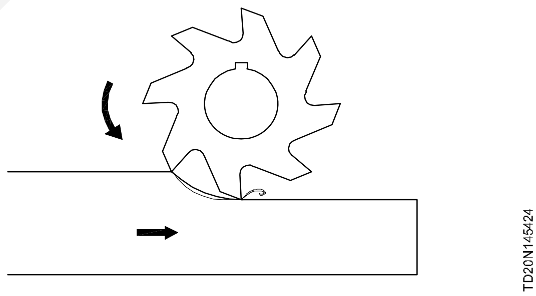
In down-milling or climb-milling the feed moves in the same direction as the rotation of the cutter. (Fig 3)

Fig 3



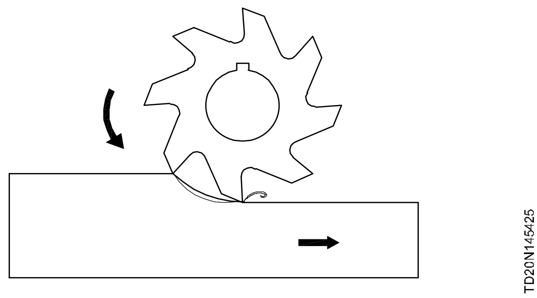
In this method, chip removal starts at the thickest part. The cutter cuts into the material straightway and does not slide. (Fig 4) As a result less heat is developed and there is less wear on the cutter.

Fig 4



In this method, the workpiece is pressed down on the work table by the cutting pressure and thus prevents the workpiece from lifting. (Fig 5) This is an advantage, especially when milling long workpieces.

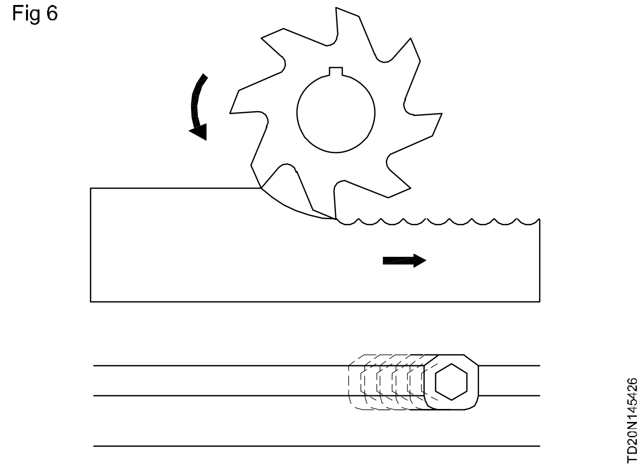
Fig 5



Because in climb-milling the cutter is pulling the workpiece, it should only be carried out on a machine having a 'backlash eliminator'. This is because the backlash eliminator takes up any clearance in the feed drive mechanism and prevents the cutter from being pulled along.

If down-milling is tried on a machine without a backlash eliminator, the workpiece will move along in a series of jerks (Fig 6), resulting in very poor finish and even breakage of the cutter.

Fig 6



Advantages

- The service of the cutter is comparatively longer.
- The cutting rates are higher.

Disadvantages

- There are restrictions on using down-cut milling.
- As a tooth starts to cut, the cutter tends to climb.
- Play in the spindle thread causes the milling table to move jerkily.
- Displacing the table leads to more material being removed than is planned.

The consequence of the above are:

- the cutter teeth break
- the workpiece is damaged
- there is risk of the workpiece being wrenched or pulled free.

This method is not recommended unless the machine table is fitted with a backlash eliminator.

Cutting speed and feed

Objectives: At the end of this lesson you shall be able to
- **calculate the revolution per minute for various cutters and materials**
- **select and calculate proper feeds for various cutters and materials**
- **explain the correct procedure for taking roughing and finishing cuts.**

Cutting speeds, feeds and depth of cut

The efficiency of a milling operation depends upon the cutting speed, feed, and depth of cut.

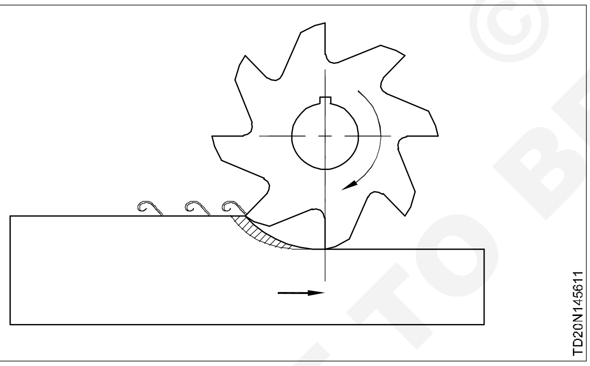
If the cutter is run too slowly valuable time will be wasted, while excessive speed results in loss of time in replacing and regrinding cutters. Somewhere between these two extremes is the efficient cutting speed for the material being machined.

The rate at which the work is fed into the revolving cutters is important. If the work is fed too slowly time will be wasted and chatter may occur which shortens the tool life of the cutter. If the work is fed too fast, the cutter teeth can be broken. Much time will be wasted if several shallow cuts are taken instead of one deep or roughing cut. Therefore speed, feed and depth of cut are the three important factors in any milling operation.

Cutting speed

The cutting speed for a milling cutter is the speed at which the cutting edge or tooth cuts into the workpiece. (Fig 1)

Fig 1



It is expressed in metres per minute.

The following important factors must be considered when determining the proper revolutions per minute at which to machine a metal.

- Type of work material
- Cutter material
- Diameter of the cutter
- Surface finish required
- Depth of cut being taken
- Rigidity of the machine and work set up

Since different types of metals vary in hardness, structure and machineability, different cutting speeds must be used for each type of metal and for various cutter materials.

The cutting speeds for the more common metals for HSS milling cutters are shown in Table 1.

Calculation

$$\text{Cutting speed (V)} = \frac{\pi DN}{1000}\ \text{m/min}$$

$$N(\text{r.p.m}) = \frac{V \times 1000}{3.1416 \times D}$$

Since only a few machines are equipped with a variable speed drive which allows them to be set to the exact calculated speed, a simplified formula can be used to calculate the revolution per minute.

The p(3.1416) on the bottom line of the formula will divide the 1000 of the top line approximately 320 times. This results in a simplified formula which is close enough for most milling operations.

$$N(\text{r.p.m}) = \frac{V\,(m) \times 320}{D\,(mm)}$$

where 'D' is diameter of the cutter.

Example

Calculate the revolution per minute required for f75 mm high speed steel cutter when cutting machine steel. (V = 30 m/min.)

$$\text{r.p.m} = \frac{30 \times 320}{75} = \frac{9600}{75} = 128$$

From the Table 2 the intersection of ∅ 75mm and cutter speed of 30 m/min. is in between 115 and 140 r.p.m. This can be taken as 128 r.p.m. as calculated.

> **Too fast a speed will shorten the cutter tool life; too slow a speed will waste time.**

Milling feeds and depth of cut

The two other factors which affect the efficiency of a milling operation are the milling FEED or the rate at which the work is fed into the milling cutter and the DEPTH of CUT taken at each pass.

Feed

Feed is the rate at which the work moves into the revolving cutter. It is measured in millimetres per minute (m/min.)

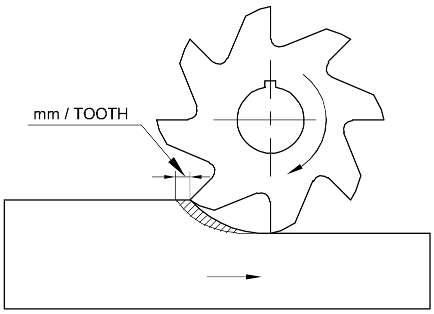
Feed rate is specified in mm/min.

The feed is expressed in milling machines by following three different methods.

Feed per tooth

Feed per tooth is defined by the distance the work advances and the time between engagement by two successive teeth. It is expressed in mm/tooth of the cutter.(Fig 2)

Fig 2



Feed per cutter revolution

Feed per cutter revolution is the distance the work advances in the time when the cutter runs through one complete revolution. It is expressed in mm/revolution of the cutter.

Feed per minute

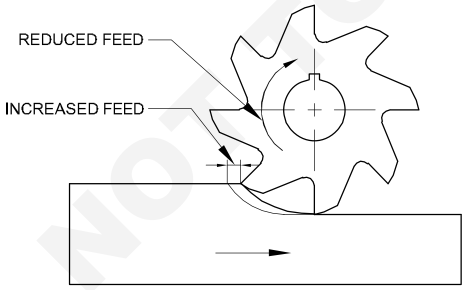
Feed per minute is defined by the distance the work advances in one minute. It is expressed in mm/ minute.

The rate of feed has an effect on the life of the cutter. An increase in feed, using the same cutting speed and depth of cut will reduce the amount of wear of the cutter.

In general we can say that the

- cutting speed should be be reduced when feed is increased (Fig 3)

Fig 3

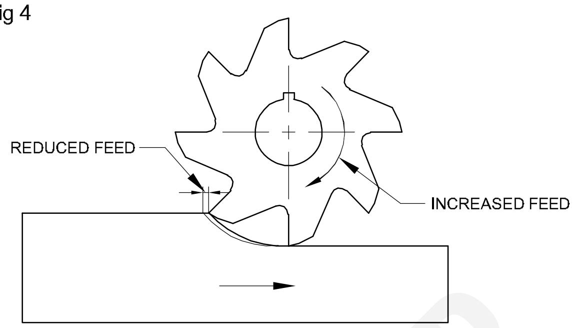


- cutting speed should be increased when feed is reduced. (Fig 4)

The feed rate on a milling machine depends on a variety of factors such as

- width and depth of cut
- type of cutter
- sharpness of the cutter
- workpiece material
- strength and uniformity of the workpiece
- type of finish and accuracy required
- power and rigidity of the machine.

Fig 4



Calculation

The formula used to find the work feed is

feed mm/min.(S) = N x Cpt x r.p.m.

where N = number of teeth in milling cutter

Cpt = chip per tooth for a particular cutter

r.p.m. = revolution per minute of the milling cutter.

Example 1

Calculate the feed in mm/min. for a Æ75, six-teeth helical carbide milling cutter when machining a cast iron workpiece (V = 60 and Cpt = 0.18).

First calculate the r.p.m. of the cutter

$$\text{r/min.} = \frac{60 \times 320}{75} = 256$$

Feed (mm/min.) = N x C.p.t x r.p.m.

= 6 x 0.18 x 256

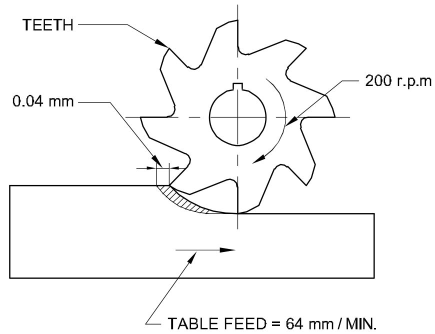
= 276.4

= 276 mm/min.

The spindle speed (revolution per minute) must always be calculated before the feed rate can be calculated.

Example 2 (Fig 5)

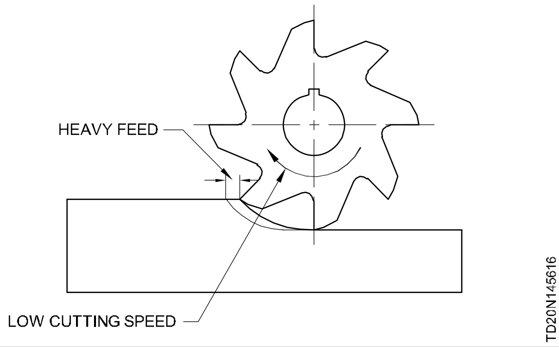
Fig 5



A cutter having 8 teeth is to have a feed of 0.04 mm/tooth. The spindle speed is to be 200 r.p.m. What feed, in mm/min. should be set on the machine?

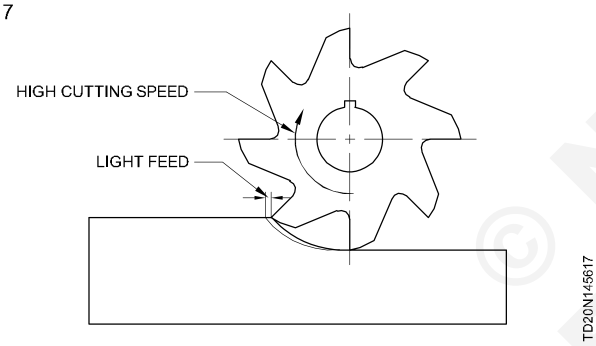
While rough milling, where the purpose is to remove surplus metal as quickly as possible and finish is not important, a heavy feed and low cutting speed are used. (Fig 6) However, the cutting speed should not be reduced too much as the cutter would then be operating under very heavy cutting forces.

Fig 6



For finish milling, the quality of the surface finish is, ofcourse, important. Therefore, a light feed and a high cutting speed are used. (Fig 7)

Fig 7



Cutting speed should be reduced when the feed is increased.

Cutting speed should be increased when the feed is reduced.

Depth of cut

The depth of cut is the depth to which the cutter penetrates the workpiece surface during a given cut. It is the perpendicular distance (Fig 8) measured between the original and the final surface of the workpiece.

Where a smooth and accurate finish is needed, it is a good practice to take roughing and finishing cuts. Roughing cuts should be deep with a feed as heavy as the work and machine will permit with low cutting speed. (Fig 9) Heavier cuts may be taken with helical cutters having fewer teeth than with those having many teeth. Cutters with fewer teeth are stronger and have greater chip clearance than cutters with more teeth.

Fig 8

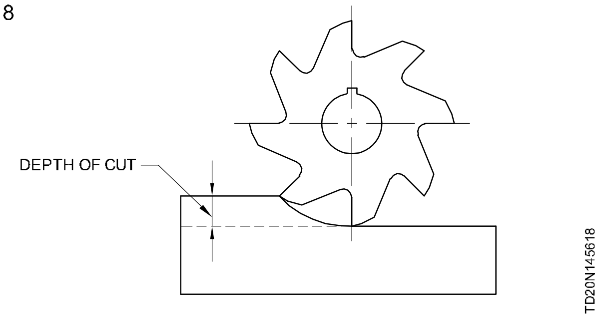
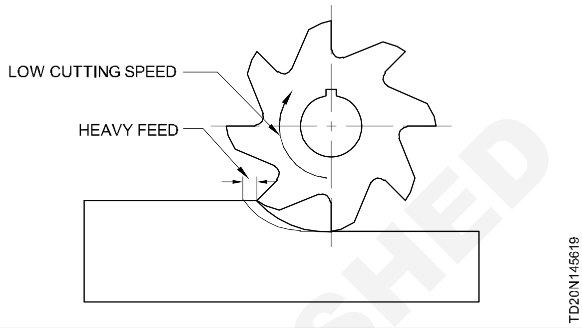


Fig 9



Finishing cuts should be light with a fewer and finer feed than is used in roughing cuts. (Fig 10) The depth of cut should be atleast 0.4 mm. Light cuts and extremely fine feeds are not advisable, since the chip taken by each tooth will be thin and the cutter will often rub the surface of the work. When a fine finish is required, the feed should be reduced rather than the cutter speed; more cutters are dulled by high speeds than by high feeds.

The table 1 shows the cutting speed (V) in metres per minute (m/min) for various materials, using high speed (HSS) milling cutters of various types. They must be considered as average values which may vary according to actual working conditions.

Fig 10

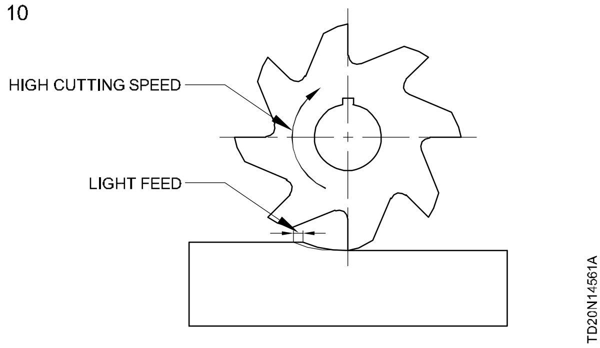


Table 1

Table for selecting cutting speeds for high speed steel milling cutters

Material to be machined	BHN hardness	Shell and Mill	End mill	S& F cutter	Cylind. cutter	Slot'g cutter	Form cutter	In.tooth face mill
Mild steel	150	20-30	20-30	15-25	15-25	15-25	30-45	20-30
Medium carbon steel	200	15-25	15-20	15-20	20-30	15-20	15-25	15-25
High carbon steel	300	10-15	10-15	10-15	12-20	10-15	13-20	13-20
Stainless steel	200	22-30	22-30	15-25	15-25	20-30	15-25	20-30
Malleable iron	160	15-22	15-22	15-20	15-20	20-30	15-20	18-25
Soft cast iron	180 max	15-20	15-25	15-20	15-20	20-30	15-20	15-25
Hard cast iron	Over 180	13-17	10-15	10-15	10-15	10-25	10-15	13-17
Hard brass & hard bronze	-	40-60	40-60	30-45	30-45	70-90	30-45	50-60
Soft brass & soft bronze	-	40-60	40-60	25-35	25-35	70-90	25-35	40-50
Copper	-	30-45	30-45	30-45	30-45	70-90	25-35	50-60
Aluminium alloy	-	200-300	200-300	150-300	150-300	200-300	150-250	200-400

Carbide cutters are able to cut at a much higher speed than HSS cutters and they are made in a variety of grades. If you are going to use a carbide cutter, ask your instructor what cutting speed you should select, as he will have the values for the particular grade of carbide used in the cutter in your workshop.

Machining time for milling

Objectives: At the end of this lesson you shall be able to
- **state the importance of machining time**
- **calculate the machining time.**

Importance of machining time

Time means money. Any engineering product should be manufactured at the shortest time possible to the required accuracy and quality. Machining time is one of the main factors for cost estimation of a product.

It is always good to plan how much time is needed for a particular milling operation. This will also enable you to plan for the appropriate machine, the right type of cutters, selection of speed, feed etc. In order to save time and increase productivity.

The total machining time depends on

- the length of the workpiece to be machined
- the method of milling employed i.e. rough or finish milling
- the size of the cutter.

Selection of a milling cutter should be such that the width of the job is covered by the cutter in one pass. If the width of the workpiece is more than the width of the cutter, machining has to be done by more than one pass. The total time therefore depends on the number of passes also.

The total time required to mill a surface for any milling operation is calculated from the formula

Total time for milling (th)

$$= \frac{\text{total travel of the cutter (L)}}{\text{feed per revolution (S)}}$$

The total travel of the cutter 'L' is equal to l + la +lx

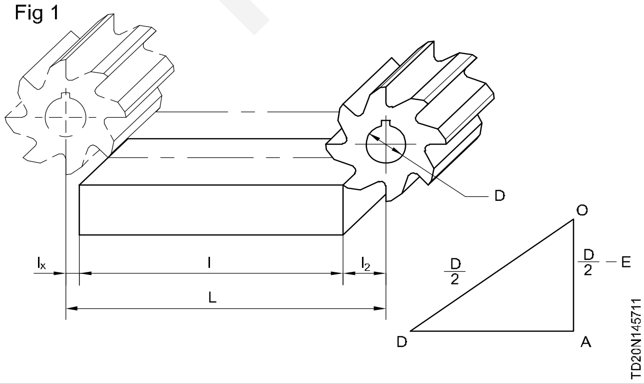
where l = length of workpiece to be milled

la = run up travel of the cutter from start of cut to go to the full given depth.

lx = run over the minimum distance the cutter has to move away from the work after the cut is taken.

Calculation of la (Fig 1)

Fig 1



By Pythogoras Theorem, we have

$$\left(\frac{d}{2}\right)^2 = (la)^2 + \left(\frac{d}{2} - a\right)^2$$

$$(la)^2 = \left(\frac{d}{2}\right)^2 - \left(\frac{d}{2} - a\right)^2$$

la = $d.a - a^2 = a(d-a)$

la = $\sqrt{a(d-a)}$

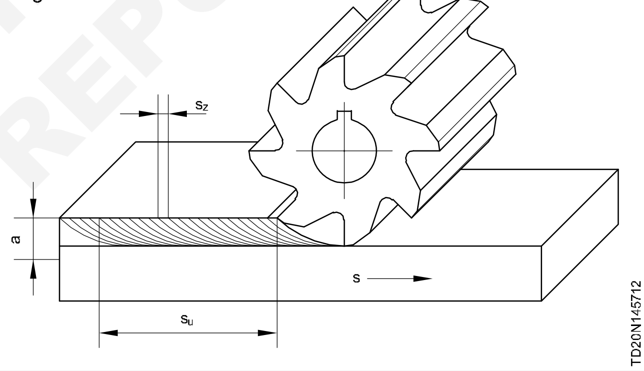
Where

d = cutter diameter

a = cutting depth.

Calculation of 'U' (Fig 2)

Fig 2



Feed per tooth = Sz in mm

Feed per revolution S = Sz x Z m

Feed for 'n' revolutions 'u' = Sz x Z x N mm/min.

Where

'U' = total feed per minute

Z = number of teeth of cutter

n = r.p.m. of the cutter

Calculation of 'th' (Fig 3)

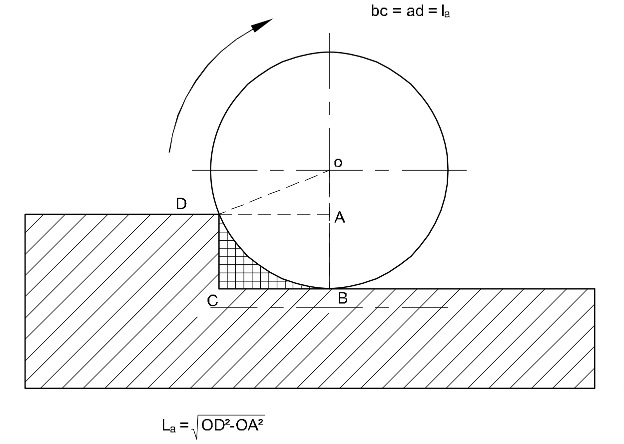
$$\text{The milling time} = \frac{\text{Cutter travel x number of travels (i)}}{\text{feed per minute}}$$

$$= \frac{L \times i}{u} = \frac{L \times i}{Sz \times Z \times n}$$

$$= \frac{\text{Cutter travel x number of travels}}{\text{feed per tooth x no. of teeth of the cutter x r.p.m.}}$$

To determine the milling time, the total travel of the cutter L is calculated and it is equal to l + la + lx where l is the length of work, la is the run up to be calculated as equal to $\sqrt{a(d-a)}$ and lx will be the known value.

Fig 3



Example (Fig 4)

A milling cutter of 80 mm diameter has nine teeth. It is to mill a 240 mm long workpiece. The run over is 4 mm and the depth of cut is 5 mm. A cutting speed of 12 metres/ min. and feed per tooth of 0.15 mm have been selected.

Determine the milling time.

$$n = \frac{V \times 1000}{d \times \pi} = \frac{12 \times 1000}{80 \times \pi} = 48 \text{ r.p.m}$$

$$\text{Total milling time} = \frac{L \times i}{Sz \times Z \times n}$$

(L = l + la + lx)

$$la = \sqrt{a(d-a)} = \sqrt{5(80-5)}$$

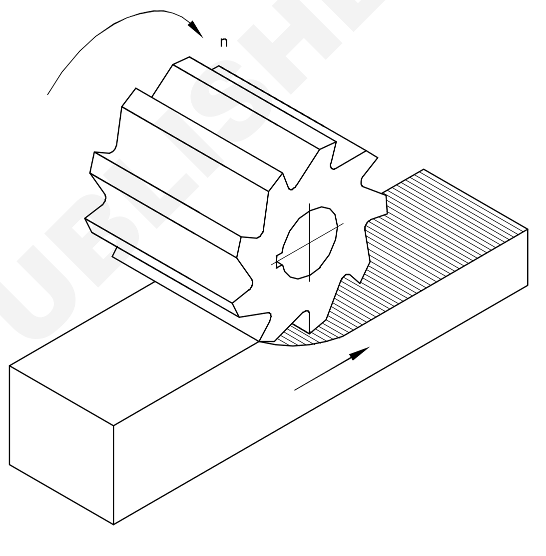
$$= \sqrt{5 \times 75 \text{ mm}} = 19.4 \text{ mm}$$

la = 4 mm

L = l + la + lx = 240 + 19.4 + 4 mm = 263.4 mm

$$2 \text{ th} = \frac{263.4 \times 1}{0.15 \times 9 \times 48} = 4.07 \text{min.}$$

Fig 4



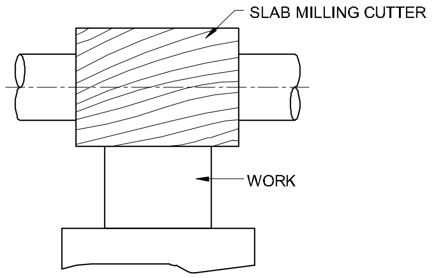
Milling machine operations

Objectives: At the end of this lesson you shall be able to
- **explain the plain, face, angular and form milling**
- **describe slot, gang and straddle milling**
- **explain up and down milling.**

Plain milling

It is the operation of production of a flat surface parallel to the axis of rotation of the cutter. It is also called as slab milling. Plain milling cutters and slab milling cutters are used to perform this operation. Fig 1 shows plain milling operation.

Fig 1

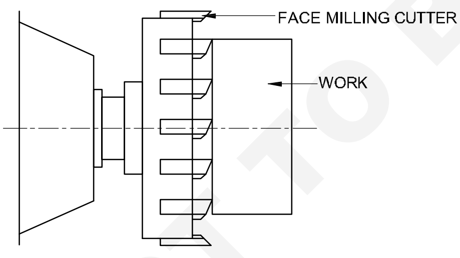


PLAIN MILLING OPERATION

Face milling

The face milling is the operation performed by the face milling cutter rotated about an axis at right angles to the work surface. End mills and side & face milling cutter are also used at times to perform this operation. Fig 2 shows face milling operation.

Fig 2



FACE MILLING OPERATION

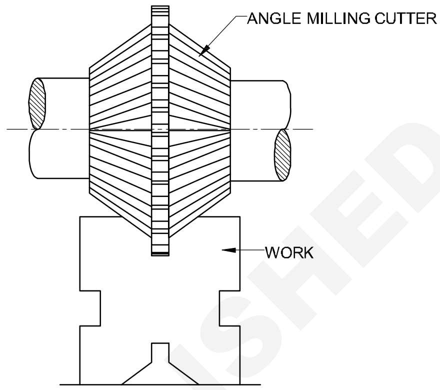
Angular milling

Production of an angular surface on a work piece other than at right angles to the axis of the milling machine spindle is known as angular milling. Example of angular milling is the production of the “V” blocks. Fig 3 shows angular milling operation.

Form milling

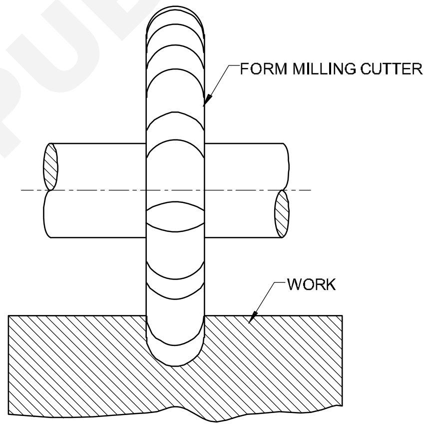
The form milling is the operation of production of irregular contours by using form cutters. Machining convex and concave surfaces and gear cutting are some examples of form milling. Fig 4 shows form milling operation.

Fig 3



ANGULAR MILLING OPERATION

Fig 4

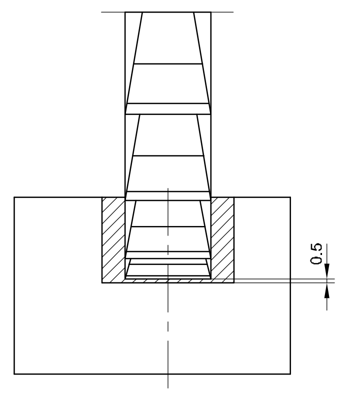


FORM MILLING OPERATION

Slot milling (Fig 5)

The operation of production of slot of different sizes can be produced in a milling machine by using a plain milling cutter and by an end mill or side milling cutter.

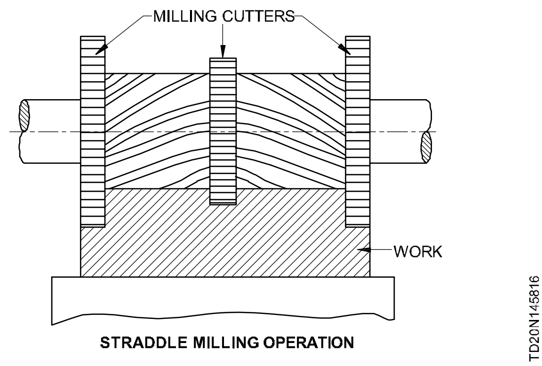
Fig 5



Gang milling

It is the operation of machining several surfaces of work simultanously by feeding the table against a number of cutters (either of same type of different type) mounted on the arbor of the machine. This method saves much of machining time and mostly used in production work. Fig 6 shows gang milling operation.

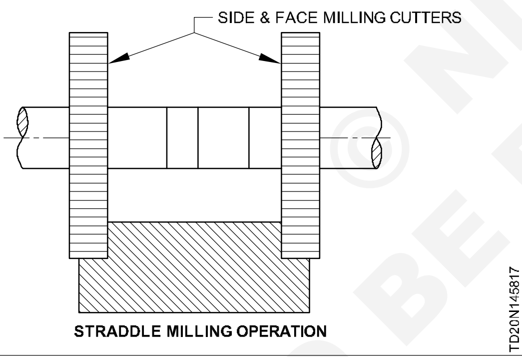
Fig 6



Straddle milling

It is the operation of production of two vertical surfaces on both sides of the work by two side milling cutters mounted on the same arbor. By using suitable spacing collars, the distance between the two cutters is adjusted correctly. The straddle milling is commonly used to produce square or hexagonal surfaces. (Fig 7)

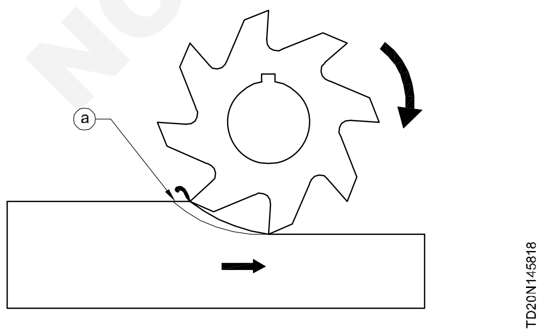
Fig 7



Up-cut milling

The most commonly used method of feeding is to bring the work against the direction of rotation of the cutter. (Fig 8) This is otherwise known as convention milling. This is the most commonly used method.

Fig 8



In up-milling the removal of chip by each cutting edge starts at the thinnest part of the chip (a) and progresses to the thickest part. The cutting edge slides in the material before it starts to cut. This scraping causes a good deal of heat and wear on the cutter. As the cutter teeth emerge from the material, the accumulated cutting forces are suddenly released.

The cutter and workipiece suddenly spring apart, the machine chatters, and the material surface is flawed by ripples. The spindle for the milling is tensioned in the direction of the feed. The forces involved are taken up by the flank of the thread in contact.

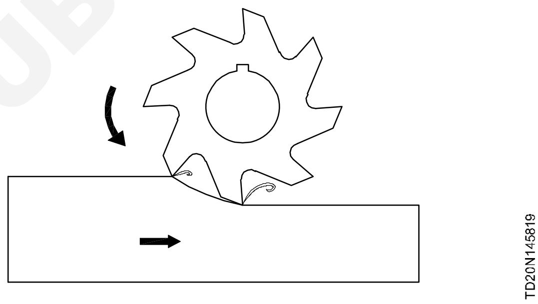
Down-cut milling

In down-milling or clamp-milling the feed moves in the same direction as the rotation of the cutter. (FIg 9)

In this method, chip removal starts at the thickest part. The cutter cuts into the material straight way and does not slide. As a result less heat is developed and there is less wear on the cutter.

In this method, the work piece is pressed down on the work table by the cutting pressure and thus prevents the work piece from lifting. This is an advantage, especially when milling long workpieces.

Fig 9



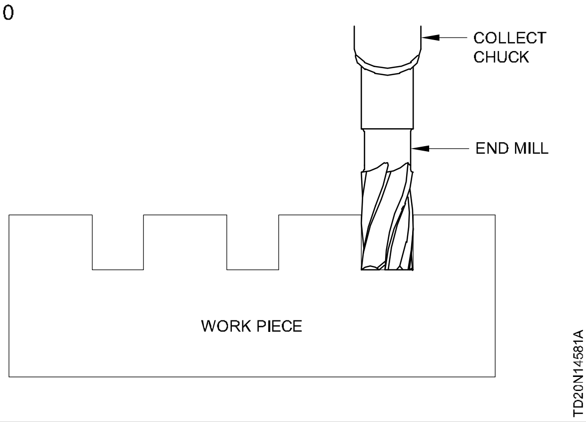
End milling (Fig 10)

End milling is the operation of producing a flat surface which may be veritcal, horizontal or at an angle in reference to the table surface.

The cutter used is an end mill. The end mill cutters are also used for production slot, grooves and key ways.

A vertical milling machine is more suitable for end milling operation.

Fig 10



Milling attachments

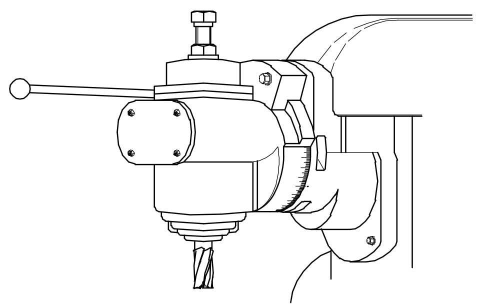
Objective: At the end of this lesson you shall be able to
- **state the different types of milling machine attachments and their application.**

The application of a milling machine to various machining operations can be enhanced by the use of different types of attachments used in conjunction with the fundamental features of the machine.

Vertical milling attachment (Fig 1)

This attachment is mounted on the front of the column and can be driven by the spindle of plain and universal milling machines. The attachment consists of a housing, a spindle head and bevel drive gears with 1:1 ratio. The spindle has a standard taper and drive lugs for the cutters.

Fig 1



HEAVY DUTY VERTICAL ATTACHMENT

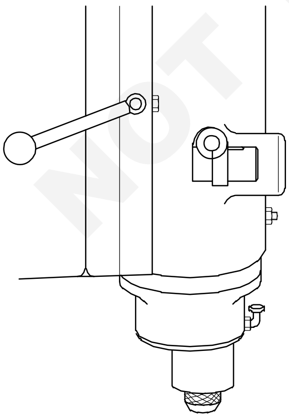
TD20N145911

The spindle housing has a swivel mechanism to permit the spindle to be angled with respect to the table of the milling machine. A graduated scale is fitted to the housing to permit setting of a selected angle. A lock is provided to hold the attachment at the selected angle.

It is used to perform milling operations which would otherwise need to be performed on a vertical milling machine.

High speed attachment (Fig 2)

Fig 2



HIGH SPEED ATTACHMENT

TD20N145912

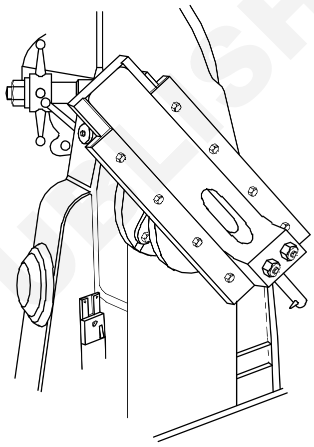
The high speed attachment is used to drive small milling cutters at high speeds. The attachment consists of a housing containing a step-up gearing and a small spindle by means of which the spindle speed can be increased by four or six times.

It may be fitted to the vertical or horizontal machine as required.

Slotting attached (Fig 3)

The slotting attached is used to provide a reciprocating drive for a single point cutting tool. It may be used for cutting at any angle in the vertical plane to produce keyways, slots and corners.

Fig 3



SLOTTING ATTACHMENT

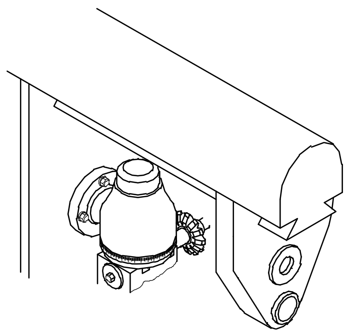
TD20N145913

Universal spiral attachment (Fig 4)

This attachment is used when it is required to mill spirals with a plain milling machine.

It consists of a housing fitted with a swivel plate for mounting on the face of the milling machine column. The body may then be swivelled about the axis of the machine spindle.

Fig 4



UNIVERSAL SPIRAL ATTACHMENT

TD20N145914

The attachment is fitted with a small spindle rotated by the machine spindle through the gearing in the housing.

The small spindle is also capable of swivelling with respect to the housing.

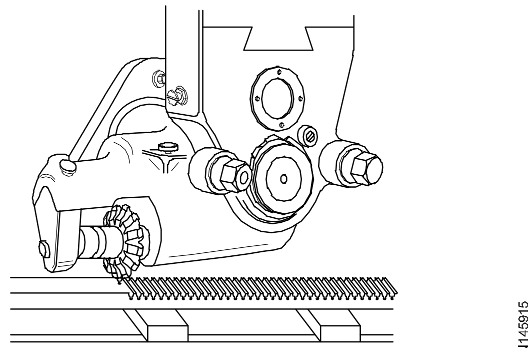
The combination of the two swivelling actions permits the small spindle to be set at any compound angle with respect to the column of the milling machine.

This attachment is particularly useful for cutting helical threads, gears, worms and racks.

Rack milling attachment

The rack milling is mounted between the face of the column and the outer support on a plain or universal milling machine. It consists of a fixed housing fitted with an angle drive and a spindle. The spindle axis is fixed parallel to the table. (Fig 5)

Fig 5

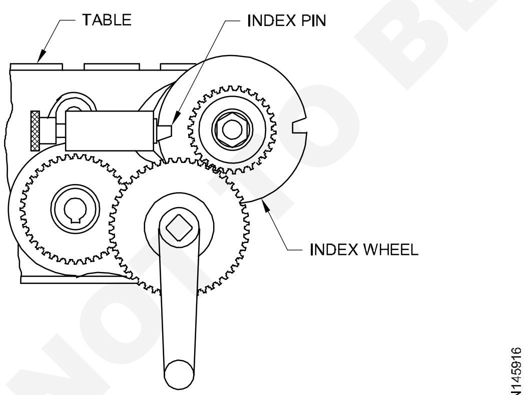


RACK MILLING ATTACHMENT

In rack milling, the cross-feed is used to move the cutter into the workpiece, and the longitudinal feed is used to index the cutter to produce the rack teeth.

A special rack indexing attachment (Fig 6) is used to move the work longitudinally, the exact amount needed for accurate spacing of the teeth.

Fig 6



RACK INDEXING ATTACHMENT

At one end of the table is fastened a bracket which carries a locking indexing wheel together with change gears for gearing to the table feed screw.

To index any required spacing, change gears are selected which will produce one or more complete turns of the indexing wheel. For each indexing operation, the index pin is withdrawn and the table advanced by turning the table feed screw until the pin drops into the slot again, and locks the wheel.

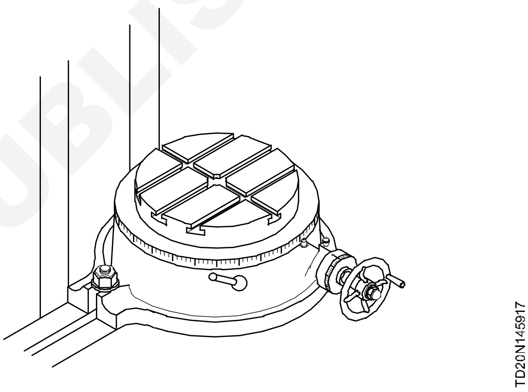
This method is positive and much more reliable than setting the table to a graduated dial directly fitted to the table feed screw.

Circular table attachment

This attachment is used for profile milling, surfacing quantities of small pieces in the one set up, and for many other circular operations in the horizontal plane such as slotting and dovetailing.

It consists of a base, a worm gear drive mechanism and a small circular work table. (Fig 7)

Fig 7



The base is bolted to the table and the drive mechanism connected with the feed mechanism of the milling machine. A crank for manual feed is also provided.

The workpiece is secured to the circular table and the table is rotated by the feed mechanism for circular cuts.

By combining the rotary motion with one or more of the other feeds of the machine, profiles of almost any shape can be milled. The hand crank can be replaced by an indexing device for requiring accurately spaced slots, holes or grooves.

Coolants

Objectives: At the end of this lesson you shall be able to
- **state the purpose of using coolant**
- **state the properties of coolant**
- **list the type of coolant.**

Coolant

It is matter/substance used to reduce the heat produced by tool and work. The heat affects the life and accuracy of machine, tool and job becomes hardened.

Purpose of coolant

To cool the job to avoid expansion by heat

To cool the cutting points of the tool and save temper and cutting efficiency.

To wash away the chip

To obtain a smooth finishing

To reduce friction between the tool and work

To prevent the machine from corrosion

Properties of coolant

Higher the viscosity

Good oiliness

Should have high fire point

Should be chemically stable

Low sulphur content (less than 3%)

Should be harmless to skin of operator

Odorless - Should not have bad smell.

Types of coolant

The most common machine coolants used today belong to one of two categories based on their oil content.

Oil based machine coolants - Including straight oils and soluble oils

Chemical machine coolants - Including synthetics and semi synthetics

Fluids vary in suitability for metal working operations due to their excellent lubricity while water miscible fluids provide the cooling properties required for most turning and grinding operations.

Oil based machine coolants

Straight oils - 100% petroleum oil

Soluble oils - 60% to 90% petroleum oil

Chemical machine coolants

Synthetics - No petroleum oils

Semi synthetics - 2% to 30% petroleum oil

Lubricants

Objectives: At the end of this lesson you shall be able to
- **state the purpose of using lubricants**
- **state the properties of lubricants**
- **state the qualities of a good lubricant.**

With the movement of two mating parts of the machine, heat is generated. If it is controlled the temperature may rise resulting in total damage of the mating parts. Therefore a film of cooling medium with high viscosity is applied between the mating parts which is known as a 'lubricant'.

A 'lubricant' is a substance having an oily property available in the form of fluid, semi-fluid, or solid state . It is the lifeblood of the machine, keeping the vital parts in perfect condition and prolonging the life of the machine. It saves the machine and its parts from corrosion, wear and tear, and it minimise friction.

Purposes of using lubricants

- Reduces friction.
- Prevents wear.
- Prevents adhesion.
- Aids in distributing the load.
- Cools the moving elements.
- Prevents corrosion.
- Improves machine efficiency.

Properties of lubricants

Viscosity

It is the fluidity of an oil by which it can withstand high pressure or load without squeezing out from the bearing surface.

Oiliness

Oiliness refers to a combination of wettability, surface tension and slipperiness. (The capacity of the oil to leave an oily skin on the metal.)

Flash point

It is the temperature at which the vapour is given off from the oil (it decomposes under pressure soon).

Fire point

It is the temperature at which the oil catches fire and continues to be in flame.

Pour point

The temperature at which the lubricant is able to flow when poured.

Emulsification and de-emulsibility

Emulsification indicates the tendency of an oil to mix intimately with water to form a more or less stable emulsion. De-emulsibility indicates the readiness with which subsequent separation will occur.

Methods of applying lubricant

Objective: At the end of this lesson you shall be able to
- **list and explain the different methods of lubrication.**

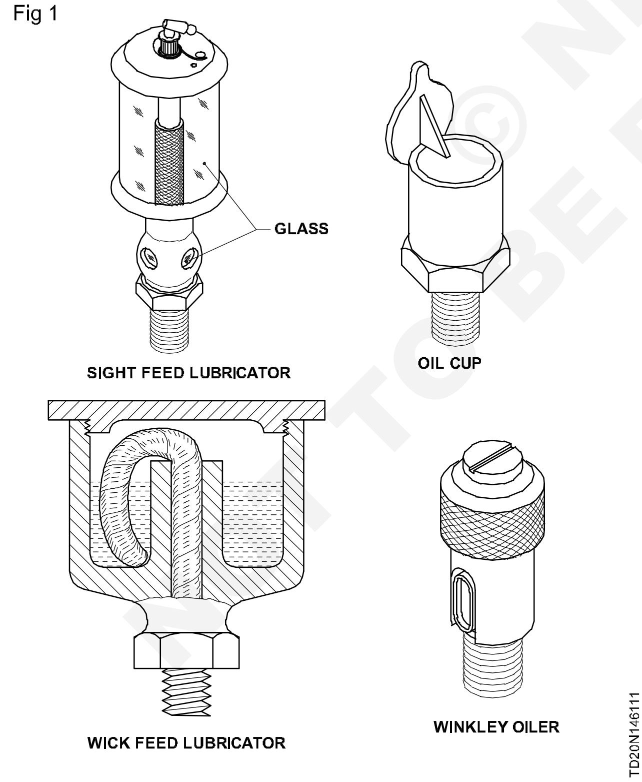
The following methods are used for efficient lubrication.

- Gravity feed method
- Force feed method
- Splash method

Gravity feed method

There are numerous ways of employing this principle, varying from the simple oil hole to the more elaborate wick and glass-sided drip feed lubricators in which the flow of the oil may be controlled and observed through the glass. A selection of these lubricators is shown in Fig 1.

Fig 1

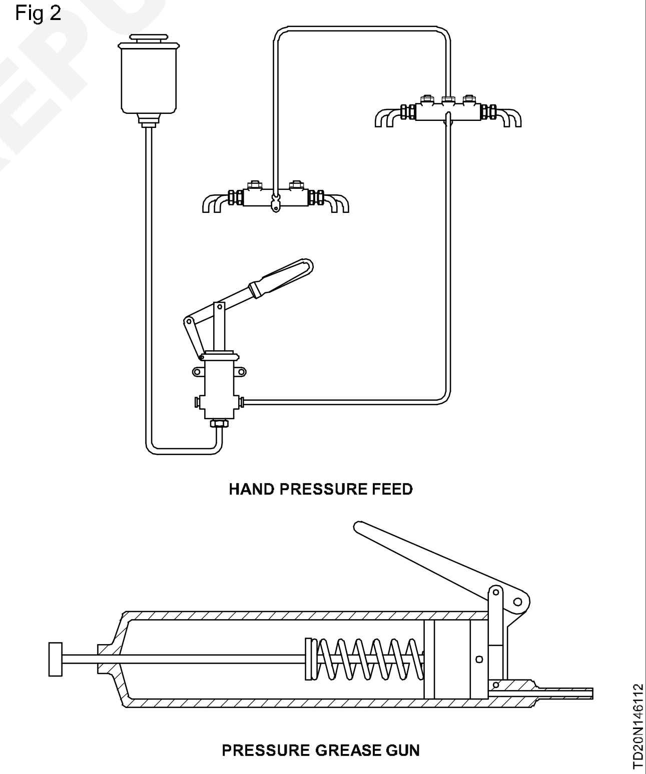


Force (Pressure) feed method

There are various systems of lubrication employing a pressure feed to the lubricant, and the most important of such systems may be classified roughly into the following.

- Continuous feed of oil under pressure to each bearing concerned. In this method an oil pump driven by the machine delivers oil to the bearings and back to a sump from which it is drawn by the pump.
- Pressure feed by hand pump in which change of oil is delivered to each bearing at intervals (once or twice a day) by the machine operator. (Fig 2)

Fig 2



- Oil or grease gun method. The oil hole leading to each bearing is fitted with a nipple and by pressing the nose of the gun against this and the lubricant is forced into the bearing. (Figs 3 a, b, c & d)

Fig 3

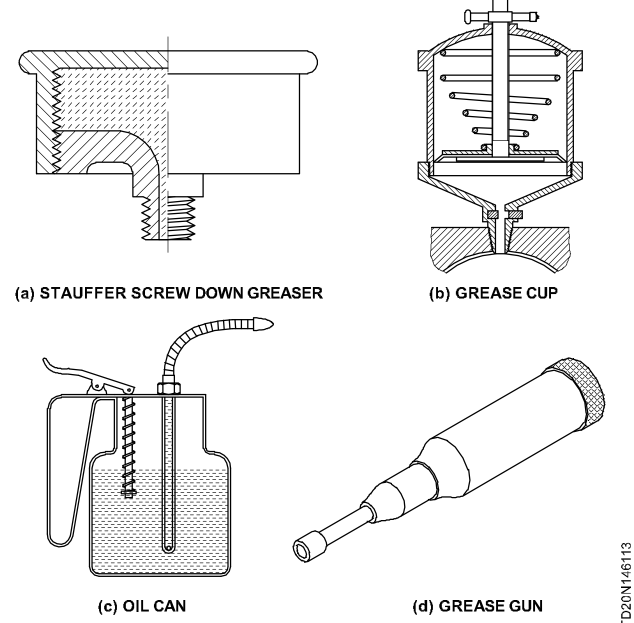
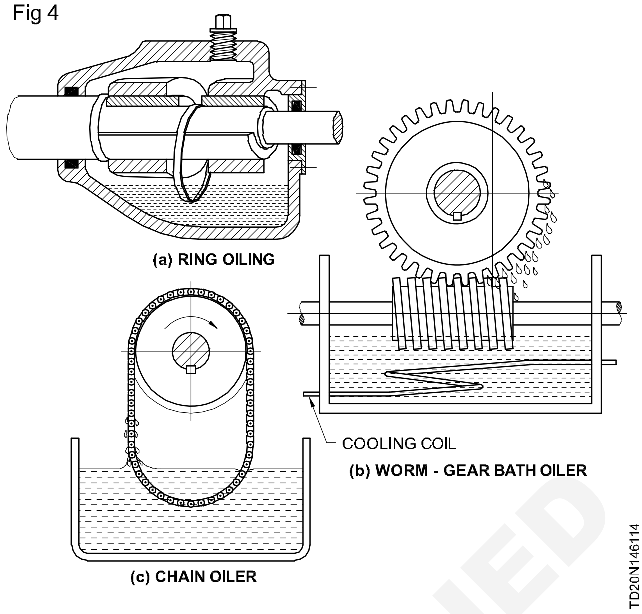


Fig 4



Splash method

In this method the shaft, or something attached to it, actually dips into the oil and a stream of lubricant is continually splashed round the parts requiring lubrication.

This method is employed for the gears and bearings inside all gear drives, the lower parts of the gears actually dipping in the oil. (Figs 4a, b and c)

A common method of employing splash lubrication is known as 'ring oiling.'

Classification of lubricants

Objectives: At the end of this lesson you shall be able to
- **state solid lubricants and their application**
- **state liquid and semi-liquid lubricants and their application**
- **state liquid and semi-liquid lubricants and their application**
- **differentiate between coolants and lubricants.**

Lubricants are classified in many ways. According to their state, lubricants are classified as:

- solid lubricants
- semi-solid or semi-liquid lubricants
- liquid lubricants.

Solid lubricants

These are useful in reducing friction where an oil film cannot be maintained because of pressure and temperature. Graphite, molybdenum disulphide, talc, wax, soap- stone, mica and french chalk are solid lubricants.

Semi-liquid or semi-solid lubricants

Greases are semi-liquid lubricants of higher viscosity than oil. Greases are employed where slow speed of heavy pressure exists. Another type of application is for high temperature components, which would not retain liquid lubricants.

Liquid lubricants

According to the nature of their origin, liquid lubricants are classified into:

- mineral oil
- animal oil
- synthetic oil.

According to the product line of Indian Oil Corporation the lubricants are classified as:

- automotive lubricating oils
- automotive special oils
- rail-road oils
- industrial lubricating oils
- metal working oils
- industrial special oils
- industrial greases
- mineral oils.

For industrial purposes the commonly used lubricants for machine tools are:

- turbine oils
- circulating and hydraulic oils (R & O Type)
- circulating and hydraulic oils (anti-wear type)
- circulating oil (anti-wear type)
- special purpose hydraulic oil (anti-wear type)
- fire-resistant hydraulic fluid

- spindle oil
- machinery oils
- textile oils
- gear oils
- straight mineral oils
- morgan bearing oils
- compressor oils.

In each type, there are different grades of viscosity and flash point. According to the suitability, lubricants are selected using the catalogue.

Example 1

Spindle oils are graded according to their viscosity and flash point.

Servospin - 2

Servospin - 5

Servospin - 12

Servospin - 22

Servospin oils are low viscosity lubricants containing anti-wear, anti-oxidant, anti-rust and anti-foam additives. These oils are recommended for lubrication of textile and machine tool spindle bearings, timing gears, positive displacement blowers, and for tracer mechanism and hydraulic systems of certain high precision machine tools.

Example 2

Gear oils are graded according to their viscosity and flash point.

Servomesh - 68

Servomesh - 150

Servomesh - 257

Servomesh - 320

Servomesh - 460

Servomesh - 680

Servomesh oils are industrial gear oils blended with lead and sulphur compounds. These oils provide resistance to deposit formation, protect metal components against rust and corrosion, separate easily from water and are non-corrosive to ferrous and non-ferrous metals.

These oils are used for plain and anti-friction bearings subjected to shock and heavy loads, and should be used in systems where the operating temperature does not exceed 90° C. These oils are not recommended for use in food processing units.

Servomesh A-90 is a litumenous product which contains sulphur-lead type and anti-wear additive. It is specially suitable for lubrication of heavily loaded low-speed open gears.

Servomesh SP 68

Servomesh SP 150

Servomesh SP 220

Servomesh SP 257

Servomesh SP 320

Servomesh SP 460

Servomesh SP 680

Servomesh SP oils are extreme pressure type industrial gear oils, which contain sulphur-phosphorous compounds and have better thermal stability and higher oxidation resistance compared to conventional lead-napthenate gear oils.

These oils have good de-emulsibility, low foaming tendency and provide rust and corrosion protection to metal surfaces. These oils are recommended for all heavy duty enclosed gear drives with circulation or splash lubrication system operating under heavy or shock load conditions up to a temperature of 110° C.

Similarities between coolant and lubricant

- A particular cutting fluid can serve as a coolant as well as lubricant at the same time. However, cooling and lubrication capabilities vary from one cutting fluid to another.
- Irrespective of its purpose (cooling or lubrication), the cutting fluid can be delivered to the cutting zone in several ways including flood cooling and MQL (minimum quantity lubrication).
- Irrespective of cooling or lubrication purpose, a cutting fluid should have high boiling point and high thermal stability.
- Both cooling and lubrication actions require good wettability so that the cutting fluid can quickly spread over the relevant surfaces.

Differences between coolant and lubricant

Coolant	Lubricant
Basic purpose of coolant is to take away generated cutting heat from the cutting zone, and thereby keep the cutting zone temperature low.	Basic purpose of lubricant is to reduce coefficient of friction between rake surface of cutter and chip, and thereby minimize rate of heat generation.
Coolant acts on the generated heat. It cannot reduce rate of heat generation.	Lubricant can reduce the rate of heat generation. It has no influence on removing heat that is already generated.
Coating on cutting tool has no role as coolant.	Coating layer on cutting tool sometimes acts as lubricant.
To act as good coolant, the cutting fluid should possess high specific heat capacity.	To act as good lubricant, the cutting fluid should possess high lubricity.
Water is considered as a good coolant owing to its higher specific heat capacity. Note that water is highly corrosive, so rarely used as cutting fluid.	Water is not good lubricant. it has poor lubrication capability or low lubricity.

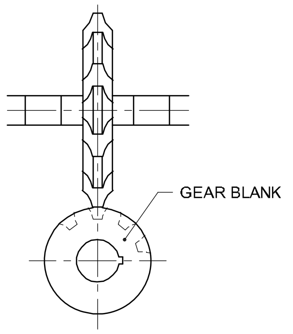
Indexing or dividing head

Objectives: At the end of this lesson you shall be able to
- **state what is indexing**
- **list the types of indexing heads**
- **state the principle of direct indexing.**

What is indexing?

It is an operation of dividing the circumference of a workpiece into equally spaced divisions (Fig 1) for milling gears, splines, squares, cutting of flutes in reamers, etc.

Fig 1



Indexing is also used to rotate the workpiece at a predetermined ratio to the table feed rate to produce cams, helical grooves etc.

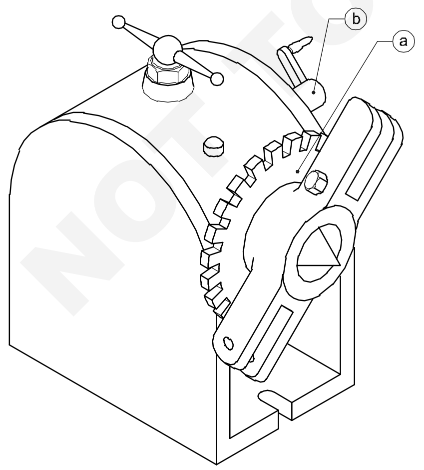
This operation is performed on a milling machine by means of an indexing attachment which is called as indexing head or the dividing head.

Types of indexing heads

The most common types of indexing heads dividing heads are

- direct indexing head (Fig 2)

Fig 2



- simple indexing head (Fig 3)
- universal indexing head. (Fig 4)

Fig 3

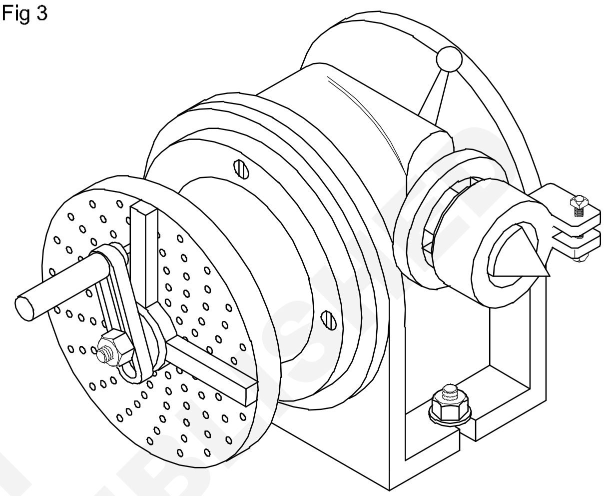
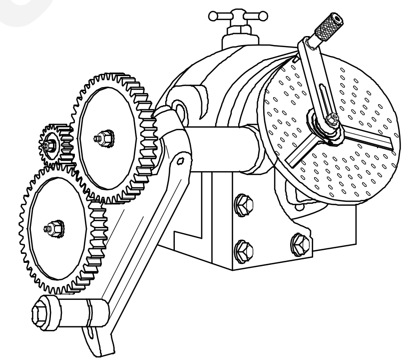


Fig 4



Direct indexing

Purpose of direct indexing

Direct indexing is a rapid method of indexing. It is used where a large number of identical pieces are indexed by very small number of divisions. Usually, this type of indexing can be performed on a direct indexing head.

Principle of direct indexing

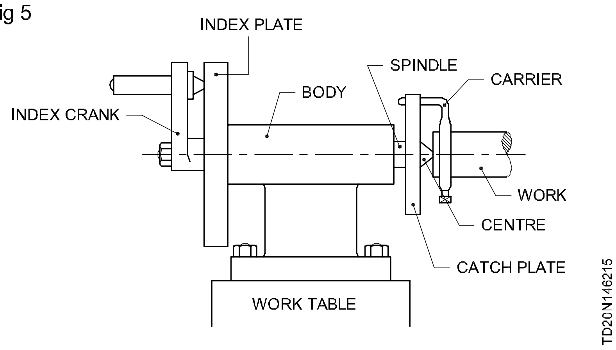
Direct indexing may be employed whenever the number of divisions required can be divisible without remainder into the number of holes or slots in the direct index plate.

Direct indexing mechanism (Fig 5)

It consists of a housing, a spindle with a driving lug and an indexing crank. The rear of the housing is fitted with a flat index plate which has a number of holes spaced around the circumference of the circles of different radii.

The number of holes varies from circle to circle. The index plate usually has three circles of holes with 24, 30, 36 holes respectively.

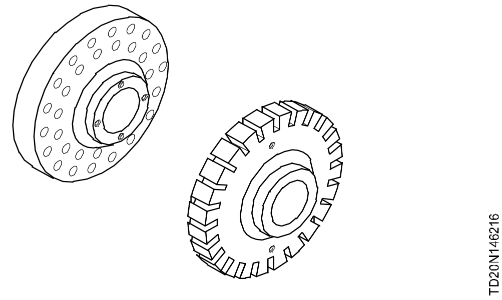
Fig 5



Another type of index plate (Fig 6) which is having a number of slots (ie.24 or 30 or 36 slots) on the periphery of the index plate, is fitted to the front end of the spindle nose. (Fig 2) The spindle is rotated by hand and locked by a pin.

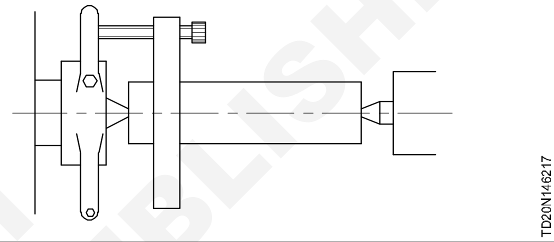
The handle of the indexing crank, which can be moved radially, is fitted with a spring-loaded index pin. The index pin engages with the holes in the indexing plate.

Fig 6



While indexing, the pin is taken out from the index plate hole. The spindle is rotated by the crank and after the required position is reached, it is again locked by the pin.The workpiece can be set up in a chuck on the indexing head spindle or between centres using the indexing head and a matching tailstock. For centre work, any suitable carrier may be used to engage the workpieces with the driving lug on the indexing head spindle. (Fig 7)

Fig 7



Plain or simple indexing

Objectives : At the end of this lesson you shall be able to
- **state the purpose of simple indexing**
- **explain the simple indexing mechanism.**

Purpose of simple indexing

Simple indexing is used to obtain a greater number of divisions that cannot be obtained by direct indexing. This operation may be performed in both simple and universal dividing heads.

Principle of simple indexing

It is carried out using 40:1 ratio of the worm and worm-wheel mechanism. One rotation of the worm, rotates the worm-wheel spindle 1/40 of a complete turn. A fractional part of 1/40 of the revolution of the worm-wheel can be performed by using the index plate.

Simple indexing mechanism (Fig 1)

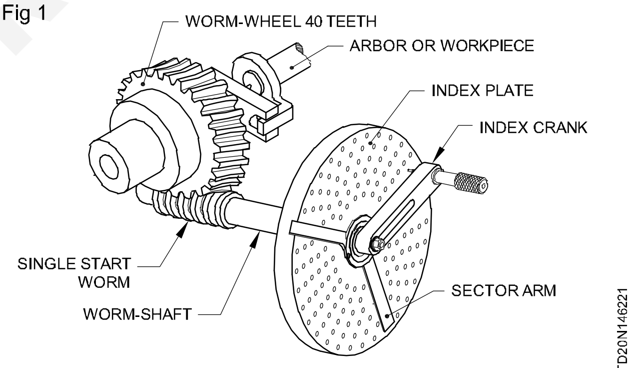
The simple indexing mechanism consists of a 40 tooth worm-wheel fastened to the spindle, a single start worm, a crank for turning the work shaft and an index plate and sector.

The worm-wheel is keyed to the spindle shaft, and meshes with the warm shaft. When the worm is given one complete turn, the worm-wheel advances one tooth or, as it has 40 teeth it will revolve 1/40 of a revolution.

In other words, 40 turns of the index crank will make the spindle revolve one complete revolution.

To facilitate indexing to fraction of a turn, index plates are used to cover practically all the numbers.

Fig 1



Index plate

The index plate is mounted behind the index crank. It is a circular plate provided with a circle of holes representing different divisions of the circle.

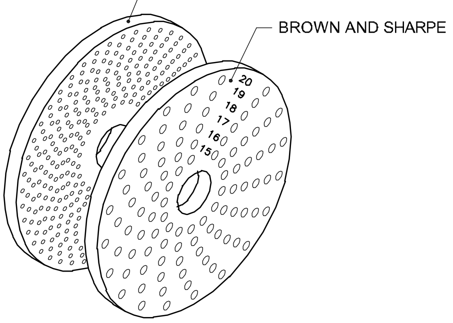
The object of these plates is to allow the worm to be moved a fractional part of a turn.

The two systems in common use are the Brown and Sharpe index system and the Cincinnati index plate system.

Brown and Sharpe index plate system (Fig 2)

The Brown and Sharpe system has three index plates and each plate has six circles of holes.

Fig 2



INDEX PLATES

Plate No.1 - 15, 16, 17, 18, 19, 20

Plate No.2 - 21, 23, 27, 29, 31, 33

Plate No.3 - 37, 39, 41, 43, 47, 49

With the three index plates, simple indexing can be used for all divisions up to 50, even numbers up to 100, except 96.

Cincinnati and parkinison index plate system

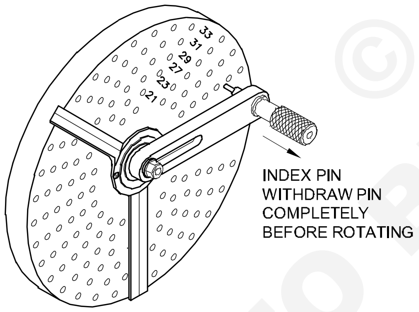
The Cincinnati and parkinison system uses one index plate with eleven circles of holes on each side. The plate is reversible and this gives twenty two cycle of holes.

First side **-** 24, 25, 28, 30, 34, 37, 38, 39, 41, 42, 43

Second side **-** 46, 47, 49, 51, 53, 54, 57, 58, 59, 62, 66

Index crank (Fig 3)

Fig 3



INDEX CRANK

The index crank is fitted to the end of the worm-shaft. The crank carries a spring-loaded index pin to engage the hole in the plate.

The crank is rotated by withdrawing the knob, turning the crank around a selected circle of holes and releasing the knob to engage the pin in the required hole.

Sector arms (Fig 4)

There are two sector arms which fit on the face of the index plate.

The arms can be set apart to cover a required number of holes between them.

Adjustment is made by slackening the lock screws, moving the arms to the desired setting and re-tightening the screws to lock the arm in position.

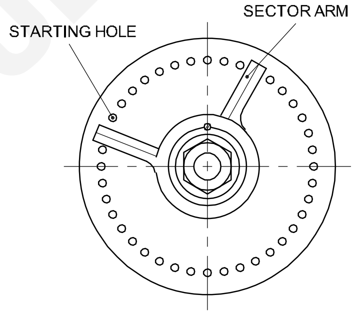
The arms eliminate the need for counting the number of holes each time the crank has to be turned by a set number of divisions.

Formula for simple indexing

$$\text{Index crank movement} = \frac{40}{N}$$

'N' is the number of turns of the crank required.

Fig 4



Angular indexing

Objectives : At the end of this lesson you shall be able to
- **state the need for angular indexing**
- **state the principle of angular indexing.**

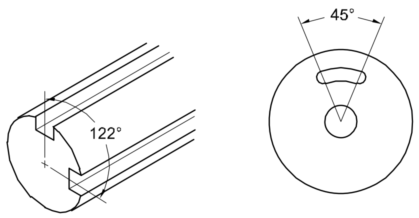
It is often necessary to index for a certain number of degrees as when machining, keyways, grooves, flats or other features located at angles to each other. (Fig 1)

Angular indexing can be carried out using a simple indexing head equipment.

Principle

Most indexing heads require 40 turns of the crank to rotate the spindle once.

Fig 1



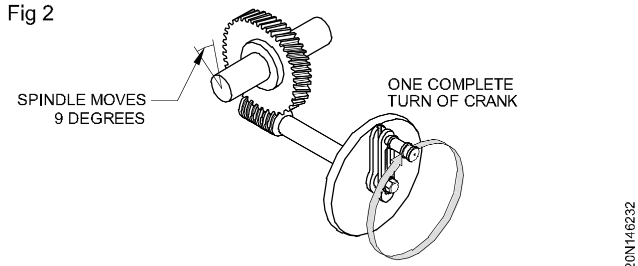
EXAMPLES OF ANGULAR INDEXING

One revolution of the spindle = 360 degrees or 40 turns of the crank.

If the crank is turned once, the spindle rotates 1/40 of a turn and 1/40 of 360° is 360/40 = 9 degrees. (Fig 2)

It follows that 1/9 turn of the crank will give a spindle movement of 1 degree.

Fig 2



PRINCIPLE OF ANGULAR INDEXING

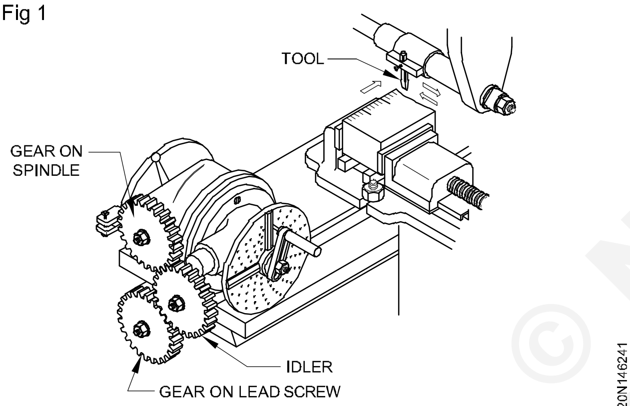
Linear Indexing

Objectives : At the end of this lesson you shall be able to
- **state the purpose of linear indexing**
- **state the principle of linear indexing, using an indexing head**
- **state the crank movement for linear indexing.**

Linear indexing

The operation of producing accurate spaces on a piece of flat stock, or that of linear graduating is easily accomplished on the horizontal milling machine Fig 1.

Fig 1



LINEAR INDEXING

In this process the work may be clamped to the table or held in a vice, depending on the shape and size of the part. Care must be taken to align the workpiece parallel to the table travel.

Principle

To produce an accurate longitudinal movement of the table, the dividing head spindle is geared to the lead screw of the milling machine.

If the dividing head spindle and the lead screw are connected with gears of 1:1 ratio, ie. With equal number of teeth and the index crank turned one revolution, the spindle and lead screw on an inch milling machine, would revolve one fortieth (1/40th) of a revolution. This rotation of the lead screw having 4 threads per inch 1/40 x 1/4 (is equal to) 1/160 = 0.00625 inch (0.15875 mm). Thus five turns of the index crank would move the table 5 x 0.00625, or 1/32 in (0.79375 mm).

Calculating of indexing

Objectives: At the end of this exercise you shall be able to
- **explain the indexing movement required for direct indexing**
- **explain the indexing movement required for simple indexing**
- **explain the indexing movement for angular indexing**
- **explain the crank movement for linear indexing.**

Direct indexing

If six slots are to be machined and the 30 circles located. No.of index crank movement

$$= \frac{\text{No.of holes in the index plate}}{\text{No.of divisions required}} = \frac{30}{6} = 5$$

The pin is inserted in every fifth hole of the 30 hole circle.

All divisions that are exactly divisible by 360° can also be obtained - 180°, 120°, 90°, 45°, 30° and 15°. This indexing can also be performed by simple indexing head by disengaging worm from worm wheel. (Fig 8)

Simple indexing

Find the number of turns of the crank required to index 12 divisions.

$$\text{Index crank movement} = \frac{40}{N} = \frac{40}{12} = 3\frac{1}{3}$$

To index the fractional part of a turn, select an index plate which has a circle of holes exactly divisible by the denominator of the fraction. In this case 3.

Assuming that a plate is chosen having a circle of holes equal to 24, then the number of holes that the index pin would have to move for 1/3 of a turn would be

$\frac{24}{1} x \frac{1}{3} = 8$ holes

Each division would require three complete turns of the index crank and 8 holes on the 24 hole circle.

Example

Find the number of turns of the index crank for indexing 43 holes.

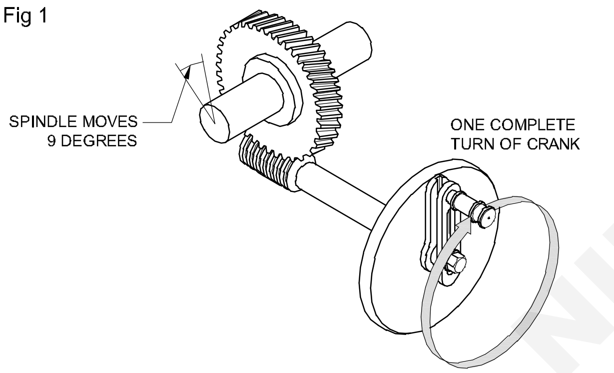
$$\text{Index crank movement} = \frac{40}{N} = \frac{40}{43}$$

In this case a partial turn of the crank for each division is only required. A circle having 43 holes is available, so that for each division, the index crank is turned 40 holes in the 43 hole circle.

Angular indexing

Index for 45°

Fig 1



PRINCIPLE OF ANGULAR INDEXING

$$T = \frac{D}{9}$$

$$= \frac{45}{9} = 5 \text{ complete turns}$$

Index for 122°

$$T = \frac{D}{9}$$

$$= \frac{122}{9} = 13\frac{5}{9} \text{ turns}$$

That is 13 complete turns and 5/9 of a turn. This is obtained by setting an index plate with a circle of holes divisible by 9 and setting the sector arms of 5/9 of a turn.

If a 18 hole circle is used

5/9 x 18 = 10 holes

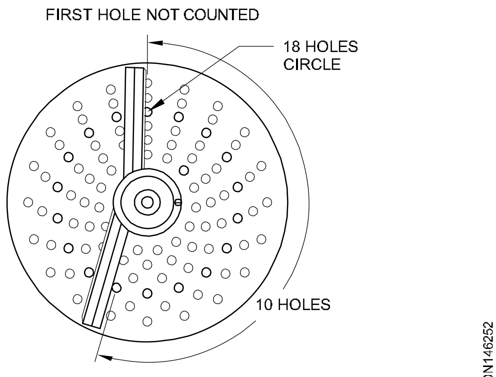
Then T = 13 turns + 10 holes of the 18 holes in the circle. (Fig 3) using brown and sharp company index plate No.1

Indexing in minutes

One revolution of the crank gives a spindle movement of 9° and to convert the degrees into minutes multiply by 60.

9° = 9 x 60 = 540 minutes.

Fig 2



SECTOR ARMS SET FOR 5/9 TURN

The number of turns of crank is found by dividing the angle to be indexed in minutes by 540.

$$\text{No.of turns} = \frac{\text{Angles to be indexed in minutes (M)}}{540}$$

Example (Figs 4 to 6)

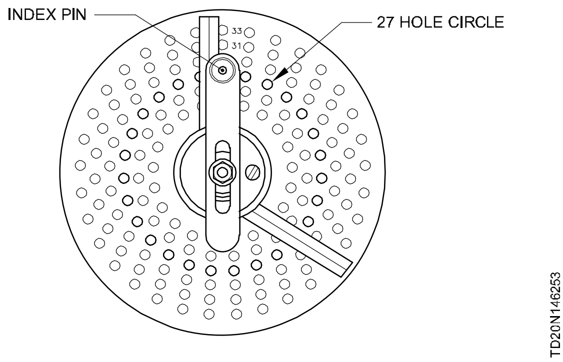
To index an angle of 34°40'

Angle in minutes = 34 x 60 + 40 = 2080

$$T = \frac{M}{540} = \frac{2080}{540} = \frac{208}{54} = \frac{104}{27} = 3\frac{23}{27}$$

3 complete turns plus 23 holes spaces in the 27 hole circle will give an index angle of 34° 40' by using brown and sharp company plate No.2

Fig 3



Indexing in seconds

One revolution of the crank gives a spindle movement of 9° and to convert the degrees into seconds multiply by 60 x 60.

9° = 9 x 60 x 60 = 32400 seconds

Index crank movement =

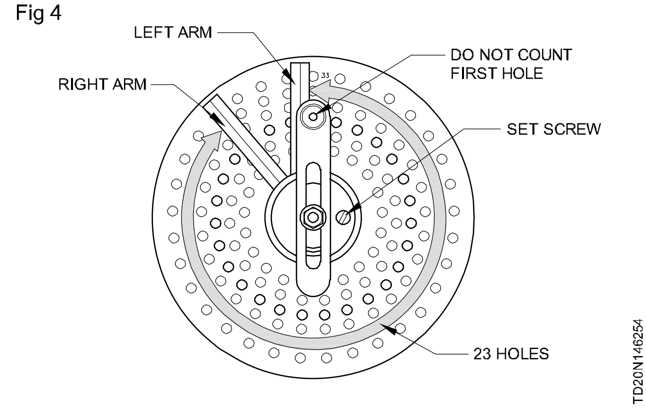
$$\frac{\text{Angular displacement of work in seconds}}{32400}$$

Linear indexing

Calculation

The formula for calculating the indexing for linear graduations in thousands of an inch is

Fig 4

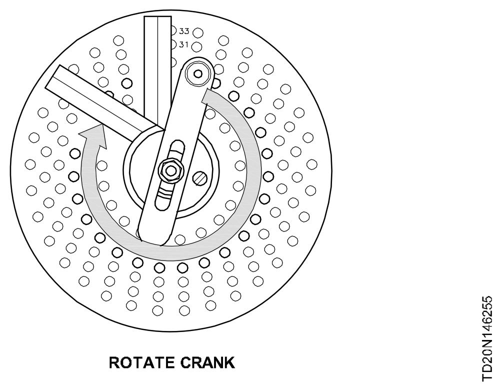


where N = spacing required in inch

T = No.of turns of index crank

If the lead screw of a metric milling machine has a pitch of 5mm, one turn of the index crank would move the table one-fortieth of 5mm, or 0.125 mm. Therefore, it would require four complete turns on the crank to move the table by 0.5 mm.

Fig 5



The formula for calculating the indexing for linear graduations in millimeters is

where N = spacing required in mm

T = No.of turns of index crank

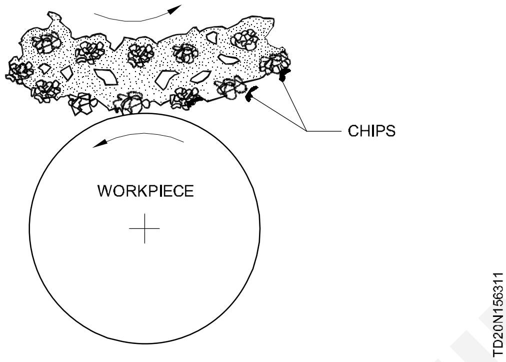
Other suitable movements may be obtained by using the appropriate hole circle and/or different change gears ratio, other than 1:1.

Introduction to grinding machine types

Objectives: At the end of this lesson you shall be able to
- **state the importance of the grinding operation**
- **list the common types of precision grinders and its purposes.**

Grinding is a metal cutting operation performed by means of a rotating abrasive wheel that act as a cutting (multipoint) tool (Fig. 1). Mostly grinding is a finishing operation with high degree of surface quality accuracy of shape and dimension. It removes comparatively less metal (0.25 to 0.50mm) in most operations.

Fig 1



Grinding has three advantages over other metal cutting methods.

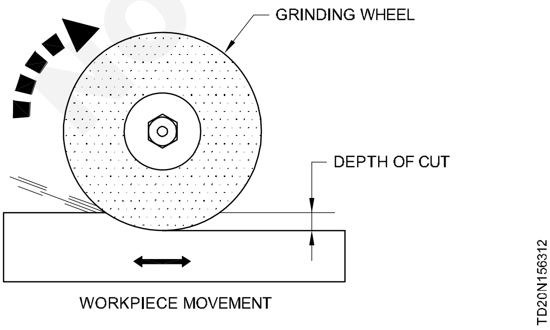
- It is the only economical method of cutting hard materials like hardened steel.
- It produces very smooth surfaces up to N4, suitable for bearing surface.
- Surface pressure is minimum in grinding. It is suitable for light work, which will spring away from the cutting tool in the other machining processes.

Types of grinding operations

There are four main grinding operations.

Surface grinding (Fig 2)

Fig 2

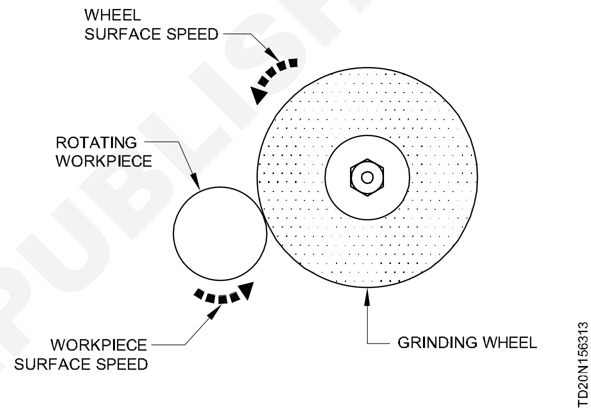


It is the operation of using precision grinding machines to produce flat or plain surfaces on workpieces. The workpiece is at a constant speed below the grinding wheel.

External cylindrical grinding (Fig 3)

It produces a straight or tapered cylindrical surface. The workpiece is rotated about its own axis between centres as it passes lengthwise across the face of a revolving grinding wheel.

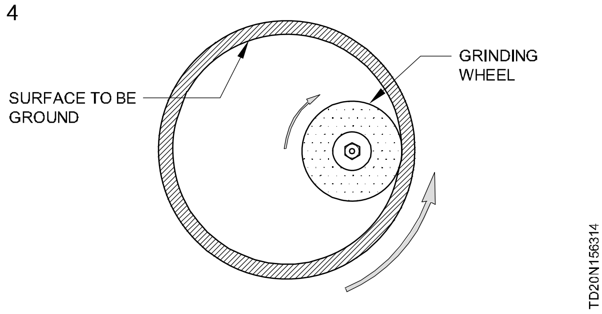
Fig 3



Internal cylindrical grinding (Fig 4)

The workpieces are held in the chuck and rotated precisely about their axis. A revolving grinding wheel, smaller than the dia. of the hole to be ground, is set against the rotation of the workpiece and traverses along the surface of the hole.

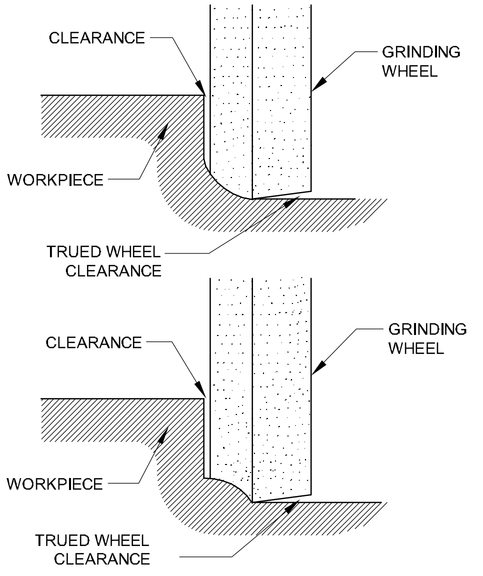
Fig 4



Form grinding (Fig 5)

It produces formed surfaces. Specially shaped grinding wheels grind the formed surfaces as is the case in grinding gear teeth, threads, splined shafts etc.

Fig 5



RADIUS WITH INCLUDED FACES

Grinding Machines

Grinding machines are precision machine tools, designed to remove metal from a workpiece to close tolerances (up to 0.0025 mm) and to produce high quality surface finish (up to N4).

There are two major groups of grinding machines.

- Off hand or rough grinders
- Precision grinders

The common types of precision grinders are:

- Surface grinders
- Cylindrical grinders
- Tool and cutter grinders.

Surface grinders

Surface grinders are used to grind flat, parallel surfaces or stepped surfaces. The surface produced by a surface grinder is more economical and more accurate than the surface obtained by filling or scraping.

Cylindrical grinder

Cylindrical grinders are used to grind external and internal cylindrical surfaces. The cylindrical surfaces produced may be plain, tapered or stepped.

Tool and cutter grinder

A tool and cutter grinder is mainly used to sharpen single point cutting tools, milling cutters etc. It also can be used as a surface and cylindrical grinder along with some attachments.

Surface grinding machine

Objectives: At the end of this lesson you shall be able to
- **state the uses of a surface grinder**
- **state the four common types of surface grinders**
- **state the main parts and their functions of a surface grinder**
- **state the specification of surface grinder.**

Surface grinding machine

It is a precision grinding machine to produce flat surface, Parallel surface or stepped surface. It is a move economical and more practical method of accurately finishing flat surface than filing and scraping. The abrasive wheel is used as a cutting tool. It has more number of cutting edges than single point cutting tool, hence removes metal fastly and accurately.

Types of surface grinders

There are four types of surface grinders

Horizontal spindle reciprocating table (Fig 1)

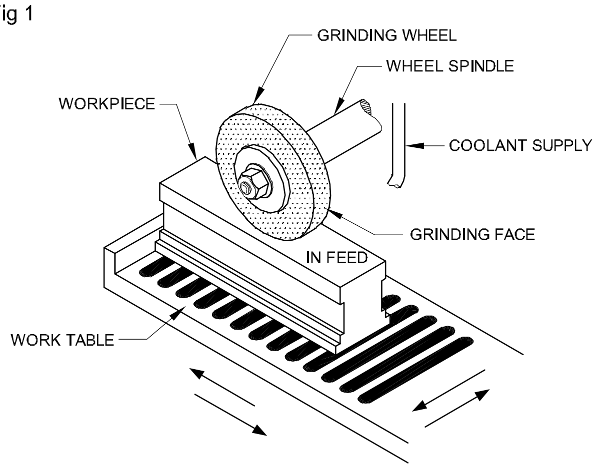
Horizontal spindle rotary table (Fig 2)

Vertical spindle reciprocating table. (Fig 3)

Vertical spindle rotary table. (Fig 4)

A horizontal spindle type surface grinders widely used, grinding is normally done on the periphery of the wheel. The area of contact is small and the speed is uniform over the grinding surface.

Fig 1



Horizontal spindle reciprocating table surface grinder

Main parts (Fig 5)

- Base
- Saddle
- Table
- Wheel head

Fig 2

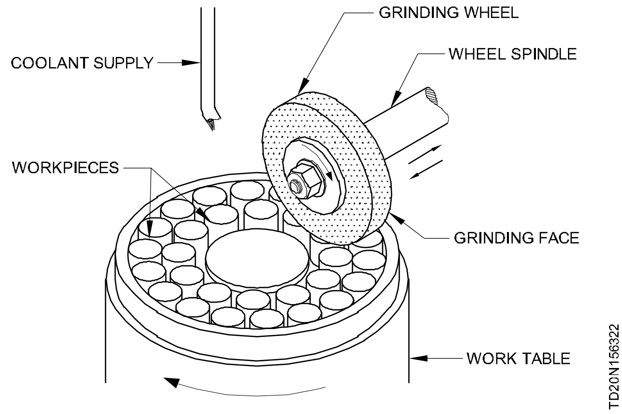


Fig 3

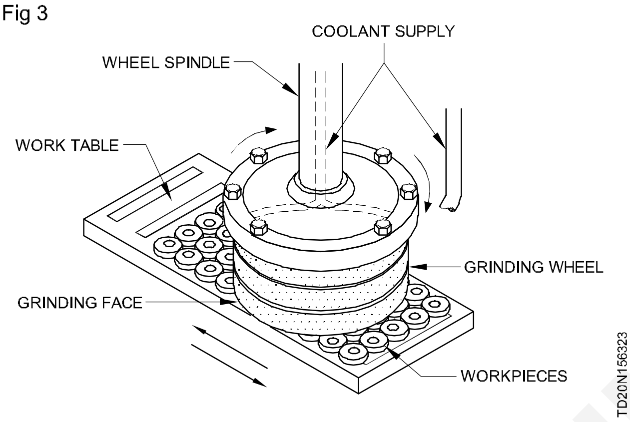
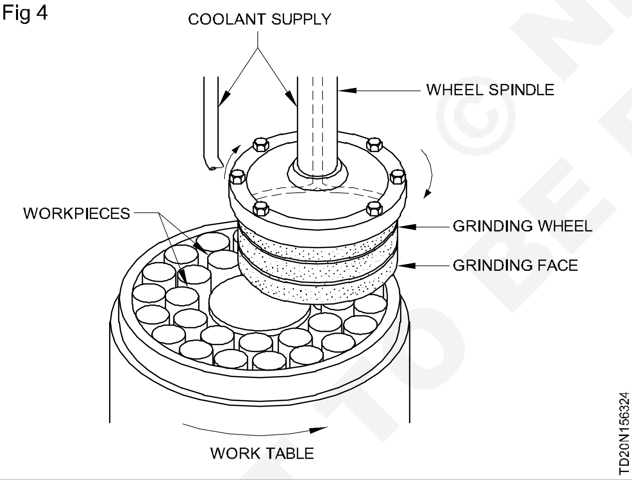


Fig 4



Base

It is a rigid rectangular box contains the driving mechanism (hydraulic device tank and motor). It has a column at the back for supporting the wheel head on the top of the base provide precision guide ways for moving saddle.

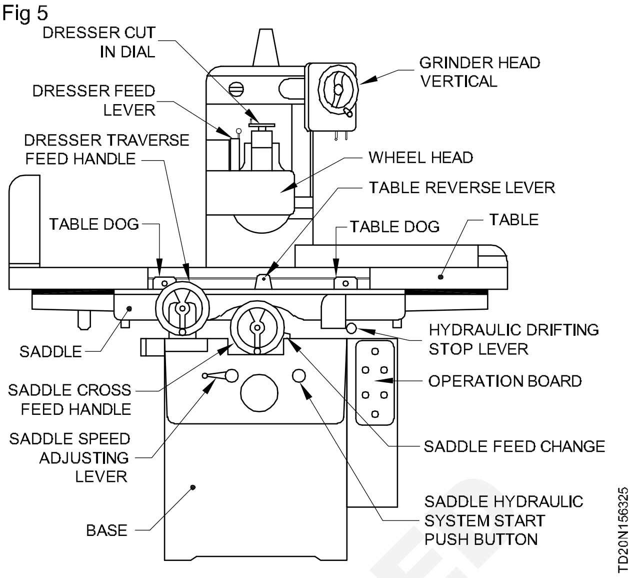
Saddle

It is a frame. It contains the table in its cross wise movement. It is used to give cross feed to the work. It can be moved by hand or auto feed.

Table

It is fitted on the saddle. It is reciprocates along the guide ways to provide the longitudinal feed to the work. The surface is accurately machined and T-slots are provided

Fig 5



for clamping of workpieces directly on the table or for clamping magnetic chuck and grinding fixtures. It is moved by hand or auto feed.

Wheel head

It is mounted on the column secured to the base. It can be moved vertically up and down to by rotating a hand wheel accommodate work piece of different height and set the wheel for depth of cut. The wheel rotates at a constant wheel speed. (1500 rpm)

Some surface grinding machines the dressing unit mounted top of the wheel head and slide to dressing the wheel with help of rotating micrometer collar handle. Dress the wheel 0.015mm to 0.025mm to giving a feed.

Specification of surface grinder

- Maximum dia. of the wheel that can be held on the spindle.
- Working surface of table (length & wide)
- Vertical traverse of wheel head (height)
- Table cross and traverse (movement) feed rate.
- Speed of wheel.
- HP of wheel head motor.
- Types of drive (Incase of hydraulic drive hydraulic pump HP)
- Floor space required.
- Net weight of the machine.

Eg. Praga, 175mm wheel 400mm stroke, hydraulic surface grinder. Model 451.

Detailed specifications and dimensions of a surface grinder are furnished by the manufacturer in the operator's instructional manual. Refer to one such manual of your section with the help of your instructor.

Cylindrical grinders

Objectives: At the end of this lesson you shall be able to
- **state the purpose of a cylindrical grinder**
- **name the 4 types of cylindrical grinders**
- **state the parts and functions of a plain centre type cylindrical grinder**
- **specify a cylindrical grinder.**

Cylindrical grinders are used to grind the external or internal surfaces of a cylindrical workpiece.

By cylindrical grinding the diameter of a workpiece can be maintained to a close tolerance (up to 0.0025 mm), and a high quality surface finish can be obtained (up to N4).

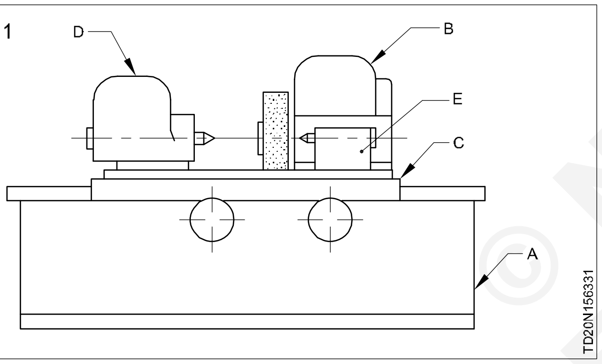
The four types of cylindrical grinders are:

- external cylindrical grinders
- internal cylindrical grinders
- universal cylindrical grinders
- centreless grinders.

Plain centre type cylindrical grinder (Fig 1)

It is mainly intended to produce plain, stepped or tapered external cylindrical surfaces.

Fig 1



Parts

The main parts of this type of a cylindrical grinder are the:

a base

b wheel head

c table

d headstock

e foot-stock.

Functions

Base (A) is made out of cast iron. It is heavy and provides rigidity to the machine. The top surface is machined to form guideways to the table.

The wheel head (B) is mounted on the cross-slide. It moves perpendicular to give depth of cut.

The table (C) is mounted on the bed-ways. It reciprocates past the wheel. It can be swivelled to grind taper. Trip dogs are provided to control reciprocation.

The headstock (D) is mounted on the table at the left end. It has a motor with 2 or 4 speed steps to drive the work. A dead centre is mounted in the spindle of this head to support the workpiece between centres.

The foot-stock (E) is mounted on the table at the right hand side. It can be moved and locked at any place along the table length to hold different lengths of work. The spindle is spring-loaded and carries a dead centre to support the work.

The spring tension provides even, stiff support.

Specification

A cylindrical centre type grinder is specified by the maximum diameter and length of the workpiece that can be accommodated between centres. The diameter of the workpiece should not be more than half of the capacity given.

Tool and cutter grinder

Objectives: At the end of this lesson you shall be able to
- **state the purpose of a tool and cutter grinder**
- **state the parts and functions of a tool and cutter grinder**
- **specify a tool and cutter grinder.**

In a machine shop, much of the machining operations is done by single point tools or multi-point tools called milling cutters. The cutting tools become blunt due to constant use, and need re-sharpening for continued production.

Such re-sharpening is done in tool rooms, where a tool and cutter grinder is used for this purpose.

A universal tool and cutter grinder is used to re-sharpen reamers, taps, single point tool and milling cutters, dies and punches.

Parts and construction of a tool and cutter grinder (Fig 1)

Base

It supports all the other parts of the machine. The body and frame are in one unit. The driving and feed mechanism are fitted in the body.

Saddle

It is on the top of the body; it carries the table and traverses crosswise to the table movement.

Table

It is on the saddle. It reciprocates and can be swivelled to the required angle.

Column

It is on the back of the machine and it carries the wheel head which moves up and down for the depth of cut.

Wheel head

It has two grinding wheels on both ends of the spindle and can be swivelled to the required angle (360°).

Work head

It is a separate part and is fitted on the table for cylindrical works. It is driven by a separate motor fitted with the work head. It moves along the table and can be swivelled to any angle (180°).

A tailstock is provided for supporting the mandrel between centres.

Specification of tool and cutter grinder

1 Maximum dia. of the wheel that can be held in the spindle.

2 Maximum height of the job that can be ground.

3 Maximum length of the job that can be ground.

4 Maximum breadth of the job that can be ground.

5 Type of drive

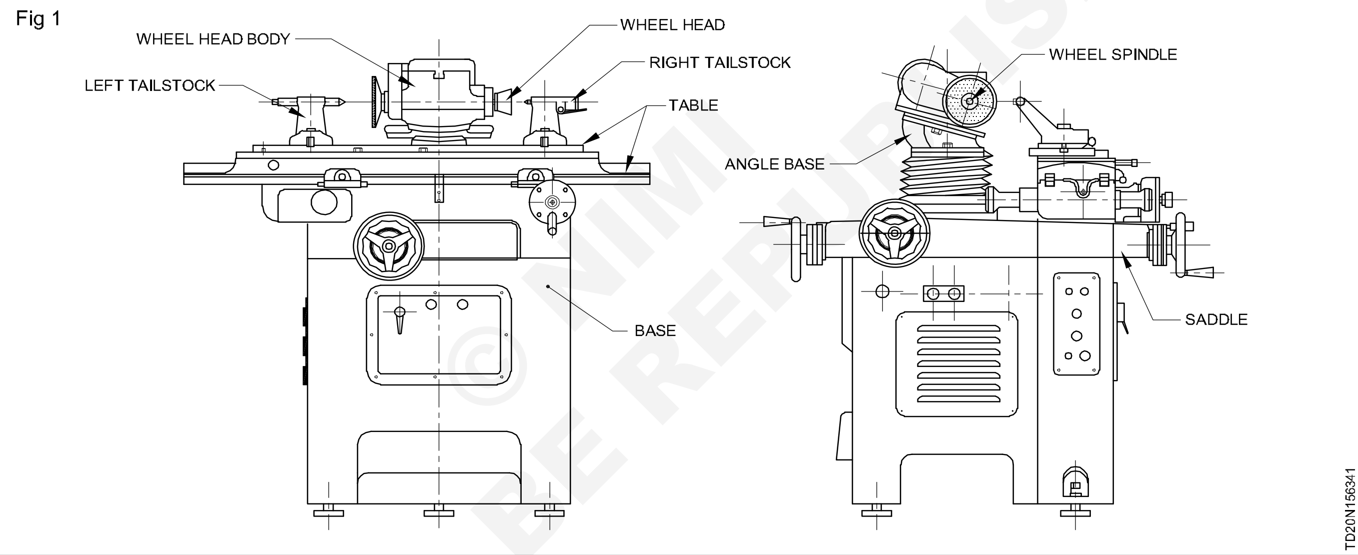
- hydraulic
- electrical.

6 Number of attachments

Attachments

A tool and cutter grinding machine is also used as a surface grinding, cylindrical grinding and internal grinding machine with the help of certain attachments.

Fig 1



Safety to be observed while working on grinding machine

Objective: At the end of this lesson you shall be able to
- **state the precautions to be observed while working on grinding.**

Safety precautions

All grinding machines have parts that move at high speed.

The machines are fitted with guards to protect the operator form injury and to make operation of the machine as safe as possible

Despite this, accidents still happen.

These accidents are usually caused by :

- Ignorance
- Thoughtlessness
- Carelessness

Lack of consideration for the safety of others.

These accidents can be prevented by thinking before doing.

Various unsafe conditions and procedure are mentioned throughout this manual. Learn to recognize them and gain a clear understanding of what should be done in each case.

The safety precautions to be taken when using grinding machines may be divided into four areas.

- General
- Machine
- Personal

General safety precautions

- Key the work area around machines free of obstacles and waste a material.

- Immediately clean up any oil, grease or coolant spilled on the floor.
- Place cleaning cloths and waste materials in the proper containers after use
- Store hand tools and accessories away from machines after use.
- Do not handle workpieces which may be hot as a result of grinding operations.
- Use the correct hand tool for the job in hand.
- Seek assistance when handling heavy machine accessories, grinding wheels or workpieces.
- Learn the location of the nearest fire alarm.
- Learn where fire extinguishers are located and how to use them,
- Stop, look and think before starting any new operation.
- Ensure lighting is adequate.
- Always be courteous, considerate an obliging to others.

Machine safety precautions

- Operation machines only when you are authorized by your instructor to do so.
- Follow your instructor's directions carefully.
- Keep your fingers away from the moving parts of the machine
- Do not start a machine unless all machine guards are correctly fitted.
- Make sure the workpiece is securely fitted to the work table before starting a grinding operation.
- Do not handle the surface of the workpiece while the machine is operating.
- Do not use your hand to stop movement of any part of the machine.
- Use a brush, not your hand, to clean ground material from the workpiece and machine.
- Keep the machine free of tools, accessories and parts not being used at the time.
- When setting the work table for automatic traverse, allow the wheel to over travel the workpiece in each direction.
- Do not clamp hardend workpiece too tightly in the jaws of a vice.
- Whenever possible, use a coolant during a grinding operation.
- If a grit exhaust system is fitted to the machine, use it all times during grinding.
- Stop the machine before cleaning or oiling it or before making any adjustments to the accessories or to the workpiece.
- Dot not leave a machine while it is still running.
- Do not touch or lean on a machine someone else is using.
- Do not divert eh attention of someone else using a machine.

Personal safety precautions

- Wear goggles at all times when using a grinding machine.
- Report any injuries, however slight, to your instructor or supervisor
- Wear close - fitting clothes.
- Avoid wearing a tie and long sleeves. If necessary, truck your tie carefully inside your shirt or keep in inside of outer clothing, buttoned or zipped up high and roll up your sleeves.
- If your hair is long, wear a protective head covering and make sure your hair is completely enclosed in side it.
- Do not wear a watch, rings or other loose ornaments.
- Do not wear gloves.
- Wipe your hands clean before operating a machine, adjusting accessories or handing a workpiece.

Grinding wheels, types, application

Objectives: At the end of this exercise you shall be able to
- **explain the type and names of different shapes of grinding wheels**
- **state the application of each type of grinding wheel.**

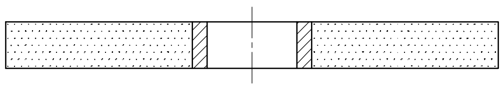
Grinding wheels are made in different shapes and sizes for grinding different jobs and for use in different machines. The size may differ in diameter, face width and bore dia.

The following are the standard shapes of grinding wheels.

Straight wheel: Type 1 (Fig 1)

This type of wheels is used on cylindrical, surface and centreless grinders for grinding cylindrical and flat surfaces. Sometimes this type of wheel is used on rough grinders for off hand grinding.

Fig 1



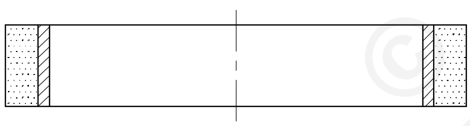
STRAIGHT

TD20N156711

Cylinder: Type 2 (Fig 2)

This type of wheel is used on both horizontal and vertical spindle surface grinders for the surface grinding operations.

Fig 2



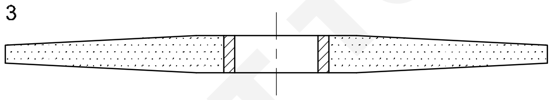
CYLINDER

TD20N156712

Tapered (both sides) Type 4 (Fig 3)

It is mainly used for rough grinding. The tapered sides reduce the chance of breaking.

Fig 3



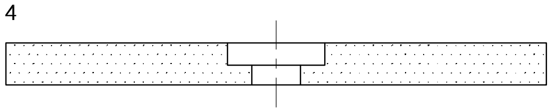
TAPERED TWO SIDES

TD20N156713

Recessed one side: Type 5 (Fig 4)

It is used for cylindrical, surface and centreless grinding. The recess provides clearance for the flange.

Fig 4



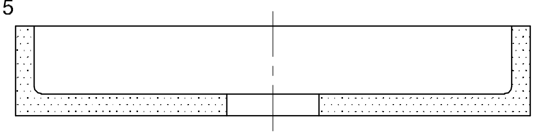
RECESSED ONE SIDE

TD20N156714

Straight cup: Type 6 (Fig 5)

It is used on surface grinders and on tool and cutter grinders to grind flat surfaces.

Fig 5



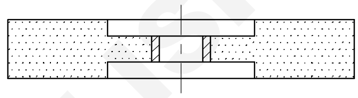
STRAIGHT CUP

TD20N156715

Recessed both sides: Type 7 (Fig 6)

Used on cylindrical, surface and centreless grinders. The recesses provide clearance for both flanges.

Fig 6



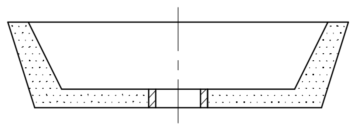
RECESSED TWO SIDES

TD20N156716

Flaring cup: Type 11 (Fig 7)

It is used on tool and cutter grinders mainly to sharpen milling cutters and reamers.

Fig 7



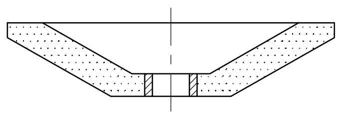
FLARING CUP

TD20N156717

Dish: Type 12 (Fig 8)

Used on tool and cutter grinders to sharpen milling cutters with narrow slots like formed relieved cutters, hobs etc.

Fig 8



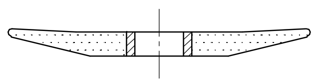
DISC WHEEL

TD20N156718

Saucer: Type 13 (Fig 9)

It is used for sharpening circular and handsaws. It is also used for gashing milling cutter teeth.

Fig 9



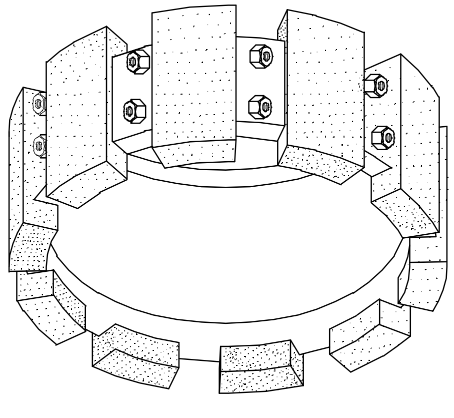
SAUCER WHEEL

TD20N156719

Segmented wheels (Fig 10)

This type of wheels is formed by holding segments of abrasives using a metal holder. This is mainly used on a vertical spindle surface grinder.

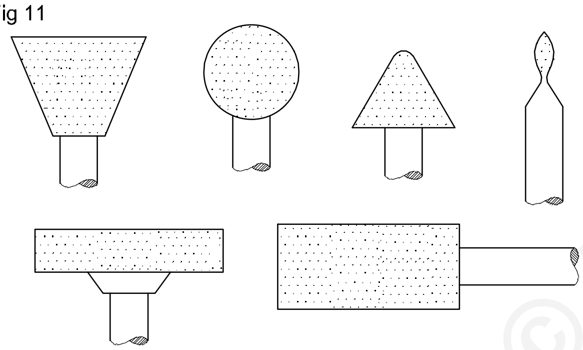
Fig 10



Mounted wheels (Fig 11)

These are wheels with less than 50 mm dia. formed on a steel shank to various shapes. Mounted wheels are mainly used for die grinding, deburring and for finishing operations. Used on pneumatic or electric grinders.

Fig 11



Types of wheel faces

To do different operations different types of wheel faces are produced by manufacturers. (Fig 12)

Grinding wheel specification

A grinding wheel is specified by its marking, shape, outside dia. bore dia. thickness etc. (Fig 13a)

A recessed wheel is specified with all the above given particulars plus the dia. of the recess and the depth of the recess. (Fig 13b)

Fig 12

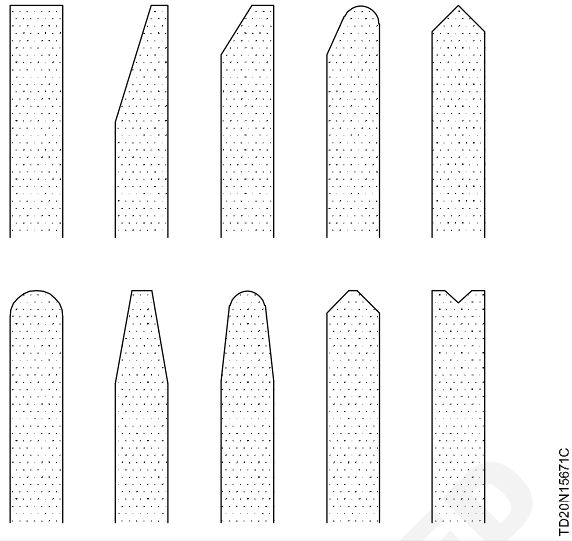
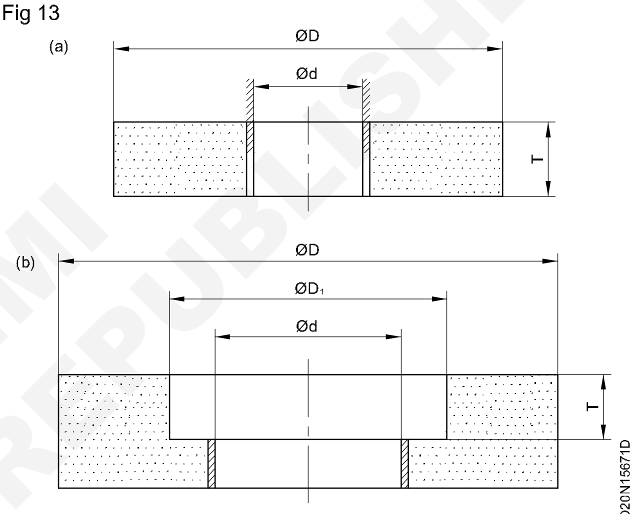


Fig 13



Grinding wheel marking system

eg. A 56 K 5 V 75

1st Position 'A' denotes 'Abrasives' ie., Aluminium Oxide

2nd Position '56' denotes 'Grit size' ie., Medium

3rd Position 'K' denotes 'Grade' ie., Medium

4th Position '5' denotes 'Structure' ie., Dense

5th Position 'V' denotes 'Bond' ie., Vitrified

6th Position '75' denotes manufacturer's Code if any

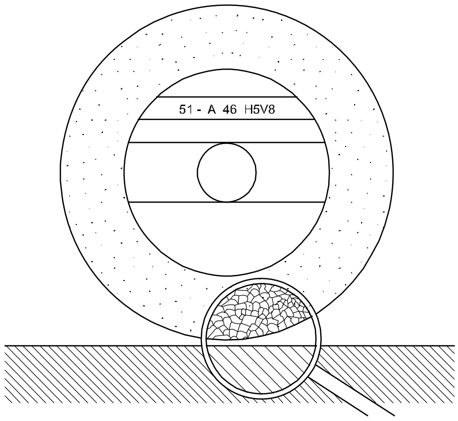
Marking system and selection of grinding wheel

Objectives: At the end of this lesson you shall be able to
- **explain marking system of grinding wheel**
- **state the selection of grinding wheel.**

Introduction

Standard wheel marking specifying all the important wheel characteristics. The marking system comprises of seven symbols which are arranged in the following order. (Fig 1)

Fig 1



Example

Marking system

51 - A46 H5V8

Specification of grinding wheels

A grinding wheel is specified by the:

standard wheel marking

outer diameter of the wheel

bore diameter of the wheel

thickness of the wheel

type (shape) of the wheel

Example

32 A46 H8V 15

250 x 20 x 32

Straight wheel

Table 1

Position 0	Position 1	Position 2	Position 3	Position 4	Position 5	Position 6
Manufacture's symbol for abrasive (optional)	Type of abrasive	Grain size	Grade	Structure (optional)	Type of bond (optional)	Manufacture's own mark
51	A	46	H	5	V	8

Selection of grinding wheels for internal grinding

Objectives: At the end of this lesson you shall be able to
- **list the factors considered while selecting grinding**
- **list three types of applications of internal grinding.**

Internal grinding are widely used in bearings and automobile industry.

Such as bore grinding of inner rings, bore grinding of gears, track grinding of outer rings and steering nuts etc.

In internal grinding, the grinding of internal surfaces the confirmity between the wheel and workpiece, is substantially higher.

For selection of grinding wheels IS: 1249 - 1958 provides recommendations on the general considerations, which guides the selection of grinding wheels for different applications.

The main factors considered while selecting grinding wheel are:

- Material to be ground and its hardness
- Stock removal
- Severity of operation
- Surface finish
- Area of grinding contact
- Wheel speed
- Wheel and dry grinding

- Machine condition
- Work speed

The illustration and working application of internal grinding wheels one shown in Fig 1 and 2.

Fig 1

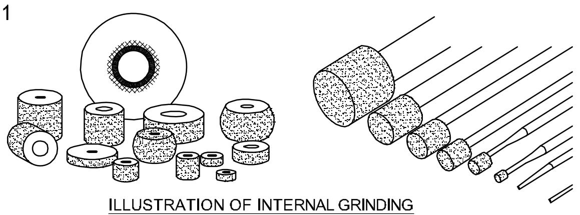
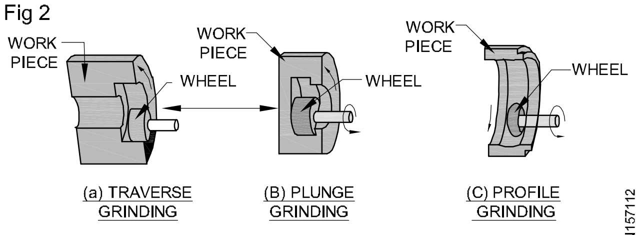


ILLUSTRATION OF INTERNAL GRINDING WHEEL & MOUNTED WHEELS FOR SMALL DIA

Fig 2



APPLICATION OF INTERNAL GRINDING WHEEL

Selection and use of grinding wheels for different applications are usually given in grinding wheel manufactures catalogue

Standard size of wheels are available.

Selection of grinding wheel for tool and cutter grinder

Objective: At the end of this lesson you shall be able to
- **state the factors which affect the selection of grinding wheel.**

For grinding a job the right grinding wheel is to be selected. The selection of a grinding wheel will depend on the following factors.

- **Factors affecting the selection of abrasive**
  - a Materials of high tensile strength, viz. alloy steel, hard bronzes steel and wrought iron.
    - Aluminium oxide
  - b Hardened tool steel high speed steel drills cutters and for cool and precision grinding.
    - White Aluminium oxide
  - c Materials of low tensile strength, viz., Aluminium, copper, cast-iron, stone ad marble.
    - Silicon carbide.
  - d Tungsten carbide tipped tools
    - Green silicon carbide.
- **Factors affecting the selection of grit.**
  - a Great amount of stock to remove - Course grain.
  - b Soft and tough materials - Course grain.
  - c Fine finish - Fine grain.
- **Factors affecting the selection of grade**
  - a Hard materials - Soft wheel.
  - b Soft materials - Hard wheel.
  - c Great area of contact - Soft wheel.
  - d Low wheel surface speed - Hard wheel.
  - e Unstable and shaky foundation of grinding machine. - Hard wheel.
  - f Off-hand grinding - Hard wheel.
- **Factors affecting the selection of structure**
  - a Soft and tough material - Open structure.
  - b Fine finish - dense structure.
  - c cylindrical and tool grinding - medium structure
  - d External grinding - dense structure.
- **Factors affecting the selection of bond.**
  - a General purpose and maximum cutting efficiency - vitrified
  - b Wheels of very large diameter and wheels required quickly to special order. - silicate
  - c very thin wheel - shellac or rubber.
  - d very high finish where rapid cutting is not important - shellac or rubber

Procedure for mounting of grinding wheels, balancing of grindng wheels

Objectives: At the end of this lesson you shall be able to
- **state what is a wheel balancing**
- **name the method of balancing**
- **state the procedure of balancing.**

Wheel balancing

Wheel balancing is an action of bringing the grinding wheel to rotate concentric to its axis and the weight and density of wheel are uniform throughout its circumference.

Before testing the balancing of the wheel it is true

Necessary of wheel balancing

- A god surface finish is possible to the work surface.
- Prevents wheel vibration and breakage
- Prevents chatter marks on the work surface
- Improves the dimensional accuracy of the work.
- Considerably increases the life of grinding wheel.
- Prevents the damage of the spindle / bearings.

Small wheel normally to not require any balancing, but larger diameters of the wheel important is the balancing . Similar equipment used to balance the wheels of motor cars.

Method of balancing

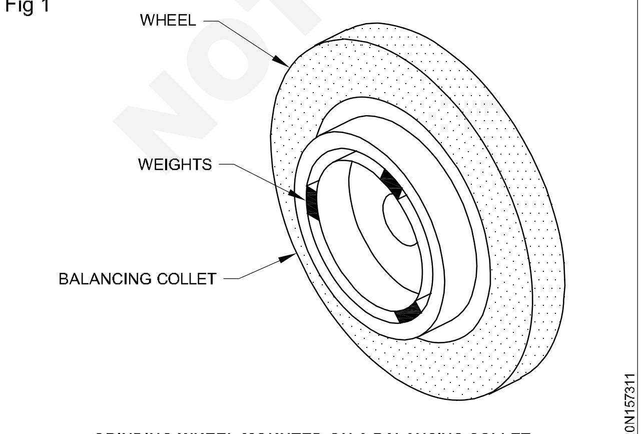
- Static balancing
- Dynamic balancing

Static balancing means that when the wheel is cantered in balancing mandrel and placed on a balancing stand.

Dynamic balancing means that when the wheel can be while it is running on the machine. For getting still better result.

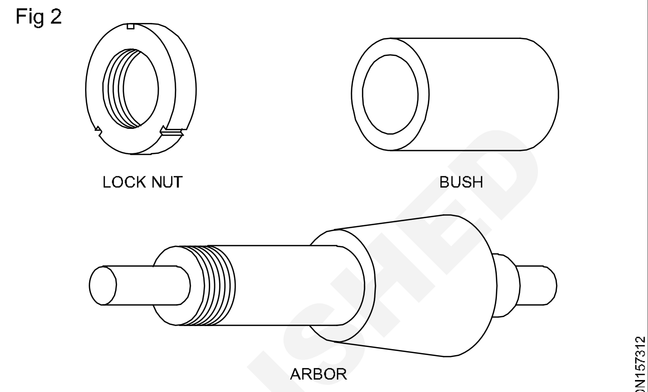
Wheel mounted on a balancing collet (Fig 1 & 2)

Fig 1



GRINDING WHEEL MOUNTED ON A BALANCING COLLET

Fig 2

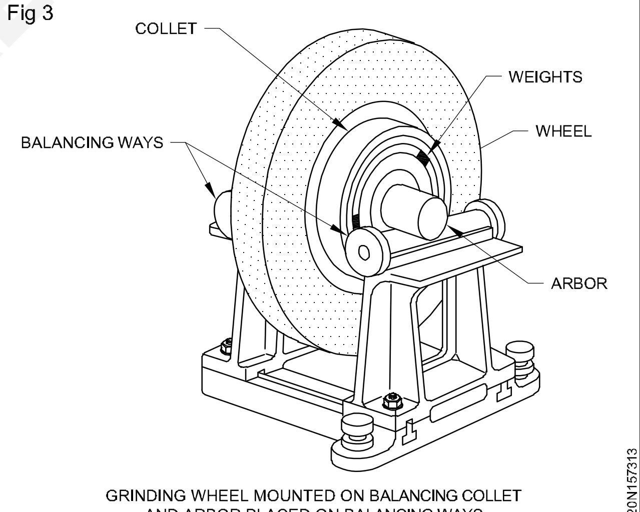


BALANCING ARBOR COMPONENTS

Large grinding wheels must be mounted on a balancing collet and balanced before being fitted to the grinding machine. The collect remains fitted to the wheel during use.

To balance a wheel proceed as follows (Fig 3)

Fig 3



GRINDING WHEEL MOUNTED ON BALANCING COLLET AND ARBOR PLACED ON BALANCING WAYS

- Mount the wheel on a balancing collect.
- Fit a balancing arbour to the collet.
- Remove the adjustable weight from the collet.
- Place the arbour collect and wheel assembly on a pair of balancing ways in a position near the centre of the ways.
- Ensure that the ways are perfectly horizontal.
- Use are accurate spirit level for this if none is mounted on the ways.

- Allow the wheel to roll slowly on the ways by a very gentle push until it comes to rest. In this rest position the heavy spot of the wheel will be on the lower part of the wheel.
- Do not push so hard that the wheel rolls off the ways.
- Make a chalk mark on the wheel at the point opposite to the heavy spot. This will be the uppermost position of the wheel when it is at rest.
- Mount the balancing weights on either side of the chalk mark.
- Test as before by allowing the wheel to rotate slowly on the balancing ways, each time moving the weights a little further back from the chalk mark after the wheel comes to rest until a perfect balance is obtained.
- The wheel is balanced when it gently comes to a stop with no tendency to roll back regardless of which portion of the wheel was the lowest at the start of the roll.
- Fix the weights in the balance position by tightening the weight set screws.
- Remove the arbor from the collect.
- The balanced wheel with its collet assembly is now ready for mounting on the grinding machine spindle.

Wheels in use should be re-balanced at intervals since the balance may change with wear of the wheel.

Caution:

If a wheel is re-dressed during service, it must be re-balanced after the dressing operation.

Grinding wheel dressing

Objectives: At the end of this lesson you shall be able to
- **state three important purposes of dressing a grinding wheel**
- **differentiate between dressing and truing**
- **state the types of wheel dressers and their uses.**

Dressing is an operation to change the cutting action of a wheel or to recondition the grinding surface. Grinding wheels should be dressed and trued regularly to improve the followings:

Work production

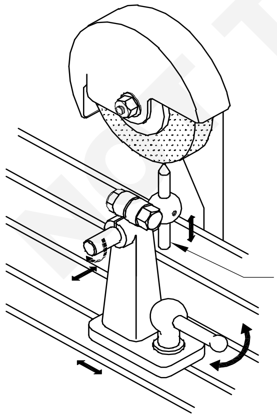
Wheel performance

Grinding economy

Dressing (Fig 1)

Dressing refers to the removing of clogs and blunt abrasive grains from the surface of the grinding wheel. Dressing grains exposes the cutting edges which restore the correct cutting action of the wheel. Dressing is done on a glazed or loaded wheel to recondition it.

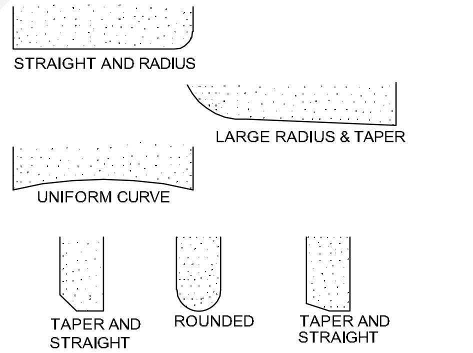
Fig 1



Truing: Truing refers to the shaping of the wheel to make it run concentric with the axis. When a new grinding wheel is mounted, it must be trued before use to remove the run out.

Truing is done on the wheel which is out of shape due to long use. Sometimes a wheel is also trued to change the shape of the grinding wheel face for a specific grinding operation like form grinding. (Fig 2)

Fig 2



In most of the cases both dressing and truing are done at the same time.

There are three basic types of wheel dressers. They are:

- diamond • steel • abrasive

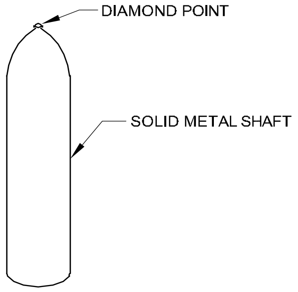
Diamond dressers (Fig 3)

A diamond dressing tool has a hard diamond mounted in a metal shank. The shank is fitted in a tool holder for location on the grinding machine to perform dressing.

Diamond dressers are most effective for dressing precision grinding wheels.

A low feed of a diamond dresser can glaze the wheel. They are specified by their weight in carats. Usually 0.5 carat to 1 carat diamonds are used for dressing upto 200 mm dia. wheels.

Fig 3



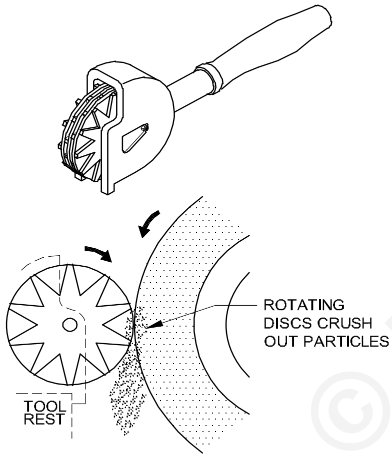
DIAMOND TOOL

Steel dressing tools (Fig 4)

Steel dressers for dressing grinding wheel have rotary cutting surfaces made from hard steel.

They are held in place against the grinding wheel by hand and moved across the face of the grinding wheel to do the dressing. A tool rest or other rigid support must be used during this operation.

Fig 4



CUTTING ACTION OF A STEEL DRESSING TOOL

The main types of steel dresser are:

Star and disc dresser (used for coarse grained wheel)

Corrugated disc dresser (used for smooth finish)

Lock disc dresser (used for medium roughing wheel)

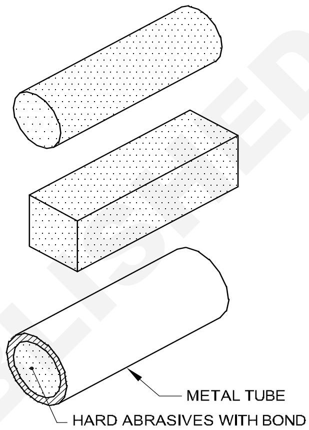
Solid cylinder dresser (used for instead of a diamond dress)

Abrasive dressers

When only light dressing is required abrasive sticks can be used. There are abrasive materials made in the form of square or round sticks or put in metal tubes for convenient handling.

This type of dressers is more convenient in tool and cutter grinders where frequent dressing and truing is necessary

Fig 5



Glazing and loading, their effects, causes and remedies

Objectives: At the end of this lesson you shall be able to
- **differentiate between glazing and loading of a grinding wheel**
- **state the effects of a glazed and loaded wheel while grinding**
- **state the causes and remedies for glazing**
- **state the causes and remedies for loading.**

Glazing

When the surface of a grinding wheel develops a smooth and shining appearance, it is said to be glazed. (Fig 1) This indicates the abrasive particles on the wheel face are not sharp. These are worned down to bond level.

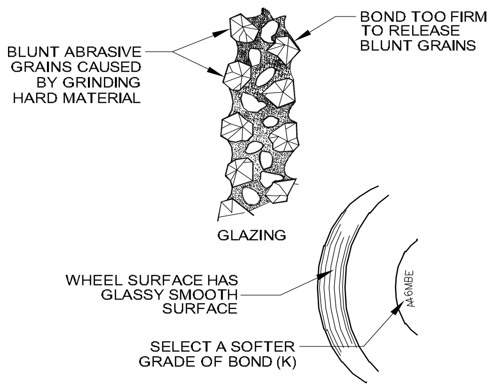
Loading

When soft materials like aluminium, copper, lead, etc. are ground the metal particles get clogged between the abrasive particles. This condition is called loading. (Fig 2)

The effects of a glazed or a loaded grinding wheel are almost the same. They are:

- excessive cutting pressure between wheel and work
- more heat generation

Fig 1



GLAZED WHEEL

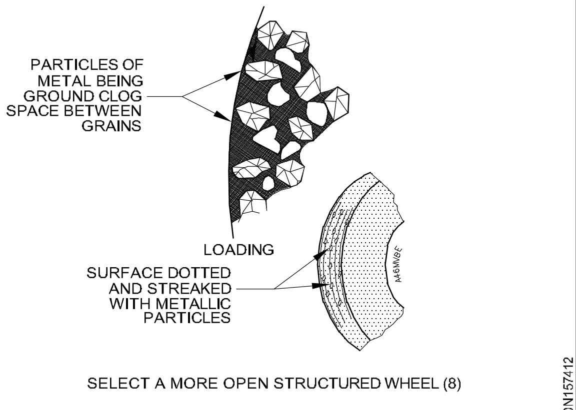
- burning of the ground surface
- poor surface finish
- inaccuracies in the size and shape of the workpiece
- wheel breakage (sometimes)

A dull or glazed wheel should be dressed for the following reasons

To reduce heat generated between the work surfaces and the grinding wheel.

To reduce the strain on the grinding wheel and the machine

Fig 2



LOADED WHEEL

To improve the surface finish and accuracy of the work

To increase the rate of metal removal

Cause and remedies of glazing

Wrong selection of glazing

Wrong selection of grinding wheels means hard grade wheel in place of soft wheel and fine grain size in place of medium grain size.

Select a grinding wheel of the right grade and size.

High wheel speed: Set the wheel to the recommended speed.

Feed too fine: Set the feed rate correctly.

Dirty coolant: Change the coolant

A glazed or a loaded grinding wheel can be reused after removing the glazed or loaded particles from the grinding wheel face.

Roughness values symbols and surface quality

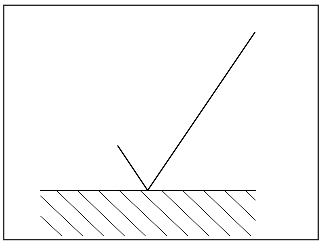
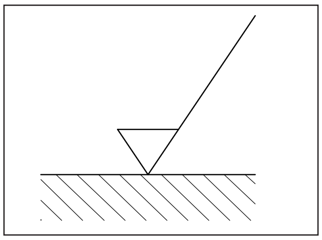
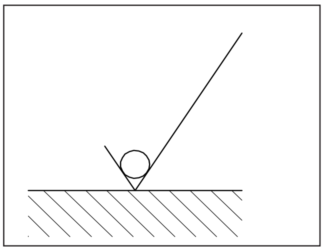
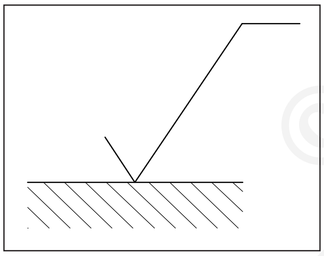
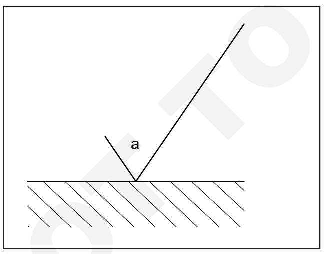
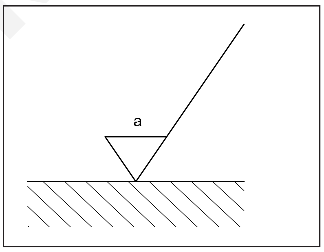
Objectives: At the end of this lesson you shall be able to
- **state the values of surface roughness**
- **state the indication of surface roughness.**

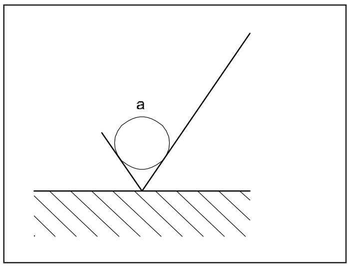
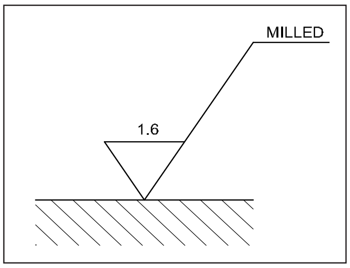
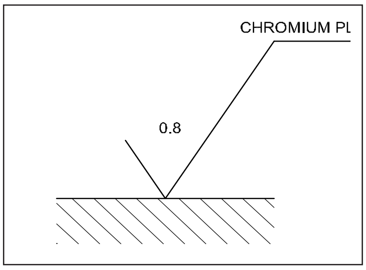
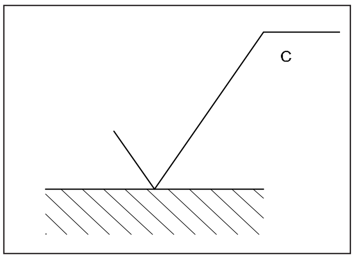
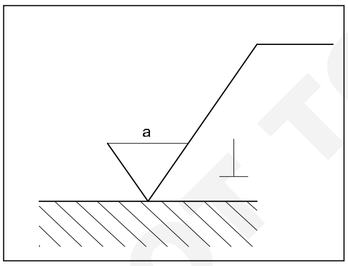
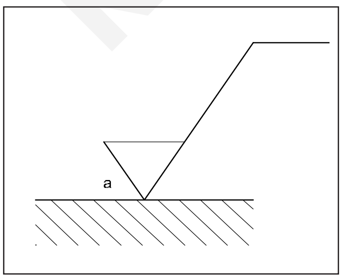
Letter symbols for roughness

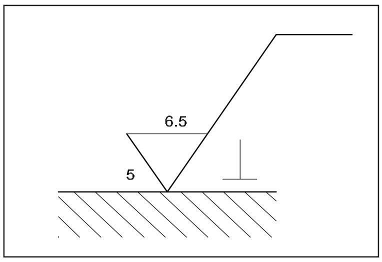
Indication of surface roughness values.

S. 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	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 indications

Symbol	Symbol stands for
	**Surface symbol indication** 1 The basic symbol consists of two legs of unequal length inclined at approximately 60°.
	A bar is added to the basic symbol, if the material removal by machining is required.
	A circle is added to the basic symbol, if the material removal is not permitted.
	A line is added to the larger leg, if some special characteristics have to be indicated.
	Surface roughness obtained by any production method.
	Surface roughness obtained by without removal of any material.

Symbol	Symbol stands for
	Surface roughness obtained by removal of material by machining.
	Indicating the production method.
	Indicating the surface treatment or coating unless otherwise stated, the numerical value of the roughness, applies to the surface roughness after treatment coating.
	Indicating the sampling length.
	Direction of lay, surface pattern by the production method employed.
	Indicating of allowance in mm.

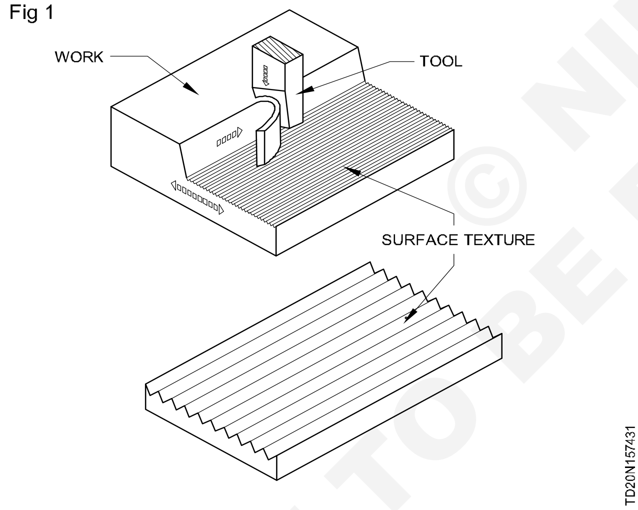
Symbol	Symbol stands for
	Surface texture.

Surface quality

Objectives: At the end of this lesson you shall be able to
- **state the meaning of roughness value**
- **state the parameters on which surface quality depends**
- **state the method of measuring roughness**
- **define the symbols for surface roughness.**

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)

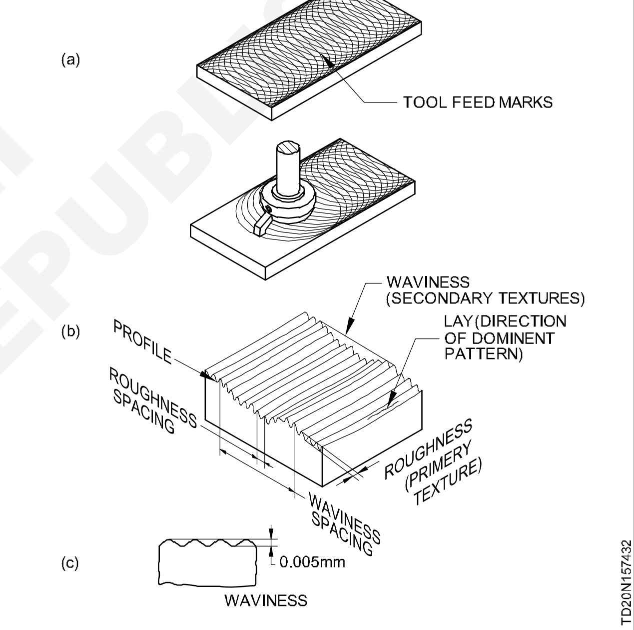
The irregularities in the surface texture result from the inherent action of the production process. These will include traverse feed marks and irregularities within them. (Fig 2a)

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.

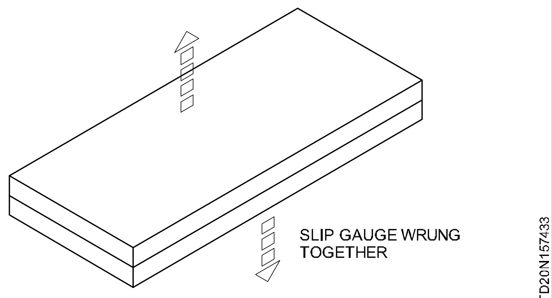
Fig 2



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 wrong together.

Fig 3

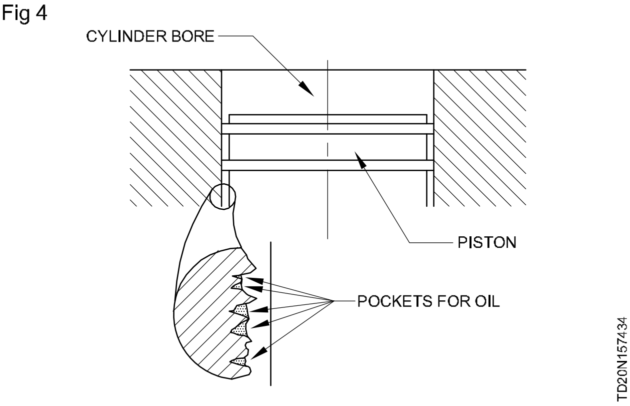


The cylinder bore of an engine (Fig 4) may require a certain degree of roughness for assisting the lubrication needed for the movement of the piston.

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

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 4



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 5

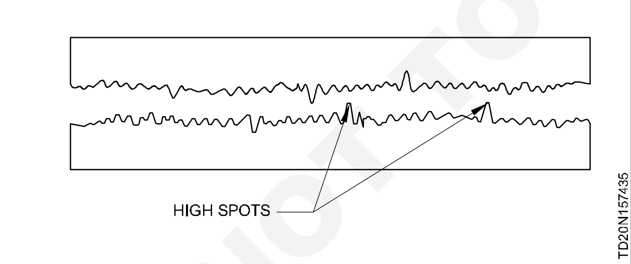
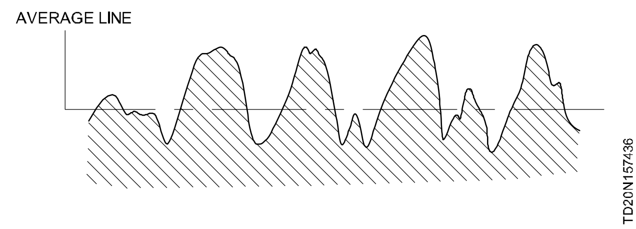


Fig 6



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

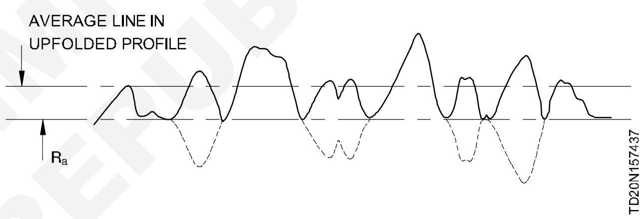
A new mean line (Fig 7) is then calculated for the curve obtained after folding the bottom half of the original profile.

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}.

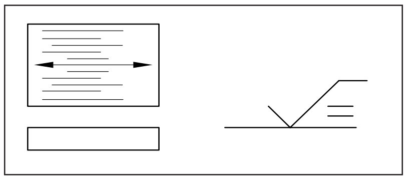
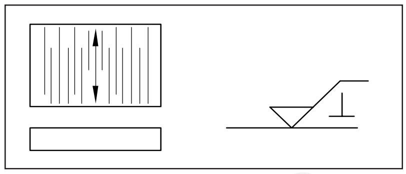
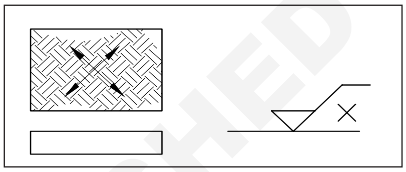
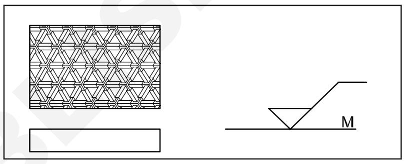
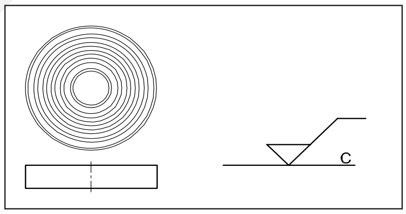
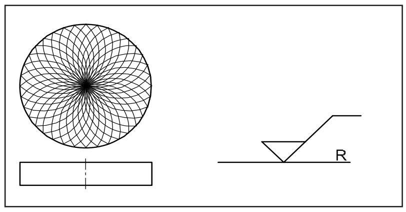
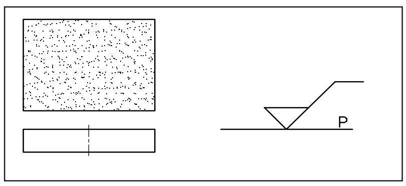
When only one 'Ra' value is specified, it represents the maximum permissible value of surface roughness.

Fig 7



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.	

Grinding wheel - Abrasives - Bond - Grade - Grit- structure

Objectives: At the end of this lesson you shall be able to
- **explain grinding wheel**
- **state the types of abrasives**
- **explain grain and grade**
- **state the types of bonds.**

Grinding wheel

A grinding wheel is multipoint cutting tool made up of many hard particles known as abrasive. The abrasive grains are held together with adhesive substance known as bond.

The wheel may consist of one piece or segments of abrasive blocks built up to a solid wheel.

Abrasives

An abrasives are hard, tough, sharp edge and resistance to fracture used for cutting other materials.

There are two types of abrasives

Natural abrasive

Artificial abrasive

Natural abrasives

The natural abrasives are emery, corundum, sandstone or solid quartz and diamond.

Emery is a natural aluminium oxide. It contains aluminium iron oxide and other impurities.

Corundum also natural aluminium oxide it contains upto 95% and remainder is impurities.

Sand stone or quartz is one of the natural abrasive stones from which grind stones are shaped.

Diamond is less than quality of gem are crushed to produce abrasive grains for making grinding wheels and lapping compound.

Artificial abrasive

The artificial abrasive are silicon carbide and aluminium oxide.

Silicon carbide (SiC)

Silicon carbide abrasives are manufactured from silica sand. Silicon carbide is hard and brittle. It is used for grinding low tensile material like brass, copper, grey cast iron, aluminium. Silicon carbide is represented by letter 'S'.

Aluminium oxide (Al_2O_3)

This is manufactured from mineral bauxite. Aluminium oxide is tough and less brittle. It is used for grinding high tensile strength material like steels. Carbon steels, malleable iron, high speed steel and wrought iron. Aluminium oxide is represented letter 'A'.

The abrasives are selected depending upon the material being ground.

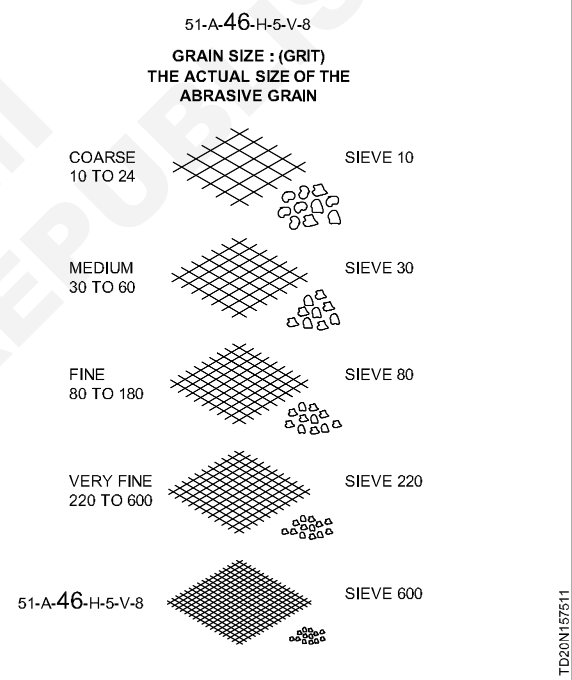
'Green' silicon carbide is used for very hard materials with low tensile strength such as cemented carbides.

'Brown' aluminium oxide is used for general purpose grinding of tough materials.

Aluminium oxide is used for grinding die steels.

Grain size (Grit size) (Fig 1)

Fig 1

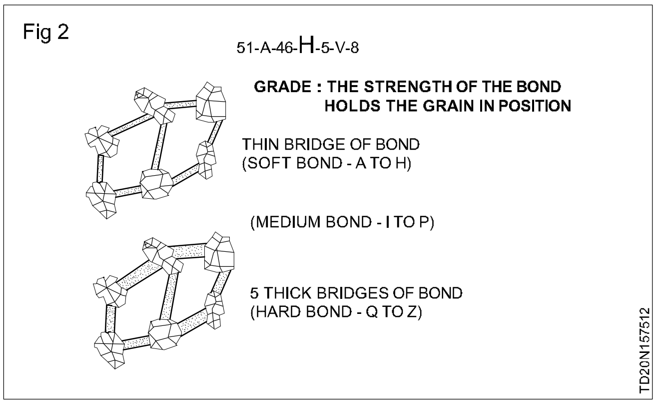


The grit or grain size refer to the actual size of the abrasive particles. The grains size is denoted by a number. The sieve used to size the grain.

The larger the grit size number the finer the grit and the smaller the grit size number the large the grit.

Grade (Fig 2)

Grade indicates the strength of the bond and, therefore, the 'hardness' of the wheel. In a hard wheel the bond is strong and it securely anothers the grit in place, and therefore, reduce the rate of wear. In a soft wheel, the bond is weak and the grit is easily detached resulting in a high rate of wear.



Alphate letters are used to indicate the grade of wheel

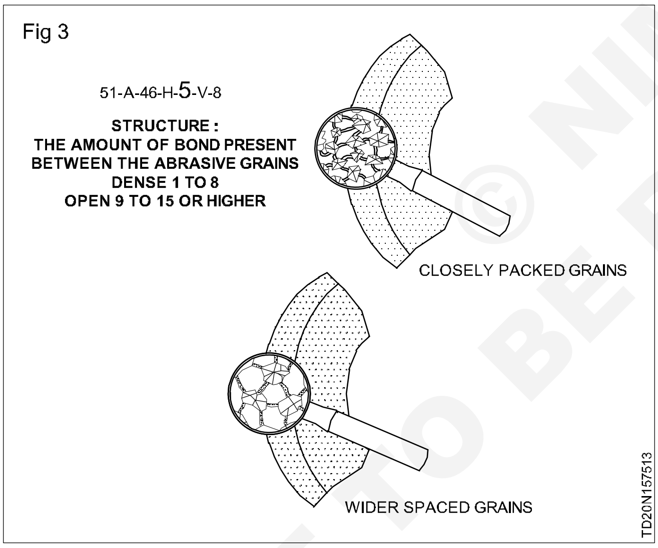
A to H - Soft

I to P - Medium

Q to Z - Hard

Structure (Fig 3)

This indicates the amount of bond present between the individual abrasive grains, and the closeness of the individual grains to each other. An open structured wheel will cut more freely. That is, it will remove more metal in a given time and produce less heat. It will not produce such a good finish as a close structured wheel.



The structure is specified by number from 1 to 15.1 is indicating dense structure 15 indicates most wider structure. 1 to 8 dense and 9 to 15 and above indicates open structure.

Open structure wheel is used for grinding soft tough and ductile metal and used rough grinding.

A closed structure wheel is used for finish grinding of hard and brittle metal.

Bond

The bond is the substance which, when mixed with abrasive grains, holds them together, enabling the mixture to be shaped to the form of the wheel, and after suitable treatment to take on the form of the wheel and the necessary mechanical strength for its work. The degree of hardness possessed by the bond is called the 'grade' of the wheel, and this indicates the ability of the bond to hold of bonding materials used for making wheels.

Types of bonds and their uses

Vetrified bond (V)

This is the most widely used bond. It has high porosity and strength which makes this type of wheel suitable for high rate of stock removal. It is not adversely affected by water, acid, oils at ordinary temperature conditions.

Silicate bond (S)

Silicate wheels have a milder action and cut with less harshness than vitrified wheels. For this reason they are suitable for grinding fine edge tools, cutlery etc. This bond is used for making large dia grinding wheels.

Shellac bond (E)

This is used for heavy duty, large diameter wheels where a fine finish is required. For example, the grinding of mill rolls.

Rubber bond (R)

This is used where a small degree of flexibility is required on the wheel as in the cutting of the cutting off wheels.

Resinoid bond (B)

This is used for high speed wheels. Such wheels are used in foundries for dressing castings. Resinoid bond wheels are also used for cutting off parts. They are strong enough to withstand considerable abuse.

Oxychloride bond (O)

The abrasive grains are mixed with magnesium chloride and magnesium oxide. This bond is used for making disc shaped wheels.

The bond ensures a cool cutting action so best for dry grinding operation. This bond is used for making segmented wheels.

AutoCAD of its screen components

Objectives: At the end of this lesson you shall be able to
- **brief the methods of starting and existing AutoCAD drawing**
- **explain the various screen components in AutoCAD**
- **describe the setting of unit of areas in AutoCAD**
- **state what s Auto CAD and its uses.**

The computer aided drafting

Starting around the mid 1970s, as computer aided design systems began to provide more capability than just an ability to reproduce manual drafting with electronic drafting, the cost benefit for companies to switch to CAD became apparent. The benefit of CAD systems over manual drafting are the capabilities one often takes for granted from computer systems today; automated generation of Bill of Material, auto layout in integrated circuits, interference checking, and many others. Eventually CAD provided the designer with the ability to perform engineering calculations. During this transition, calculations were still performed either by hand or by those individuals that could run computer programs. CAD was a revolutionary change in the engineering industry, where draftsmen, designers and engineering roles begin to merge. It did not eliminate departments, as much as it merged departments and empowered draftsman, designers and engineers. CAD is just another example of the pervasive effect computers were beginning to have on industry.

Current computer-aided design software packages range from 2D vector-based drafting systems to 3D solid and surface modelers. Modern CAD packages can also frequently allow rotations in three dimensions, allowing viewing of a designed object from any desired angle, even from the inside looking out. Some CAD software is capable of dynamic mathematical modeling, in which case it may be marketed as CADD.

CAD is used in the design of tools and machinery and in the drafting and design of all types of buildings, from small residential types (houses) to the largest commercial and industrial structures (hospitals and factories).

CAD is mainly used for detailed engineering of 3D models and/or 2D drawings of physical components, but it is also used throughout the engineering process from conceptual design and layout of products, through strength and dynamic analysis of assemblies to definition of manufacturing methods of components. It can also be used to design objects. Furthermore many CAD applications now offer advanced rendering and animation capabilities so engineers can better visualize their product designs. CAD has become an especially important technology within the scope of computer-aided technologies, with benefits such as lower product development costs and a greatly shortened design cycle. CAD enables designers to layout and develop work on screen, print it out and save it for future editing, saving time on their drawings

The AutoCAD

AutoCAD is a Computer Aided Design (CAD) program used by just about every Engineering and Design office in the world. Although there are alternative CAD packages, AutoCAD is by far the most widely used system. Autodesk's AutoCAD is the industry leader in CAD package used by Civil Engineers, Architects, Mechanical & Electrical Engineers, Aeronautical Engineers plus many other disciplines.There have been several versions of AutoCAD over the years, with each new version introducing new and more powerful features than its predecessor.

Advantages Of Using Computer Aided Design (CAD) Over Manual Drafting

Draw to Scale

The advantages of CAD include: the ability to producing very accurate designs; Drawings can be created in 2D or 3D and rotated; other computer programmes can be linked to the design software. With manual drafting, you must determine the scale of a view before you start drawing. This scale compares the size of the actual object to the size of the model drawn on paper. With CAD, you first decide what units of measurement you will use, and then draw your model at 1:1 scale.

Lay Out The Drawing

When you draft manually, you first select a sheet, which usually includes a pre-printed border and title block. Then you determine the location for views' plans, elevations, sections, and details. Finally, you start to draw. With CAD, you first draw your design, or model, in a working environment called model space. You can then create a layout for that model in an environment called paper space. A layout represents a drawing sheet. It typically contains a border, title block, dimensions, general notes, and one or more views of the model displayed in layout viewports. Layout viewports are areas, similar to picture frames or windows, through which you can see your model. You scale the views in viewports by zooming in or out.

Organize Drawing Information

With manual drafting, you can separate information onto individual transparent overlays. For example, a building plan might contain separate overlays for its structural, electrical, and plumbing components. With CAD, layers are equivalent to transparent overlays. As with overlays, you can display, edit, and print layers separately or in combination. You can name layers to help track content, and lock layers so they can't be altered. Assigning settings such as color, linetype, or lineweight to layers helps you comply with industry standards. You can also use layers to organize drawing objects for plotting. Assigning a plot style to a layer makes all the objects drawn on that layer plot in a similar manner.

Establish Drafting Standards

Manual drafting requires meticulous accuracy in drawing linetypes, lineweights, text, dimensions, and more. Standards must be established in the beginning and applied consistently. With CAD, you can ensure conformity to industry or company standards by creating styles that you can apply consistently. You can create styles for text, dimensions, and linetypes. A text style, for example, establishes font and format characteristics such as height, width, and slant. You can save styles, layers, layouts, title block and border information, and some command settings in drawing template files. Using drawing templates helps you quickly start new drawings that conform to standards.

Draw Efficiently

With manual drafting, you use drawing tools that include pencils, scales, compasses, parallel rules, templates, and erasers. Repetitive drawing and editing tasks must be done manually. In CAD, you can choose from a variety of drawing tools that create lines, circles, spline curves, and more. You can easily move, copy, offset, rotate, and mirror objects. You can also copy objects between open drawings.

Draw Accurately

With manual drafting, you must draw objects carefully to ensure correct size and alignment. Objects drawn to scale must be manually verified and dimensioned. With CAD, you can use several methods to obtain exact dimensions. The simplest method is to locate points by snapping to an interval on a rectangular grid. Another method is to specify exact coordinates. Coordinates specify a drawing location by indicating a point along an X and Y axis or a distance and angle from another point. With object snaps, you can snap to locations on existing objects, such as an endpoint of an arc, the midpoint of a line, or the center point of a circle. With polar tracking, you can snap to previously set angles and specify distances along those angles.

View Your Drawing

With manual drafting, the size and resolution of your drawing is fixed. With CAD, the size and resolution of your drawing can be changed as needed. To do detailed work, you can increase display size by zooming in. You can zoom out to display more of the drawing. To move to another section of a drawing, you pan the drawing without changing magnification. You can zoom and pan to create the best working conditions. This can be invaluable when working on large and detailed drawings, such as this health spa plan.

Create Dimensions and Text

With manual drafting, if you resize any part of the drawing, you must erase and then redraw the dimensions. Changing text can often involve relettering the whole drawing. With CAD, you create associative dimensions and text on the layout in paper space. Associative dimensions are tied to the underlying model. Changes to the model automatically update the dimension values. Standard types of dimensions include linear, radial, ordinate, angular, baseline, and more. You can easily revise the content, font, size, spacing, and rotation of text in dimensions and notes.

Modify Your Drawing

Revisions are a part of any drawing project. Whether you work on paper or with CAD, you will need to modify your drawing in some way. On paper, you must erase and redraw to make revisions to your drawing manually. CAD eliminates tedious manual editing by providing a variety of editing tools. If you need to copy all or part of an object, you don't have to redraw it. If you need to remove an object, you can erase it with a few clicks of the mouse. And if you make an error, you can quickly undo your actions. Once you draw an object, you never need to redraw it. You can modify existing objects by mirroring, rotating, scaling, stretching, trimming, and more. You can also change object properties, such as linetype, lineweight, color, and layer, at any time.

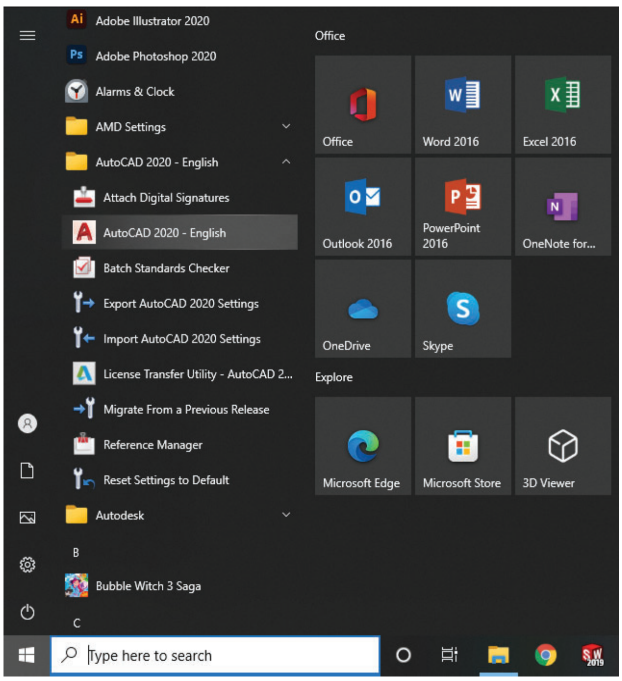
Drawing Office Management

With computer aided drafting, the size of the drawing office became considerably small. It increases the capacity of storing and preserving of the drawing without damage for a long time in less cost. Locating of drawing is easy task as comparing to conventional method of drawing office management

Starting Auto CAD

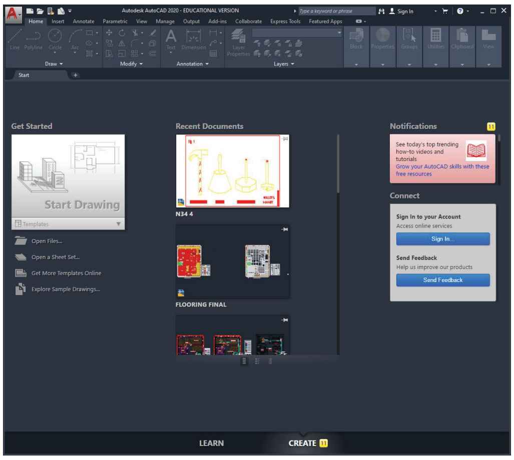
After you have installed Auto CAD, an AutoCAD icon is displayed on the desktop. You can start AutoCAD by double - licking on it. You can also start AutoCAD using the start menu. To do so, choose the start button at the bottom left corner of screen (default position); a menu will be displayed (Fig 1). From this menu, choose programs> Auto desk> AutoCAD - English > AutoCAD- English.

Fig 1



When you start AutoCAD, the welcome window will be displayed. In this window, various links are available providing information on enhancements, new features, products and services, subscription, and so on. The information available in this window is in form of text and videos. You can click on a link to find information contained in that link. For example, in the learn area, you can find information about new features and enhancements of AutoCAD. You can close the welcome widow (Fig 2) by closing the close button from it.

Fig 2



Auto CAD screen components (Fig 3)

Various components of the initial Auto CAD screen are drawing area, command window, menu bar, several toolsbars, model and layout tabs, and status bar. A title bar that has Auto CAD symbol and the current drawing name is displayed on top of the screen

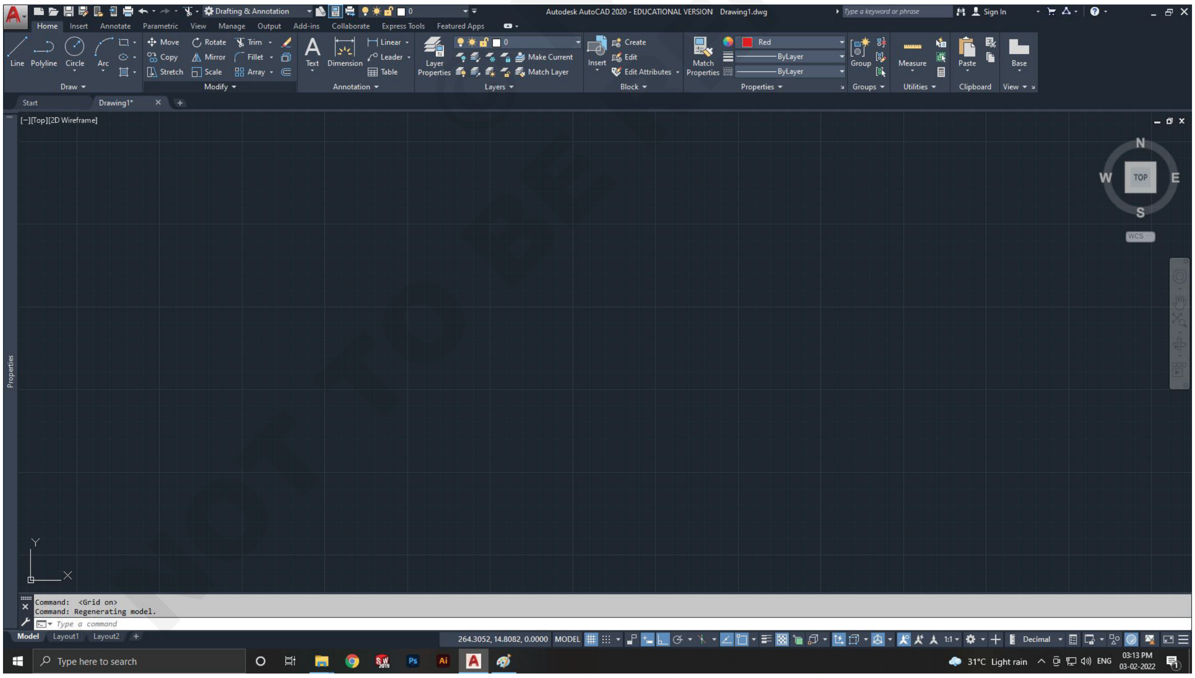
Drawing area

The drawing area covers the major portion of the screen. In this area, you can draw the objects and use the commands. to draw the objects, you need to define the coordinate points, which can be selected by using your pointing device. The position of the pointing device is represented on the screen by the cursor. The window also has the standard windows buttons such as close, minimize, scroll bar, and so on, on the top right corner. These buttons have the same functions as for any other standard window.

Command window

The command window at the bottom of the drawing area has the command prompt where you can enter the comments. It also displays the subsequent prompt sequences and the messages. You can change the size of the window by placing the cursor on the top edge (double line bar known as the grab bar) and then dragging. This way you can increase its size to see all the previous comments you have used. By default, the command window displays only three lines. You can also press the F2 key to display AutoCAD text window, which displays the previous comments and prompts.

Fig 3



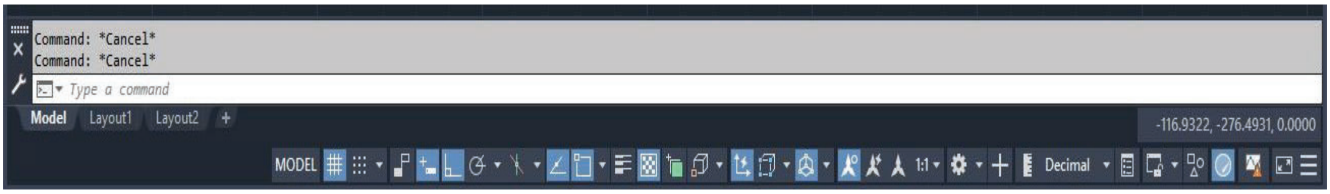
View cube

View cube is available on the top right corner of the drawing area and is used to switch between the standard and isometric views or roll the current view. The view cube and its options are discussed in later chapter.

Status bar (Fig 4)

The status bar is displayed at the bottom of the screed and is called application status Bar. It contains some useful information and buttons that make it easy to change the status of some AutoCAD functions. You can toggle between the on and off states of most of these functions by choosing them.

Fig 4



Drawing coordinates

The information about the coordinate is displayed on the left - corner of the status bar. You can select this coordinate button to toggle between the on and off states. The COORDS system variable controls the type of display of the coordinates. If the value of the COORDS variable is set to 0, the coordinate display is static, that is, the coordinate values displayed in the status bar change only when you specify a point. If value of the COORDS variable is set to 1 or 2, the coordinate display is dynamic. When the variable is set to 1, AutoCAD constantly displays the absolute coordinates of the graphics cursor with respect to the UCS origin. The polar coordinates (length < angle) are displayed if you are in an AutoCAD command and the COORDS variable is set to 2. By default, the COORDS variable is set to 1. You can also click on the drawing coordinates area to change the coordinate status from on to off and vice versa.

Infer constraints

If this button is chosen then some of the geometric constraints will be automatically applied to sketch when it is drawn. You can use CTRL + SHIFT+ I as a shortcut key to toggle this button.

Snap mode

If the snap mode button is chosen, the snap mode is on. So, you can move the cursor in fixed increments. The F9 key acts as a toggle key to turn the snap off or on.

Grind display

In AutoCAD, the grid lines are used as reference lines to draw objects. If the grid display button is chosen, the grid display is on and the grid lines are displayed on the screen. The F7 function key can be used to turn the grid display on or off.

Ortho mode

If the ortho mode button is chosen, you can draw lines at right angles only. You can use the F8 function key to turn ortho on or off.

Polar tracking

If you turn the polar tracking on, the movement of the cursor is restricted along a path based on the angles set as the polar angle. Choose the polar tracking button to turn the polar tracking on. you can also use the F10 function key to turn on this option. Note that turning the polar tracking on, automatically turns off the ortho mode.

Object snap

When the object snap button is chosen, you can use the running object snaps to snap on to a point. You can also use the F3 function key to turn the object snap on or off. The status of OSNAP (off or on) does not prevent you from using the immediate mode object snaps.

Object snap tracking

When you choose this button, the inferencing lines will be displayed. Inferencing lines are dashed lines that are displayed automatically when you select a sketching tool and track a particular key point on the screen. Choosing this button turns the object snap tracking on or off. You can also use the F11 function key to turn the object snap tracking on or off.

Allow/disallow dynamic UCS

Choosing this button allows or disallows the use of dynamic UCS. Allowing the dynamic UCS ensures that the XY plane of the UCS got dynamically aligned with the selected face of the model. You can also use the F6 function key to turn the dynamic UCS button on or off.

Dynamic input

The dynamic input button is used to turn the dynamic input on or off. Turning it on facilitates the heads-up design approach because all commands, prompts, and dimensional inputs will now be displayed in the drawing area and you do not need to look at the command prompt all the time. This saves the design time and also increase the efficiency of the use iff the dynamic input mode is turned on, you will be allowed to enter the commands through the pointer inputs boxes, and the numerical values through the dimensional input boxes. You will also be allowed to select the command options through the dynamic prompt options in the graphics window. To turn the dynamic input on or off, use the F12 key.

Show/ Hide line weight

Choosing this button in the status bar allows you to turn on or off the display of line weight in the drawing. If this button is not chosen, the display of line weight will be turned off.

Show/ Hide transparency

This button is available in the status bar and is choosen to turn on or off the transparency set for a drawing. You can set the transparency in the properties panel or in the layer in which the sketch is drawn.

Quick properties

If you select a sketched entity when this button is chosen in the status bar, the properties of the selected entity will be displayed in a panel. You can use CTRL + SHIFT + P as shown as a shortcut key to toggle this button.

Selection cycling

When this button is chosen, you can cycle through the objects to be selected, if they are overlapping or close to other entities. On selecting an entity when this button is chosen, the **Selection** list box with a list of the entities that can be selected will be displayed. You can use CTRL + W as a shortcut key to toggle this button.

Invoking tools in AutoCAD

On starting AutoCAD, when you are in the drawing area, you need to invoke AutoCAD tools to perform an operation. For example, to draw a line, first you need to invoke line tool and then define the start point and the endpoint of the line. Similarly, if you want to erase objects, you must invoke the erase tool and then select the objects for erasing. AutoCAD has provided the following methods to invoke the commands.

1 Key board

2 Ribbon

3 Application menu

4 Tool palettes

5 Menu bar

6 Shortcut menu tool bar

Key board

You can invoke any AutoCAD command from the keyboard by typing the command name and then pressing the Enter key. As you type the first letter of command, AutoCAD displays all available commands starting with the letter typed. If the Dynamic input is on and the cursor is in the drawing area, by default, the command will be entered through the pointer input box. The pointer input box is a small box displayed on the right of the cursor, as shown in Fig 5. However, if the cursor is currently placed on any toolbar or menu bar, or if the dynamic input if turned off, the command will be entered through the command prompt. Before you enter a command, the command prompt is displayed as the last line in the command window area. If it is not displayed, you must cancel the existing command by pressing the ESC (Escape) key.

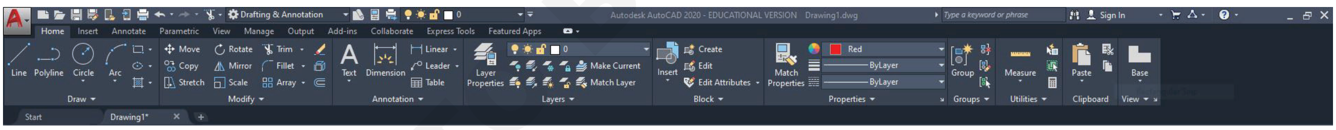
The following example shows how to invoke the line command using the keyboard (Fig 5).

Fig 5

Ribbon

In AutoCAD, you can also invoke a tool from the Ribbon. The tools for creating, modifying and annotating the 2D and 3D designs are available in the panels instead of being spread out in the entire drawing area in different tool bars and menus (Fig 6).

Fig 6



When you start the AutoCAD session for the first time, by default the ribbon is displayed horizontally below the quick access toolbar. The ribbon consists of various tabs. The tabs have different panels, which in turn, have tools arranged in rows. Some of the tools have small black down arrow. This indicates that the tools having similar functions are grouped together. To choose a tool, click on the down arrow; a drop- down will be displayed. Choose the required too from the drop- down displayed. Note that if you choose a tool from the drop- down, the corresponding command will be invoked and the tool that you have chosen will be displayed in the panel. For example, to draw a circle using the 2- point option, click on the down arrow next to the center, radius tool in the draw panel of the home tab; a flyout will be displayed. Choose the 2-point tool from the flyout and then draw the circle. You will notice that the 2-point tool is displayed in place of the center, radius tool. In this textbook, the tool selection sequence will be written as, choose the 2-point tool from home> draw> circle drop -down. Choose the down arrow to expand the panel. You will notice that a push pin is available at the left end of the panel. Click on the push pin to keep the panel in the expanded state. Also, some of the panels have an inclined arrow at the lower- right corner. When you left click on inclined arrow, a dialog box is displayed. You can define the setting of the corresponding panel in the dialog box.

You can reorder the panels in the tab. To do so, press and hold the left mouse button on the panel to be moved and drag it to the required position. To undock the ribbon, right-click on the blank space in the ribbon and choose the undock option. You can move, resize, anchor, and auto - hide the ribbon using the shortcut menu that will be displayed when you right - click on the heading strip. To anchor the floating ribbon to the left or right of the drawing area in the vertical position, right- click on the heading strip of the floating ribbon; the shortcut menu is displayed. Choose the corresponding option from this shortcut menu. The auto - hide option will be hide the ribbon into the heading strip and will display it only when you move the cursor over this strip.

You can customize the display of tabs and panels in the ribbon. To customize the ribbon, right-click on any one of the tools in it; a shortcut menu will be displayed. On moving the cursor over one of the options, a flyout will be displayed with a tick mark before all options and the corresponding tab or panel will be displayed in the ribbon. Select/ clear appropriate option to display / hide a particular tab or panel.

Application menu

The application menu is available at top - left of the AutoCAD window. It contains some of the tools that are available in the standard toolbar. Click the down arrow on the application menu to display the tools, as shown in Fig 7. You can search for tools or commands by using the search field on the top of the application menu. To search a tool or command, enter its complete or partial name in the search field; the list of related tools and commands will be displayed. If you click on a tool from the list, the corresponding command will get activated.

By default, the recent document button is chosen in the application menu. Therefore, the recently opened drawings will be listed. If you have opened multiple drawing files, choose the open documents button; the documents that are opened will be listed in the application menu. To set the preferences of the file, choose the option button available at the bottom right of the application menu. To exit AutoCAD, choose the exit button next to the options button.

Menu bar

You can also select commands from the menu bar. Menu bar is not displayed by default. To display the menu bar, click on the down arrow in the quick access toolbar, a flyout is displayed. Select the show menu bar options from it; the menu bar will be displayed. As you move the cursor over the menu bar, different tabs are highlighted. You can choose the desired item by left - clicking on it; the corresponding menu is displayed directly under the title. For example, to draw an ellipse using the center option.

Tool bar

Toolbars are not displayed by default. To display a toolbar, choose the view tab in the Ribbon and click on Toolbars in the User interface panel; a flyout will be displayed. Select the required toolbar. Alternatively, display the menu bar and then choose Tools > toolbars> AutoCAD from it; a list of toolbars will be displayed. Select the required toolbar. (Fig 8) shows the draw toolbar invoked.

Fig 7

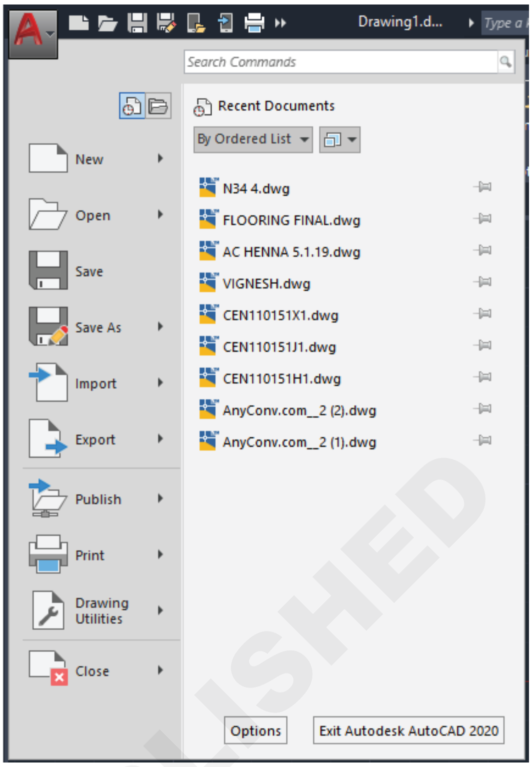
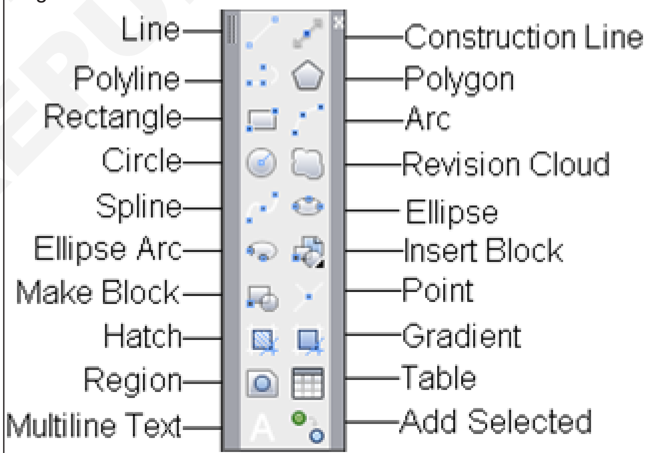


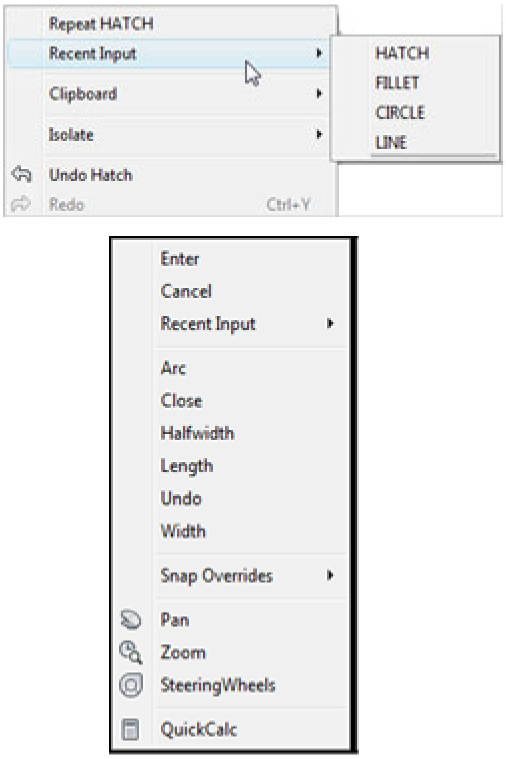
Fig 8



Shortcut Menu

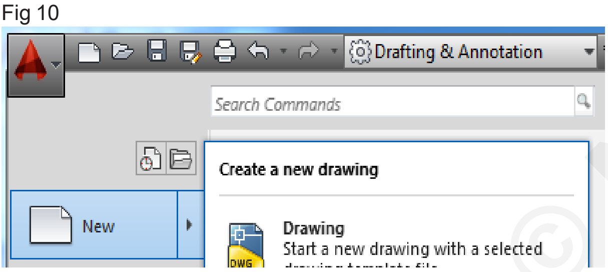
AutoCAD has provided shortcut menus as an easy and convenient way of invoking the recently used tools. These shortcut menus are context-sensitive, which means that the tools present in them are dependent on the place/ object for which they are displayed at the cursor location. You can right - click anywhere in the drawing area to display the general shortcut menu. It generally contains an option to select the previously invoked tool again, apart from the common tools for windows, refer to Fig 9.

Fig 9



Starting a new drawing (Fig 10)

Fig 10



You can open a new drawing using the new tool in the quick access toolbar. When you invoke the new tool, by default AutoCAD will display the select template dialog box, as shown in Fig 11. This dialog box displays a list of default templates available in AutoCAD. The default selected template is acad.dwt, which starts the 2D drawing environment. You can select the acad3D. dwl template to start the 3D modeling environment. Alternatively, you can select any other template to start new drawing that will use the settings of the selected template. You can also open any drawing without using any template either in metric or imperial system. To do so, choose the down arrow on the right of the open button and select the open with no template - metric option or the open with no template - imperial option from the flyout.

You can also open a drawing using the use a wizard and start from scratch options from the create new drawing dialog box. By default, this dialog box is not invoked. To invoke the create new drawing dialog box, enter startup in the command window and then enter 1 as the new value for this system variable. After setting 1 as the new value for the system variable, whenever you invoke the new tool, the create new drawing dialog box will be displayed, as shown in Fig 12. The options in this dialog box are discussed next.

Fig 11

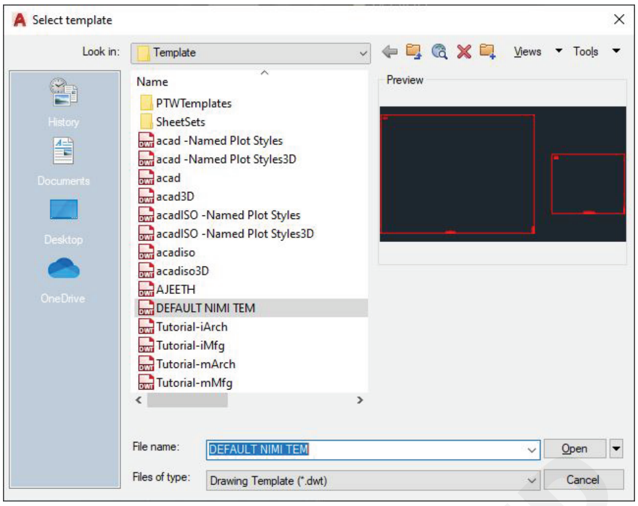
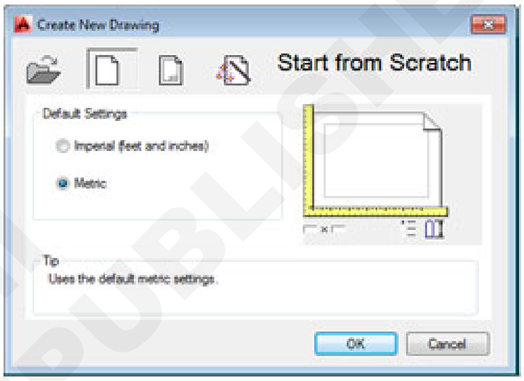


Fig 12



Open a drawing: By default, this option is not available. This option will be available when you start a new session of AutoCAD. This option is discussed later in this chapter.

Start from scratch

When you choose the start from scratch button, AutoCAD provides you with options to start a new drawing that contains the default AutoCAD setup for imperial (Acad. Dwl) or metric drawings (Acadiso.dwl). If you select the imperial default setting, the limits are 12X9, text height is 0.20, and dimensions and line types scale factors are 1.

Use a template

When you choose the use a template button in the create new drawing dialog box, AutoCAD displays a list of templates (Fig 13). The default template file is acad. Dwl or acadiso. Dwl, depending on the installation. You can directly start a new file in the 2D skating environment by selecting the acad.dwl or acadiso.dwl template. If you use a template file, the new drawing will have the same settings as specified in the temple file.

Use a wizard

The use a wizard option allows you to set the initial drawing settings before actually starting a new drawing. When you choose the use a wizard button, AutoCAD provides you with the option for using the quick setup or advanced setup (Fig 14). In the quick setup, you can specify the units and the limits of the work area. In the advanced setup, you can set the units, limits, and the different types of settings for a drawing.

Fig 13

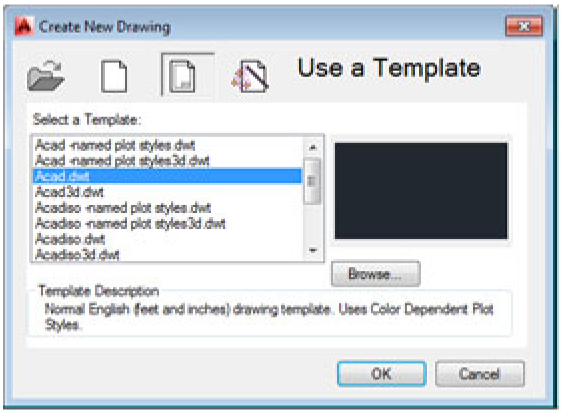
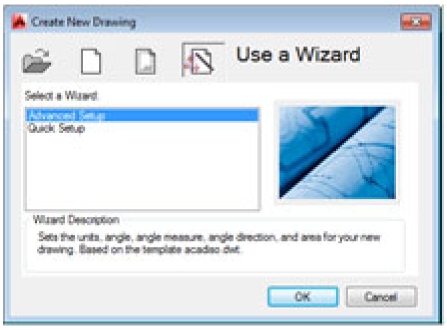


Fig 14

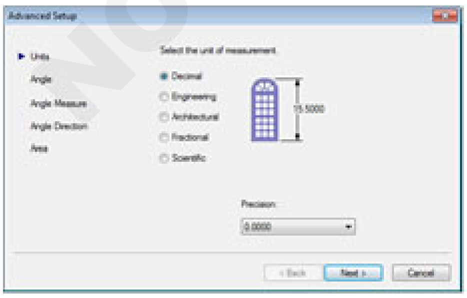


Advance setup

This option allows you to preselect the parameters of a new drawing such as the units of linear and angular measurements, type and direction of angular measurements, approximate area desired for the drawing, precision for displaying the units after decimal, and so on. When you select the Advanced Setup wizard option from the Create New Drawing dialog box and choose the OK button, the Advanced Setup wizard is displayed. In the wizard, the Units page is displayed by default, as shown in Fig 15.

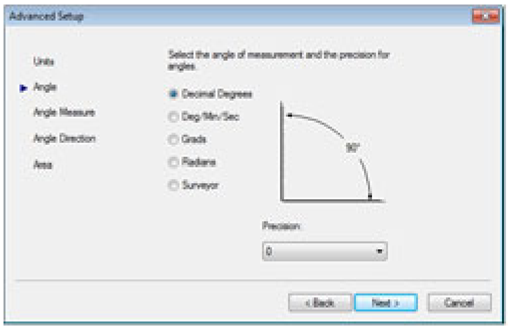
This page is used to set the units for measurement in the current drawing. You can select the required unit of measurement by selecting the respective radio button. You will notice that the preview image is modified accordingly. The different units of measurement you can choose from are decimal, engineering, architectural, fractional, and scientific. You can also set the precision for the measurement units by selecting it from the precision drop-down list.

Fig 15



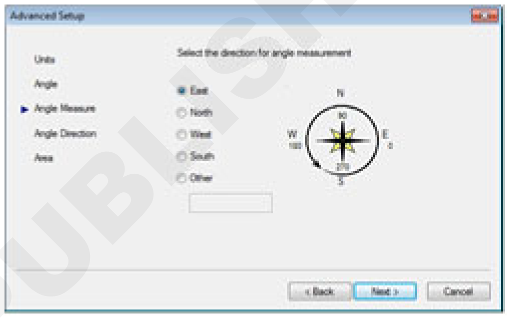
This page is used to set the units for angular measurement and its precision. The units for angle measurement are decimal degrees, deg/min/sec, grads, radians, and surveyor. The units for angle measurement can be set by selecting any one of these radio buttons as required. The preview of the selected angular unit is displayed on the right of the radio buttons (Fig 16).

Fig 16



The next page is the angle measure page, as shown in Fig 17. This page is used to select the direction of the baseline from which the angles will be measured. You can also set your own direction by selecting the other radio button and then entering the value in its edit box.

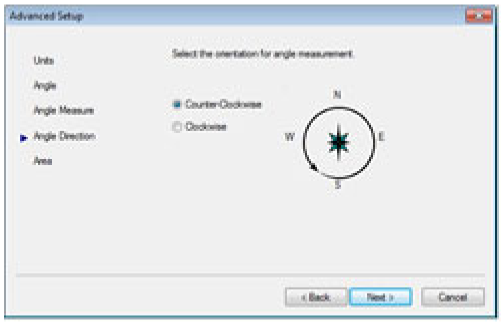
Fig 17



Choose next to display the angle direction page (Fig 18) to set the orientation for the angle measurement. By default, the angles are positive, if measured in a counter clock wise direction.

This is because the counter-clockwise radio button is selected. If you selected the clockwise radio button, the angles will be considered positive when measured in the clockwise direction.

Fig 18



To set the limits of the drawing, choose the next button; the area page will be displayed, as shown in Fig 19. You can enter the width and length of the drawing area in the edit boxes.

Quick setup

When you select the quick setup option and choose the ok button, the quick setup wizard is displayed. This wizard has two pages; units and area. The units page is opened by default, as shown in Fig 20. The only difference is that you cannot set the precision for the units in this wizard.

Fig 19

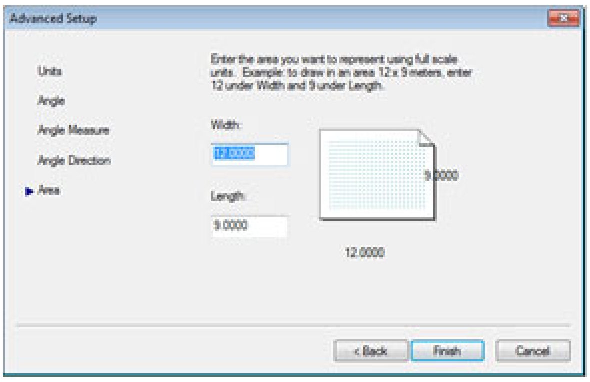
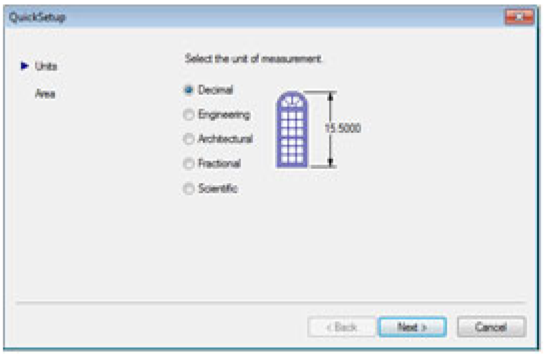
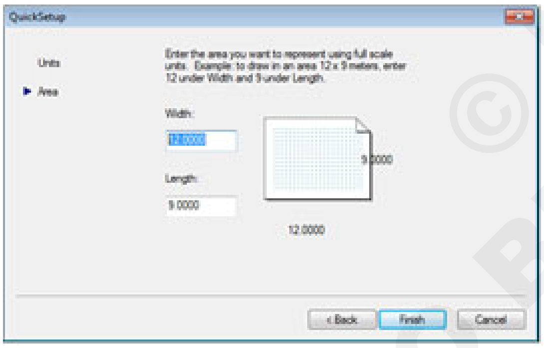


Fig 20



Choose next to display the area page, as shown in Fig 21. The area page of the quick setup is similar to that of the advanced setup wizard. In this page, you can set the drawing limits.

Fig 21



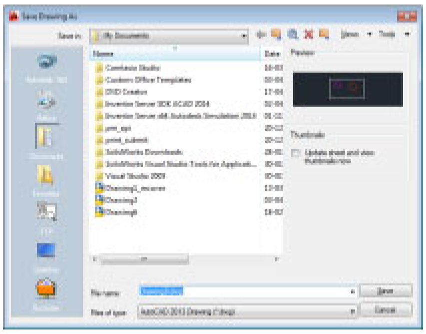
Tip: when you open an AutoCAD session, a drawing opens automatically. But you can open a new drawing using options such as start from scratch and use a wizard before entering into AutoCAD environment using the startup dialog box. As mentioned earlier, the display of the startup dialog box is turned off by default. Refer to the section of starting a new drawing to know how to turn on the display of this dialog box.

Saving your work (Fig 22)

You must save your work before you exit the drawing editor or turn off your system. Also, it is recommended that you save your drawings in regular intervals, so that in the event of a power failure on an editing error, all works saved before the problem started will be retained.

AutoCAD has provided the Qsave, save as, and save commands that allow you to save your work. These commands allow you to save your drawing by writing it to a permanent storage device, such as a hard drive or in any removable drive.

Fig 22



When you choose save as from the application menu or choose the save as tool from the quick access tool bar, the save drawing as dialog box is displayed. Even if the drawing has been saved with a file name, this tool gives you an option to save it with a different file name. in addition to saving the drawing, it sets the name of the current drawing to the file name you specify, which is displayed in the title bar. This tool is used when you want to save a previously saved drawing with different file name. you can also use this tool when you make certain changes to a template and want to save the changed template drawing but leave the original template unchanged.

File name edit box

To save your work, enter the name of the drawing in the file name edit box typing the file name or selecting it from the drop- down list. If you select the file name, it automatically appears in the file name edit box. If you have already assigned a name to the drawing, the current drawing name is taken as the default name. if the drawing is unnamed, the default name drawing / is displayed in the file name edit box.

Closing a drawing

You can close the current drawing file without quitting AutoCAD by choosing close> current drawing from the application menu or by entering close at the command prompt. If multiple drawing files are opened, choose close> all drawings from the application menu. If you have not saved the drawing after making the last changes to it and you invoke the close command. AutoCAD displays a dialog box that allows you to save the drawing before closing. This box gives you an option to discard the current drawing or the changes make to it. It also gives you an option to cancel the command. After closing the drawing, you are still in AutoCAD from where you can open a new or an already saved drawing file. You can also the close button (X) of the drawing area to close th drawing.

Opening an existing drawing using the drag and drop method

You can also open an existing drawing in AutoCAD by dragging it from the window explorer and dropping it into AutoCAD. If you drop the selected drawing in the drawing area, the drawing will be inserted as a block and as a result you cannot modify it. But, if you drag the drawing from the window explorer and drop ir any where other that the drawing area, AutoCAD opens the selected drawing.

Quitting Auto CAD

You can exit the Auto CAD program by using the EXIT or QUIT commands. Even if you have an active commands, you can choose Exit Autodesk AutoCAD from the Application Menu to quit the AutoCAD program. In Case the drawing has not been saved, it allows you to save the work first through a dialog box. Note that if you choose No in this dialog box ,all the changes made in the current list till the last save will be lost. You can also use the Close button (X) of the main AutoCAD window (present in the title bar) to end the AutoCAD session.

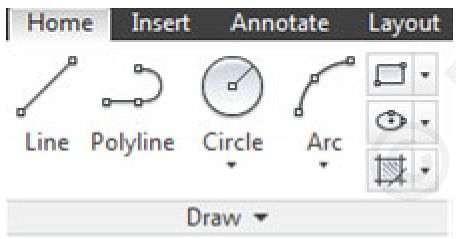
Drawing of 2D objects

Objectives: At the end of this lesson you shall be able to
- **brief how to draw lines using the line command and its options**
- **explain various coordinate systems used in AutoCAD**
- **describe the erase commands to clear the drawing area, undo, redo and oops command**
- **brief the two basic object selection methods**
- **explain how to draw circles recatangles, polygon, ellipse, point using the options of the circle command.**

Draw lines in AutoCAD

The most commonly used fundamental object in a drawing is line. In AutoCAD, a line is drawn between two points by using the line tool. You can invoke the line tool from the draw panel of the home tab in the ribbon. Besides this, you can choose the line tool from the draw tab of the tool palettes. To invoke the tool palettes, choose the tool palettes button from the palettes panel in the view tab. Alternatively, you can invoke the line tool from the draw tool. However, the draw toolbar is not displayed by default. To invoke this toolbar, choose View > User Interface > Toolbars > AutoCAD from the Ribbon. (Fig 1)

Fig 1



You can also invoke the line tool by entering line or L (L is the alias for the Line command) at the command prompt. On invoking the line tool, you will be prompted to specify the starting point of the line. Specify a point by clicking the left mouse button in the drawing area or by entering its coordinates in the dynamic input fields or the command prompt. After specifying the first point, you will be prompted to specify the second point. Specify the second point; a line will be drawn. You may continue specifying points and draw lines or terminate the line tool by pressing ENTER, ESC, or SPACEBAR. You can also right-click to display the shortcut menu and then choose the enter or cancel options from it to exit the line tool. (Fig 2)

Command: Line

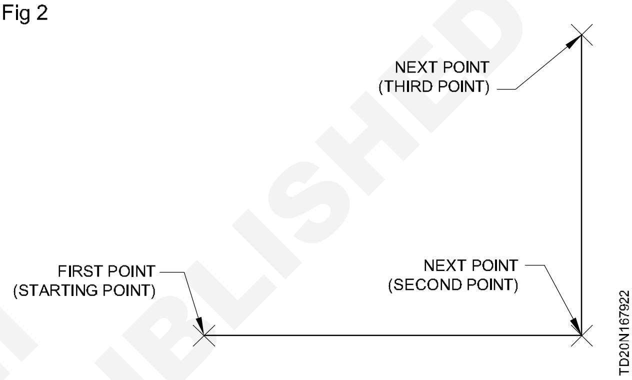
Specify first point: Move the cursor (mouse) and left-click to specify the first point.

Specify next point or [Undo]: Move and left-click to specify the second point.

Specify next point or [Undo]: Specify the third point.

Specify next point or [Close/Undo]: (Press Enter to exit the line command).

Fig 2



Note: that in the command prompt the close and undo options will be displayed while creating line using the line tool. Both these options are discussed next.

The close option (Fig 3)

After drawing two continuous lines by using the line tool, you will notice that the close option is displayed at the command prompt. The close option is used to join the current point to the start point of the first line when two or more continuous lines are drawn. If you are specifying the endpoint by using the mouse, then click at the start point of the first line or enter C at the command prompt, as given in the command prompt below.

Choose the line tool

Line specify first point: Pick the first point.

Specify next point or [Undo]: Pick the second point.

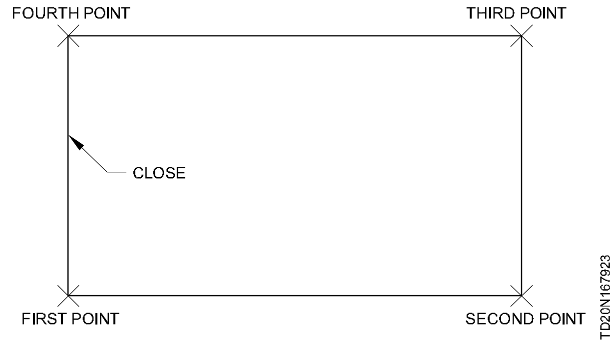
Specify next point or [Undo]: Pick the third point.

Specify next point or [Close/Undo]: Pick the fourth point.

Specify next point or [Close/Undo]: (The fifth point joint with the first point).

You can also choose the close option from the shortcut menu, which appears when you right-click in the drawing area.

Fig 3



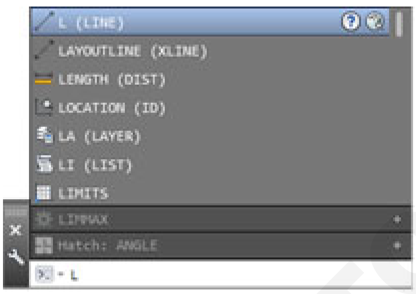
The undo option

While drawing a line, if you have specified a wrong endpoint by mistake, then you can undo the last specified point and go back to the previous stage by using the undo option of the line tool. You can use this option multiple times. To use this option, type undo (or just U) at the specify next point or [Undo] prompt. You can also right-click to display the shortcut menu and then choose the undo option from it.

Invoking tools using dynamic INPUT/Command Prompt

In AuotCAD, if you enter any alphabet at the command prompt or dynamic input, all tools whose names start with the entered alphabet will be displayed in a list at the command prompt or dynamic input. For example, if you enter L at the command prompt or dynamic input, all the tools whose names start with the alphabet L will be displayed, (Fig 4). In this way, you can view all the tool names starting with a particular alphabet and select the required tool.

Fig 4

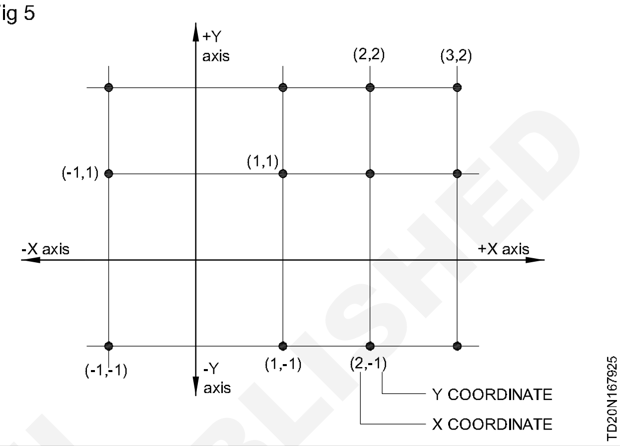


Coordinate systems (Fig 5)

In AuotCAD, the location of a point is specified in terms of Cartesian coordinates. In this system, each point in a plane is specified by a pair of numerical coordinates. To specify a point in a plane, take two mutually perpendicular lines as references. The horizontal line is called the X axis, and the vertical line is called the Y axis. The X and Y axes divide the XY plane into four parts, generally known as quadrants. The point of intersection of these two axes is called the origin and the plane is called XY plane. The origin has the coordinate values of X = 0, Y = 0. The origin is taken as the reference for locating a point on the XY plane. Now, to locate a point, say P, draw a vertical line intersecting the X axis. The horizontal distance between the origin and the intersection point will be called the X coordinate of P. It will be denoted as P (x). The X coordinate specifies how far the point is to the left or right from the origin along the X axis.

Now, draw a horizontal line intersecting the Y axis. The vertical distance between the origin and the intersection point will be the Y coordinate of P. It will be denoted as P(y). The Y coordinate specifies how far the point is to the top or bottom from the origin along the Y axis. The intersection point of the horizontal and vertical lines is the coordinates of the point and is denoted as P (x, y). The X coordinate is positive, if measured from the right of the origin and is negative, if measured from the left of the origin. The Y coordinate is positive, if measured above the origin and is negative, if measured below the origin.

Fig 5



In AuotCAD, the default origin is located at the lower left corner of the drawing area.

AuotCAD uses the following coordinate systems to locate a point in an XY plane.

1 Absolute coordinates

2 Relative coordinates

3 Polar coordinates

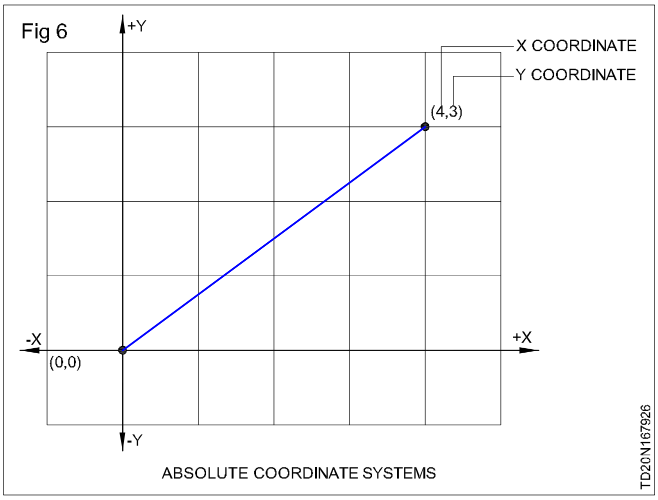
4 Direct distance entry

If you are specifying a point by entering its location at the command prompt then you need to use any one of the coordinate system.

Absolute coordinate system (Fig 6)

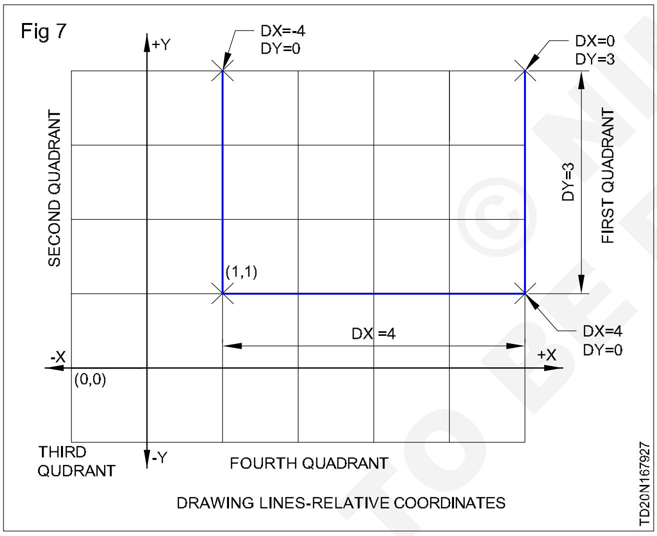
In the absolute coordinate system, points are located with respect so the origin (0,0). For example, a point with X = 5 and Y = 3 is measured 4 units horizontally (distance along the X axis) and 3 units vertically (distance along the Y axis) from the origin. In AuotCAD, the absolute coordinates are specified at the command prompt by entering X and Y coordinates, separated by a comma. Hoever, remember that if you are specifying the coordinates by using the dynamic input mode, you need to add # as the prefix to the X coordinate value. For example, enter #1,1 in the dynamic input boxes to use the absolute coordinate system. The following example illustrates the use of absolute coordinates at the command prompt to draw the rectangle.

In AuotCAD the absolute coordinates are specified by entering X and Y coordinates, separated by the comma.



Relative coordinate system (Fig 7)

In the relative coordinate system, the displacements along the X and Y axes (DX and DY) are measured with reference to the previous point rather than to the origin. In AuotCAD, the relative coordinate system is designated by the symbol @ and it should proceed any relative entry. The following prompt sequence illustrates the use of the relative rectangular coordinate system to draw a rectangle with the lower left corner at the point (1,1). The length of the rectangle is 4 units and the width is 3 units.



Remember that if dynamic input is on, you need a input a comma (,) after entering the first value in the dynamic input boxes. Else, AuotCAD LT will take coordinates in relative polar form.

Sign conention: As mentioned, in the relative rectangular coordinate system the displacements along the X and Y axes are measured with respect to the previous point, Imagine horizontal line and a vertical line passing through the previous point so that you get four quadrants. If the new point is located in the first quadrant, the displacements DX and DY are both positive. If the new point is located in the third quadrant, the displacements DX and DY are both negative. In other words, up or right are positive and down or left are negative.

Polar coordinates (Fig 8)

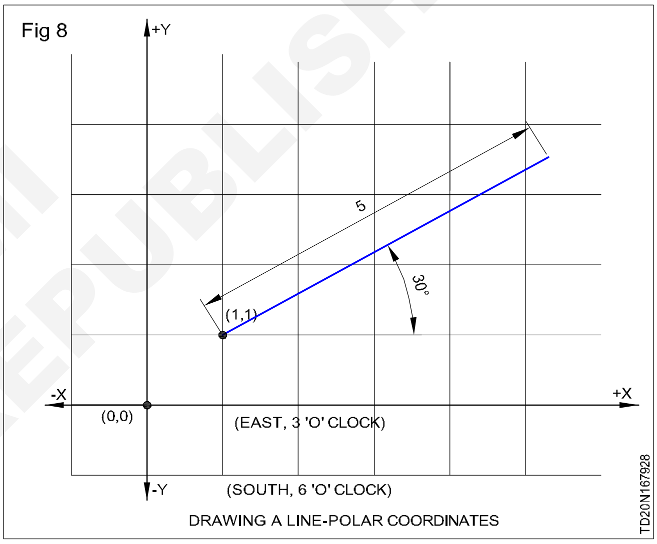
In the relative polar coordinate system, a point is located by defining both the distance of the point from the current point and the angle that the line between the two points makes with the positive X axis. The prompt sequence to draw a line from a point at 1,1 to a point at a distance of 5 units from the point (1, 1), and at an angle of 30-degree to the X axis is given next.

Command: Line

Specify first point: 1, 1

Specify next point or [Undo]: @5<30

Sign convention. By default, in the relative polar coordinate system, the angle is measured from the horizontal axis (3 'O' clock) as the zero-degree baseline. Also, the angle is positive if measured in a counter clock wise direction and negative if measured in a clockwise direction. Here, it is assumed that the default setup of the angle measurement has not been changed.



Direct distance entry (Fig 9)

The easiest way to draw a line in AuotCAD is by using the direct distance entry method. Before drawing a line by using this method, ensure that the dynamic input button is chosen in the status bar. Next, choose the line tool; you will be prompted to specify the start point. Enter the coordinate values in the text box and press enter; you will be prompted to specify the next point. Now, enter the absolute length of the line and its angle with respect to the current position of the cursor in the corresponding text boxes. Note that you can use the TAB key to toggle between the text boxes. If the ortho mode is on while drawing lines using this method. You can position the cursor only along the X or Y axis. Therefore, position the cursor at the desired angle, type the length at the command prompt, and then press enter.

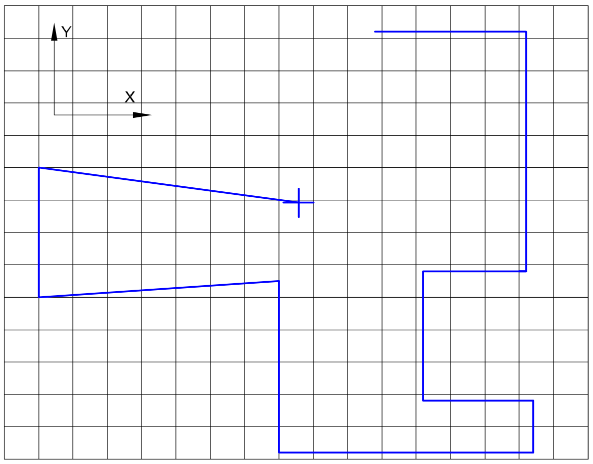
Choose the line tool.

Line specify first point: Start point

Specify next point or [Undo]: Position the cursor and then enter distance.

Specify next point or [Undo]: Position the cursor and then enter distance.

Fig 9

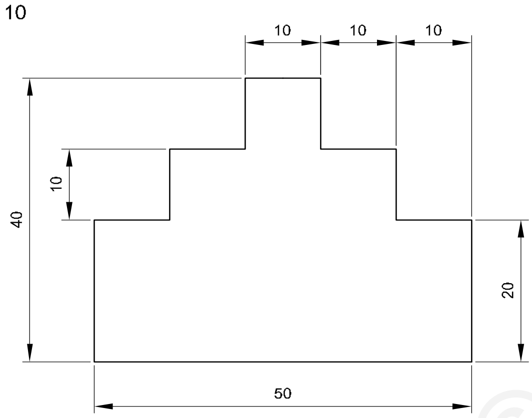


DRAWING LINES USING THE DIRECT DISTANCE ENTRY METHOD

TD20N167929

Ortho mode (F8) method (Fig 10)

Fig 10



TD20N16792A

Erasing objects (Fig 11&12)

Sometimes, you may need to erase the unwanted objects from the objects drawn. You can do so by using the erase tool. This tool is used exactly the same way as an eraser is used in manual drafting to delete the unwanted lines. To erase an object, choose the erase tool from the modify panel. You can also choose the erase button from the modify toolbar. To invoke the modify toolbar, choose view > User Interface > Toolbars > AutoCAD > Modify from the ribbon. On invoking the erase tool, a small box, known as pick box, replaces the screen cursor. To erase the object, select it by using the pick box. The selected object will be displayed in dashed lines and the select objects prompt will be displayed again. You can either continue selecting the objects or press enter to terminate the object selection process and erase the selected objects. The prompt sequence is given next.

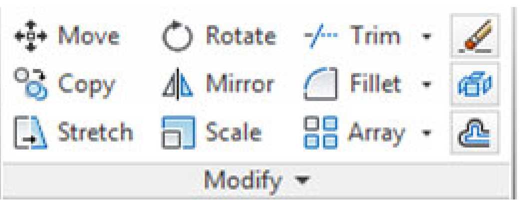
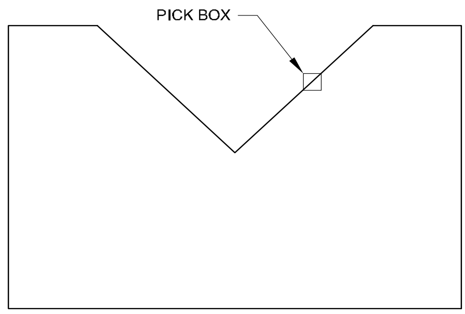


Fig 12



SELECTING THE OBJECT BY POISTIONING THE PICK BOX AT THE TOP OF THE OBJECT

TD20N16792C

If you enter All at the select objects prompt, all objects in the drawing area will be selected, even if they are outside the display area. Now, if you press enter, all the selected objects will be erased.

To erase the objects, you can also first select the objects to be erased from the drawing and then choose the erase option from this shortcut menu that is displayed on right-clicking in the drawing area.

Cancelling and undoing a operation (Fig 13)

If you are in a command and you want to cancel or get out of it, press the ESC (Escape) key on the keyboard.

Command: Erase

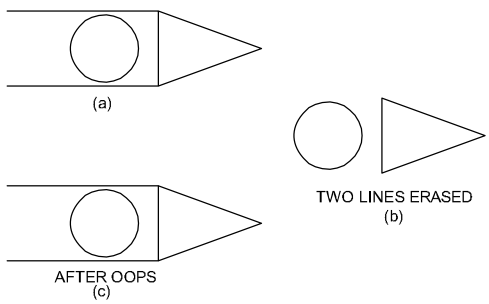
Select objects: Press ESC (Escape) to cancel the command

Similarly, sometimes you unintentionally erase some object from the screen. When you discover such an error, you can correct it by restoring the erased object by means of the oops command. The oops command restores object that have been accidentally erased by the previous erase command. You can also use the U (Undo) command to undo the last command.

Command: Oops (Restores erased objects)

Command: U (Undoes the last command)

Fig 13



TD20N16792D

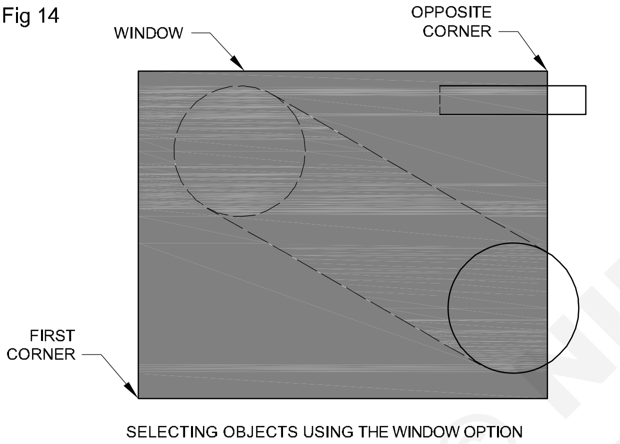
Object selection methods

The usual method to select objects is by selecting them individually. But it will be time consuming, if you have a number of objects to select. This problem can be solved by creating a selection set that enables you to select several objects at a time. The selection set options can be used with those tools that required object selection, such as erase and move. There are many object selection methods, such as Last, Add, Window, Crossing, and so on. Here you will learn two methods: Window and Crossing.

Window selection (Fig 14)

The window selection is one of the selection methods in which an object or group of objects are selected by drawing a window. The objects that are completely enclosed within the window are selected and the objects that lie partially inside the boundaries of the window are not selected. To select the objects by using the window option after invoking a tool, type W at the select objects prompt and press Enter; you will be prompted to specify the first corner of the window. Select the first corner and then move the cursor to specify the opposite corner. As you move the cursor, a blue color window of continuous line will be displayed. The size of this window changes as you move the cursor. Specify the opposite corner of the window; the objects that are enclosed in this window are displayed as dashed objects. Figure 13 show the window drawn to select by using the window option. The objects that will be selected are shown in dashed lines.

Fig 14



SELECTING OBJECTS USING THE WINDOW OPTION

You can also invoke the window option without entering W at the command prompt. To do so, specify a point on the screen at the select objects prompt. This is considered as the first corner of the window. Moving the cursor to the right will display a blue-shaded window. After enclosing the required objects, specify the other corner of the window. The objects that are completely enclosed within the window will be selected and displayed in dashed lines. The following is the prompt sequence for automatic window after invoking the erase tool:

Select objects: Select a blank point as the first corner of the window

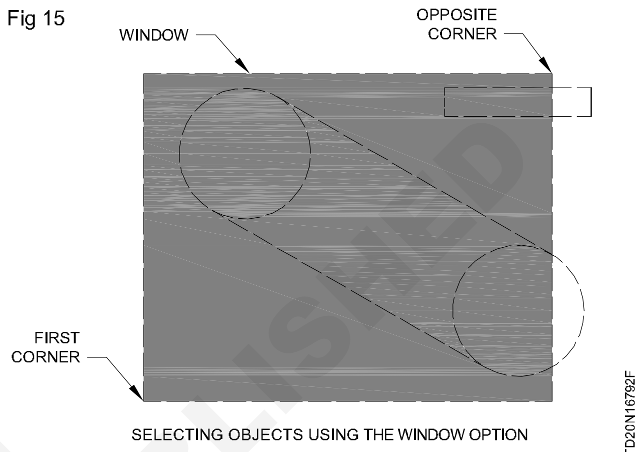
Specify opposite corner: Drag the cursor to the right to select the other corner of the window.

Select objects: Enter

Window crossing method (Fig 15)

The window crossing selection is one of the selection methods in which an object or group of objects that are completely or partially enclosed by the selection window are selected. The objects to be selected should touch the window boundaries or completely enclosed within it. To select the objects by using the window crossing method after invoking a tool, type C at the select objects prompt and press enter; you will be prompted to select the first corner of the window. Select the first corner and then move the cursor to specify the opposite corner. As you move the cursor, a green color window with dashed outline is displayed. Specify the opposite corner of the window; the objects that touch the window boundaries and the objects that are enclosed by the window are selected and displayed as dashed objects. Fig 15 shows a window drawn to select objects by using the window crossing method. The objects that will be selected are shown in dashed lines.

Fig 15



SELECTING OBJECTS USING THE WINDOW OPTION

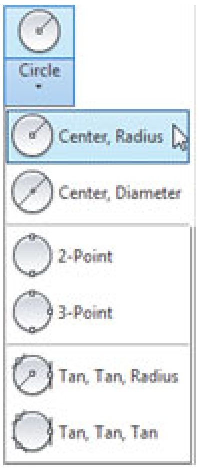
You can also invoke the window crossing method without entering C at the command prompt. To do so, specify a point in the drawing area at the select objects prompt and move the cursor to the left. As you move the cursor, a green color window with dashed outline will be displayed. Specify the opposite corner of the window; the objects touching the window boundary and that are enclosed within this window are selected and displayed as dashed objects. The prompt sequence for the automatic window crossing method when you choose the erase tool is given next.

If you do not invoke any tool and click to specify the first corner of the window for window selection or window crossing, the command prompt provides you with three selection options: Fence, Wpolygon, and Cpolygon. If you enter fence or F at the command prompt, you can select objects by drawing a fence around them. If you enter WP at the command prompt, you can select objects by drawing a polygon around them. If you enter CP at the command prompt, you can select objects by drawing a polygon around them.

Draw a circle (Fig 16)

A circle is drawn by using the circle tool. In AutoCAD, you can draw a circle by using six different tools. All these tools are grouped together in the draw panel of the ribbon. To view these tools, choose the down arrow next to the center, radius tool in the draw panel. All tools will be listed in a drop-down. Note that the name of the tool chosen last will be displayed in the draw panel.

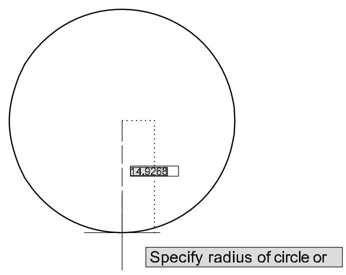
Fig 16



Draw a circle by specifying center and radius (Fig 17)

To draw a circle of specifying its center and radius first ensure that the dynamic input button is choosen, and then choose the center, radius tool from the draw panel; you will be prompted to specify the center of the circle. Type the coordinates and press enter or specify the center by using the left mouse button. After specifying the center of the circle, move the cursor to define its radius; the current radius of the circle will be displayed in the dimension input box. This radius value will change as you move the cursor. Type a radius value in the dimension input box or click to define the radius; the radius; a circle of the specified radius value will be drawn.

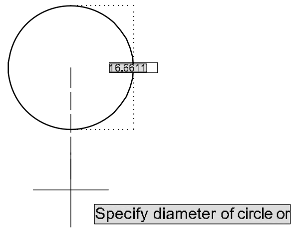
Fig 17



Drawing a circle by specifying center and diameter (Fig 18)

To draw a circle by specifying its center and diameter, first ensure that the dynamic input button is chosen and then choose the center, diameter tool too from the draw panel; you will be prompted to specify the center. Type the coordinates and press enter or specify the center by using the left mouse button. After specifying the center of the circle, move the cursor to define its diameter; the current diameter of the circle will be displayed in the dimension input box. This diameter value will change as you move the cursor. Type a diameter value in the dimension input box or click to define the diameter; a circle of the specified diameter value will be drawn.

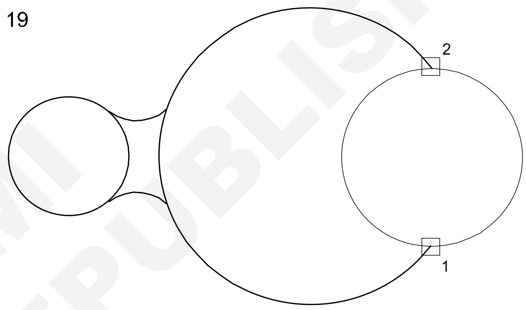
Fig 18



Drawing a circle by specifying two points (Fig 19)

Ribbon; Home > Draw > Circle drop-down > 2-point command: Circle or C > 2P you can also draw a circle by specifying its two diametrical ends. To do so, first ensure that the dynamic input button is chosen, and then choose the 2-point tool from the draw panel; you will be prompted to specify the first end of the diameter. Type the coordinates and press enter or specify the center by using the left mouse button. After specifying the center of the circle, move the cursor to define its diameter. Now, you can type the coordinates or diameter in the dimension input box.

Fig 19



A CIRCLE DRAWN USING THE 2 POINT OPTION

Drawing a circle by specifying three points on a circle (Fig 20)

To draw a circle by specifying three points on its periphery, choose the 3-point tool from the draw panel and specify the three points in succession. You can type the coordinates of the points or specify them by using the left mouse button. The prompt sequence to type the three coordinates on choosing the 3-point tool is given below.

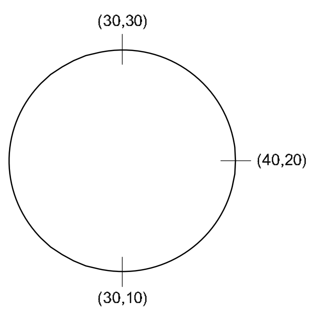
Specify center point for circle or [3P/2P/Ttr (tan tan radius)]:3p

Specify first point on circle: 30, 30

Specify second point on circle: 30, 10

Specify third point on circle: 40, 20

Fig 20

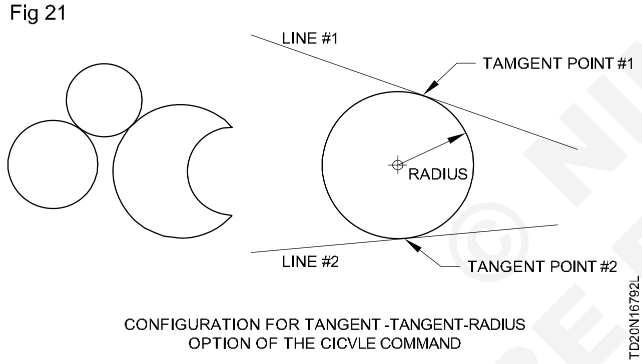


A CIRCLE DRAWN USING THE 3 POINT OPTION

Drawing a circle tangent to two objects (Fig 21)

An object (line, circle, or arc) is said to be tangent to a circle or an arc, if it touches the circumference of the circle or the arc at only one point. To draw a circle that has specified radius and is tangent to two objects, first ensure that the dynamic input button is chosen, and then choose the tan, tan, radius tool from the draw panel; you will be prompted to specify a point on the first object to be tangent to the circle. Move the cursor near the object to be made tangent to the circle; a tangent symbol will be displayed. Specify the first point; you will be prompted to specify a point on the second object to be made tangent to the circle. Move the cursor near the second object that is to be tangent to the circle; a tangent symbol will be displayed. Specify the second point; you will be prompted to specify the radius. Type the radius value in the dimension input box and press enter; a circle of the specified radius and tangent to two specified objects will be drawn. In figure the dotted circle represents the circle that is tangent to two objects. The circle actually drawn depends on how you select the objects to be made tangent to the new circle. The figures show the effect of selecting different points on the objects. If you specify too small or large radius; you may get unexpected results or the @circle does not exit@ prompt.

Fig 21



Drawing arcs (Fig 22)

An arc is defined as a part of a circle. In AutoCAD LT, it can be drawn using the Arc command. You can invoke the arc command from the draw toolbar; AutoCAD provides eleven different options to draw an arc. The default option for drawing an arc is the 3-point option. Other options can be invoked by entering the appropriate letter in the command window or by right-clicking and choosing the appropriate option from the fly out. The option that was used last will be displayed in the draw panel. The various options to draw the arcs are discussed.

Draw arcs by specifying start, center, and end (Fig 23)

You can create an arc using a start point, center, and a third point that determines the endpoint. The distance between the start point and the center determines the radius. The endpoint is determined by a line from the center that passes through the third point.

Using different options, you can specify either the start point first or the center point first.

Fig 22

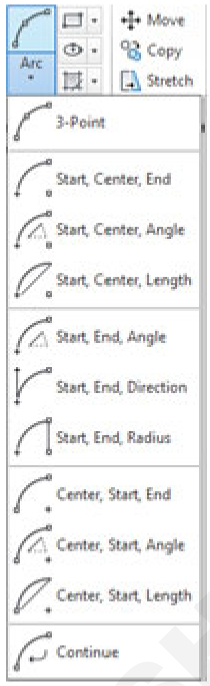
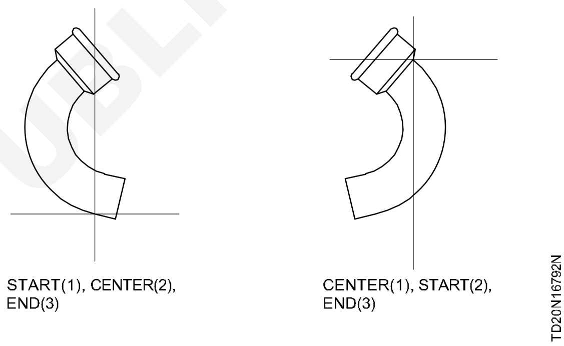


Fig 23



Draw arcs by specifying start, center, angle (Fig 24)

You can create an arc using a start point, center, and an included angle.

The distance between the start point and the center determines the radius. The other end of the arc is determined by specifying an included angle that uses the center of the arc as the vertex.

Using different options, you can specify either the start point first or the center point first.

The included angle determines the endpoint of the arc. Use the start , end, angle method when you know both endpoints but cannot snap to a center point.

Draw arcs by specifying start, center, length (Fig 25)

You can create an arc using a start point, center, and the length of a chord. The distance between the start point and the center determines the radius. The other end of the arc is determined by specifying the length of a chord between the start point and the endpoint of the arc. Using different options, you can specify either the start point first or the center point first.

Fig 24

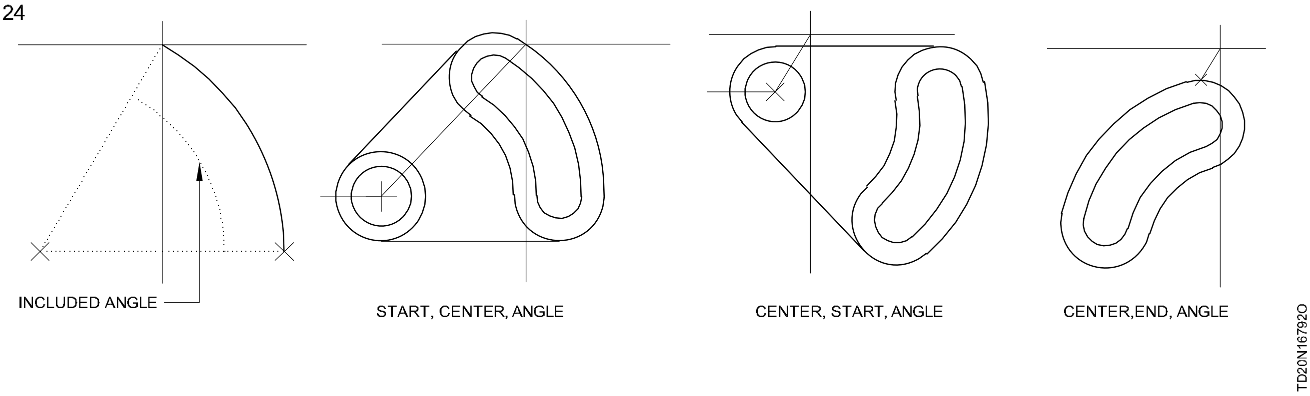
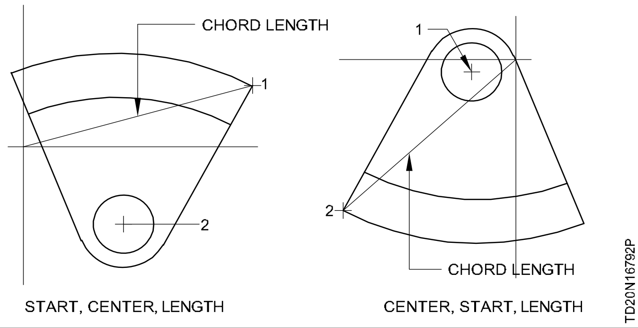


Fig 25

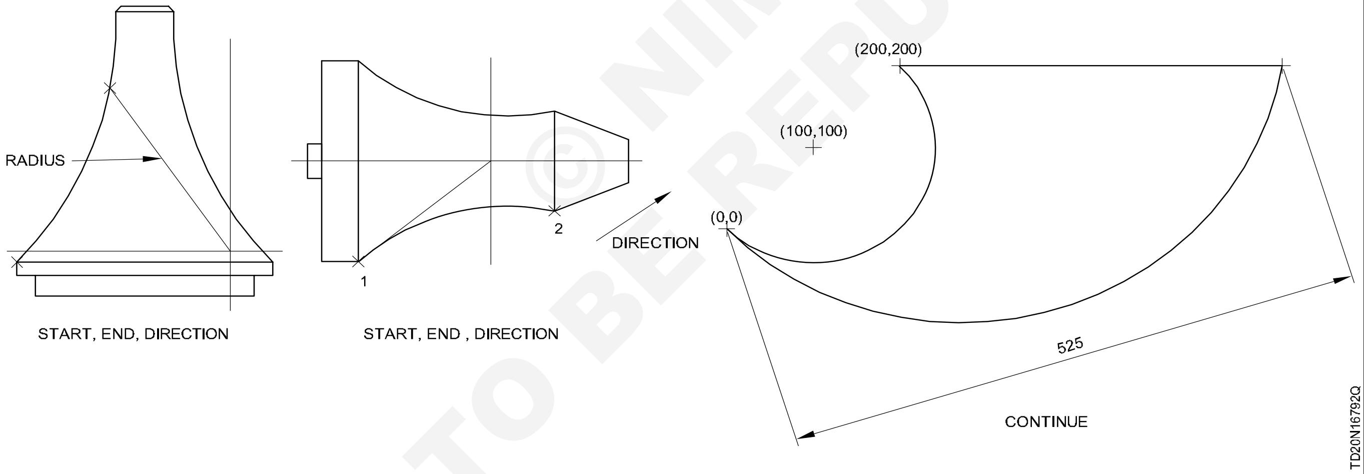


Draw arcs by specifying start, end, radius (Fig 26)

You can create an arc using a start point, endpoint, and a radius

The direction of the bulge of the arc is determined by the order in which you specify its endpoints. You can specify the radius either by entering it or by specifying a point at the desired radius distance.

Fig 26



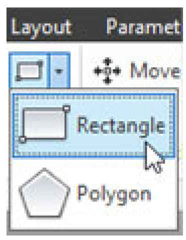
The continue option

With the option, you can continue drawing an arc from a previously drawn arc or line. Continue from the draw menu the start point and direction of the arc will be taken from the endpoint and ending direction of the previous line or arc. When this option is used to draw arcs, each successive arc will be tangent to the previous one. Most often, this option is used to draw arcs tangent to a previously drawn line.

Drawing rectangles (Fig 27)

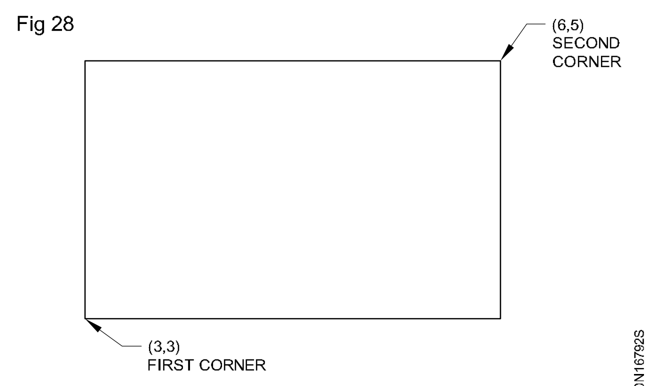
A rectangle is drawn by choosing the rectangle tool see Fig 26, from the draw panel. In AutoCAD, you can draw rectangles by specifying two opposite corners of the rectangle, by specifying the area and the size of one of the sides, or by specifying the dimensions of the rectangle. All these methods of drawing rectangles are discussed next.

Fig 27



Drawing rectangles of specifying two opposite corners

On invoking the rectangle tool, you will be prompted to specify the first corner of the rectangle. Enter the coordinates of the first corner or specify the start point by using the mouse. The first corner can be any one of the four corners. Next, you will be prompted to specify the other corner. Specify the diagonally opposite corner by entering the coordinates or by using the left mouse button, as shown in Fig 28.



DRAWING A RECTANGLE BY SPECIFYING TWO OPPOSITE CORNERS

Drawing rectangles by specifying the area and one side

To draw a rectangle by specifying its area and the length of one of the sides , first specify the start point. Next, invoke the shortcut menu by right-clicking and then choose the area option. Next, specify the parameters; the rectangle is drawn. Following is the prompt sequence to draw a rectangle whose start point is at 3, 3 has area 15 units, and length 5 units:

Choose the rectangle tool from the draw panel.

Specify first corner point or [Chamfer/Elecvation/Fillet/ Thickness/Width]: 3, 3

specify other corner point or]Area/Dimensions/Rotation]: A

Enter area of rectangle in current units <100.000>:15

Calculate rectangle dimensions based on [Length/Width] <Length>: L

Enter rectangle length <10.0000>:5

In the above case, the area and length of the rectangle were entered. The system automatically calculates the width of the rectangle by using the following formula:

Area of rectangle = Length x Width

Width = Area of rectangle/Length

Width = 15/5

Width = 3 units

Drawing rectangles by specifying their dimensions

You can also draw a rectangle by specifying its dimensions. This can be done by choosing the dimensions option from the shortcut menu at the specify other corner point or [Area/Dimensions/Rotation] prompt and entering the length and width of the rectangle. The prompt sequence for drawing a rectangle at 3, 3 with a length of 5 units and width of 3 units is given next.

Choose the rectangle tool from the draw panel.

Specify first corner point or [chamfer/Elevation/Fillet/ Thickness/Width]: 3, 3

Specify other corner point or [Area/Dimensions/Rotation]: D

Specify length for rectangles <0.0000>:5

Specify width for rectangles <0.0000>:3

Specify other corner point or [Area/Dimensions/Rotation]: click on the screen to specify the orientation of rectangle.

Here, you are allowed to choose any one of the four locations for placing the rectangle. You can move the cursor to see the four quadrants. Depending on the location of the cursor, the corner point that is specified first holds the position of either the lower left corner, the lower right corner, the upper right corner, or the upper left corner. After deciding the position, you can click to place the rectangle.

Drawing rectangle at an angle (Fig 29)

You can also draw a rectangle at an angle. This can be done by choosing the rotate option from the shortcut menu, at the specify other corner point or [Area/Dimensions/ Rotation] prompt and entering the rotation angle. After entering the rotation angle, you can continue sizing the rectangle using any one of the above discussed methods. The prompt sequence for drawing a rectangle at an angle of 45-degree is:

Choose the rectangle tool from the draw panel

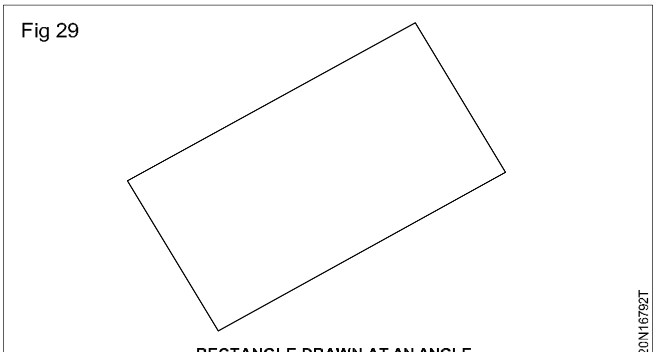
Specify first corner point or [Chamfer/Elevation/Fillet/ Thickness/Width]: Select a point as lower left corner location.

Specify other corner point or [Area/Dimensions/Rotation]: R

Specify rotation angle or [Pick points] <current>: 45

Specify other corner point or [Area/Dimensions/Rotation]: Select a diagonally opposite point

While specifying the other corner point, you can place the rectangle in any of the four quadrants. Move the cursor in different quadrants and then select a point in the quadrant in which you need to draw the rectangle. Fig 29 shows a rectangle drawn at an angle of 45 degree.



RECTANGLE DRAWN AT AN ANGLE

You can also set some of the parameters of a rectangle before specifying the start point. These parameters are the options in the command prompt and are discussed next.

Chamfer

The chamfer option is used to create a chamfer, which is an angled corner, by specifying the chamfer distances, see Fig 30. The chamfer is created at all four corners. You can give two different chamfer values to create an unequal chamfer.

Choose the rectangle tool from the draw panel.

Specify first corner point or [Chamfer/Elevation/Fillet/Thickness/Width]: C

Specify first chamfer distance for rectangles <0.0000>: Enter a value, d1.

Specify second chamfer distance for rectangles <0.0000>: Enter a value, d2.

Specify first corner point or [Chamfer /Elevation/Fillet/Thickness/Width]: Select a point as lower left corner.

Specify other corner point or [Area/Dimensions/Rotation]: Select a point as upper right corner.

Fillets (Fig 30)

The fillet option is used to create a filleted rectangle, see figure. You can specify the required fillet radius. The following is the prompt sequence for specifying the fillet:

Choose the Rectangle tool from the Draw panel

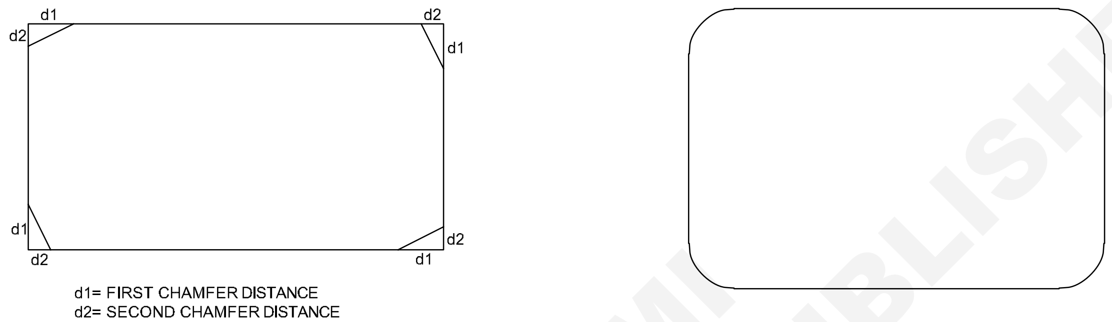
Specify first corner point or [Chamfer/Elevation/Fillet/Thickness/Width]: F

Specify fillet radius for rectangles <0.0000>: Enter a value.

Specify first corner point or [Chamfer/Elevation/Fillet/Thickness/Width]: Select a point as lower left corner.

Specify other corner point or [Area/Dimensions/Rotation]: Select a point as upper right corner.

Fig 30



DRAWING A RECTANGLE WITH CHAMFERS

DRAWING A RECTANGLE WITH FILLETS

TD20N16792U

Width

The width option is used to create a rectangle whose line segments have some specified width, as shown in Fig 31.

Specify first corner point or [Chamfer/Elevation/Fillet/Thickness/Width]: W

Specify line width for rectangles <0.0000>: Enter a value

Fig 31

DRAWING A RECTANGLE OF SPECIFIED WIDTH

TD20N16792V

Drawing ellipses

Ribbon: Home > Draw > Ellipse drop-down Toolbar: Draw > Ellipse Tool palettes: Draw > Ellipse command: Ellipse

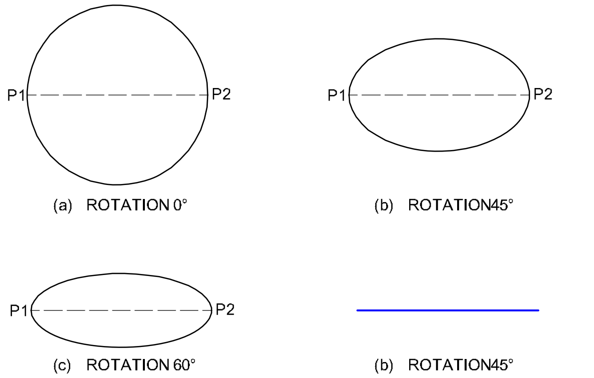
If you cut a cone by a cutting plane at an angle and view the cone perpendicular to the cutting plane, the shape created is called an ellipse. An ellipse can be created by using different tools available in the ellipse drop-down of the draw panel, refer to Fig 32. In AutoCAD, you can create a true ellipse, also known as a NURBS-based (Non-Uniform rational Bezier spline) ellipse. A true ellipse has center and quadrant points. If you select it, grips (small blue squares) will be displayed at the center and quadrant points of the ellipse. If you move one of the grips located on the perimeter of the ellipse, the size of the ellipse will be change.

Drawing ellipse using the center option

To draw an ellipse by specifying its center point, endpoint of one axis, and length of other axis, chooses the center tool from the draw panel; you will be prompted to specify the center of the ellipse. The center of an ellipse is defined as the point of intersection of the major and minor axes. Specify the center point or enter coordinates; you will be prompted to specify the endpoint. Specify the endpoint of the major or minor axis; you will be prompted to specify the distance of the other axis. Specify the distance; the ellipse will be draw.

After specifying the endpoint of one axis, you can enter R at the specify distance to other axis or [Rotation] prompt to specify the rotation angle around the major axis. In this case, the first axis specified is considered as the major axis. On specifying the rotation angle, the ellipse will be drawn at an angle with respect to the major axis. Note that the rotation angle should range between 0 and 89.4 - degree. Fig 32 shows the ellipse created at different rotation angles.

Fig 32



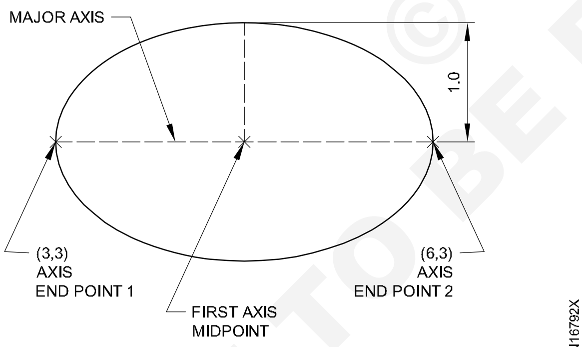
ROTATION ABOUT THE MAJOR AXIS

Drawing an ellipse by specifying its axis and endpoint

To draw the ellipse by specifying one of its axes and the endpoint of the other axis, choose the axis, end tool from the draw panel; you will be prompted to specify the axis end point. Specify the first endpoint of one axis of the ellipse; you will be prompted to specify the other endpoint of the axis. Specify the other endpoint of the axis. Now, you can specify the distance to other axis from the center or specify the rotation the specified axis.

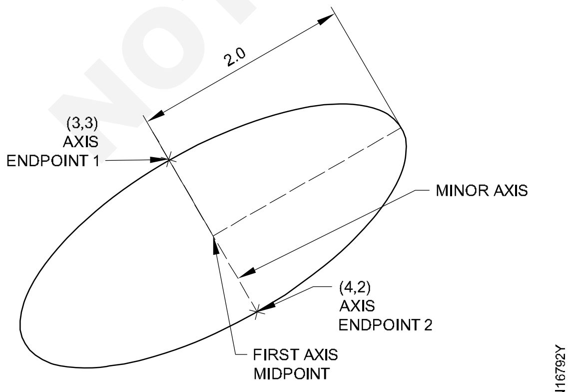
Fig 33 shows and ellipse with one endpoint of the axis located at (3, 3), the other at (6, 3), and the distance to the other axis as 1 unit. Fig 34 shown an ellipse with one endpoint of the axis located at (3, 3), the other at (4, 2), and the distance to the other axis as 2 units.

Fig 33



DRAWING AN ELLIPSE USING THE AXIS AND ENPOINT OPTION

Fig 34



DRAWING AN ELLIPSE USING THE AXIS AND ENPOINT OPTION

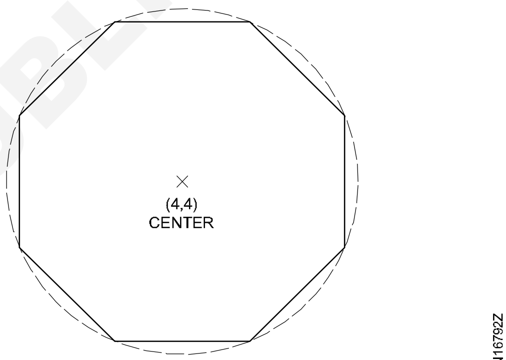
Drawing regular polygons

A regular polygon is a closed geometric entity with equal sides. The number of sides of a polygon varies from 3 to 1024. For example, a triangle is a three-sided polygon and a pentagon is a five-sided polygon. To draw a regular 2D polygon, choose the polygon tool from the draw panel; you will be prompted to specify the number of sides. Type the number of sides and press enter. Now, you can draw the polygon by specifying the length of an edge or by specifying the center of the polygon. Both these methods are discussed next.

Drawing a polygon by specifying the center of polygon

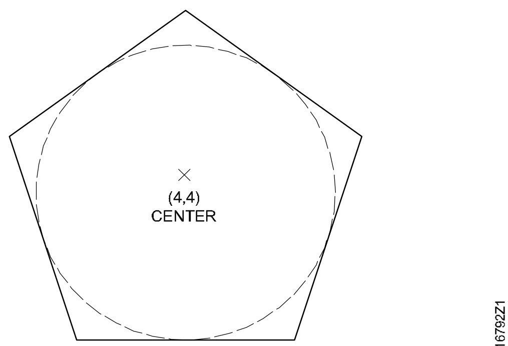
After you specify the number of sides and press enter, you will be prompted to specify the center of polygon. Specify the center point; you will be prompted to specify whether the polygon to be drawn is inscribed in a circle or circumscribed about an imaginary circle. A polygon is said to be inscribed when it is drawn inside an imaginary circle such that the vertices of the polygon touch the circle, (Fig 35) whereas, a polygon is said to be circumscribed when it is drawn outside the imaginary circle such that the sides of the polygon are tangent to the circle, (Fig 36).

Fig 35



INSCRIBED OCTAGON

Fig 36

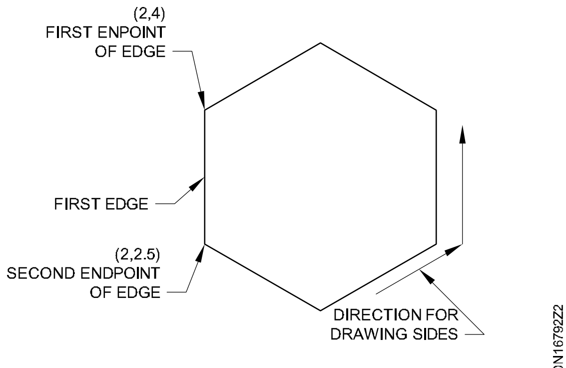


CIRCUMSCRIBED PENTAGON

Drawing a polygon by specifying and edge

To draw a polygon by specifying the length of an edge, you need to type E at the specify center of polygon or [Edge] command prompt and press Enter, Next, specify the first and second endpoints of the edge in succession; the polygon will be drawn in counter clockwise direction, as shown in Fig 37.

Fig 37



DRAWING A POLYGON (HEXAGON) USING THE EDGE OPTION

Fig 38

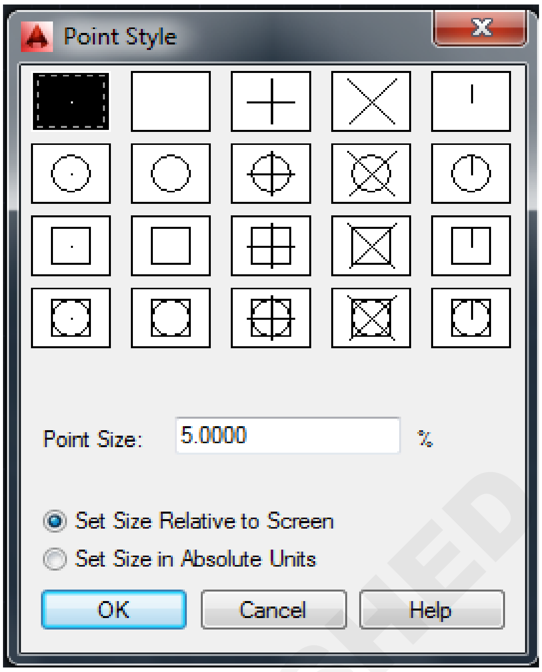
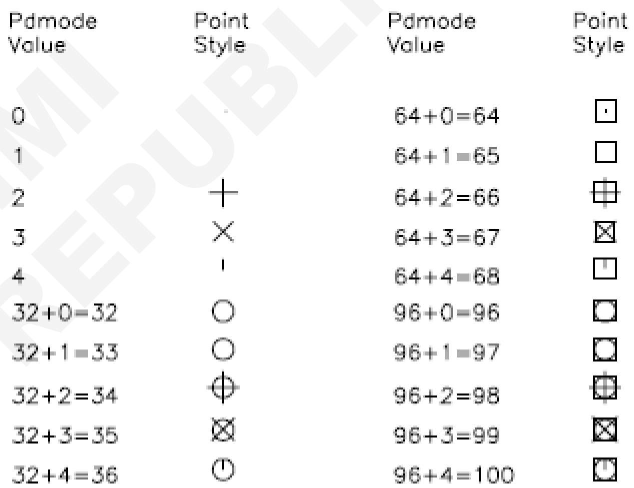


Fig 39



Placing points

A point is one of the basic drawing objects and is specified as a dot (a period). A point is defined as a geometric object that has no dimension and properties, except location. However, in AutoCAD, you can control the size and appearance (style) of a point. You will first learn to change the point style and size and then about various methods to place a point..

Changing the point style and point size

The point style and size can be set from the Point Style dialog box shown in Fig 38 There are twenty combinations of point types. Choose Format > Point Style from the Menu Browser; the Point Style dialog box will be invoked you can choose a point style in this dialog box, which is indicated by highlighting that particular point style. Next, choose the OK button. Now all the points will be drawn in the selected style, until you change it to a new style. The type of point drawn is stored in the PDMODE (Point Display MODE) system variable. You can change the point style by entering a numeric value in the PDMODE variable. Fig 39 shows the PDMODE Values for different point type areas.

Function keys

Objective: At the end of this lesson you shall be able to
- **describe various functional key in AutoCAD.**

Function and control keys

You can also use the function and control keys to change the status of the coordinate display, Snap, Ortho, Osnap, tablet, screen, isometric planes, running Object Snap, Grid, Polar, and Object tracking. Some of the keys can be used in drafting settings. The following is a list of function Table 1 and control keys.

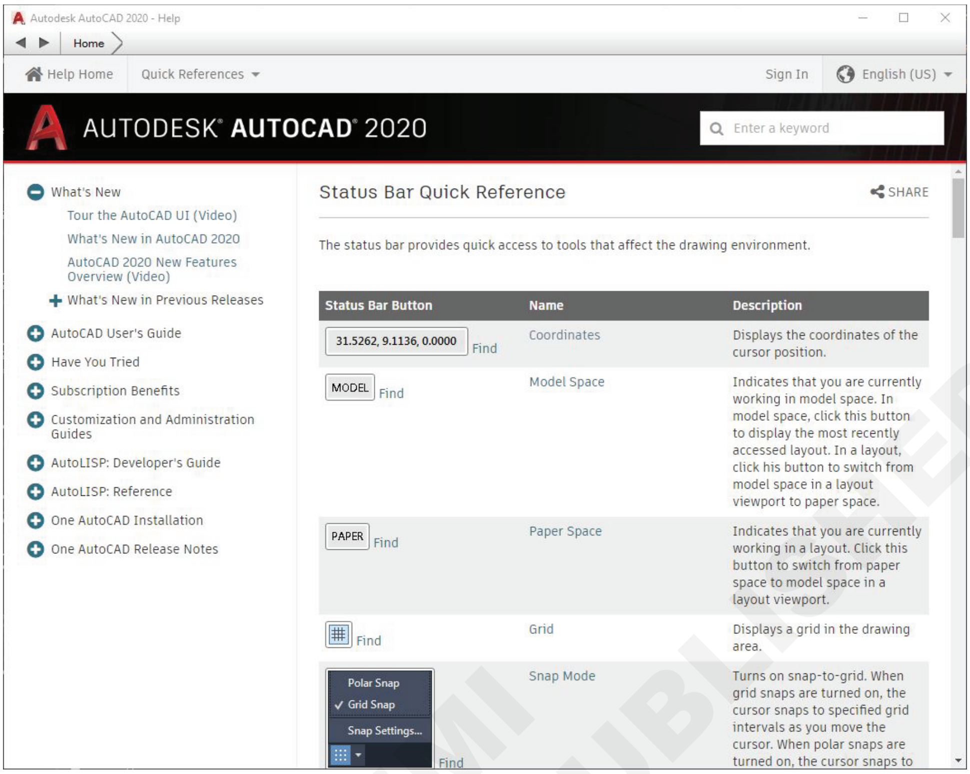
Table 1

Key	Description
FI	Help
F2	Graphics Screen/AutoCAD Text Window
F3	Osnap On/Off (CTRL+F)
F4	3D Snap On/Off
F5	Isoplane top/right/left (CTRL+E)
F6	Dynamic UCS On/Off
F7	Grid On/Off (CTRL+G)
F8	Ortho On/Off (CTRL+L)
F9	Snap On/Off (CTRL+B)
F10	Polar tracking On/Off
F11	Object Snap tracking
FI2	Dynamic Input On/Off

Help (FI)

To display Help for a menu, open the menu, and then press FI. Pressing F1 or entering 'help while a command is active displays Help for that command. Choosing the Help button in a dialog box displays Help for that dialog box.(Fig 1)

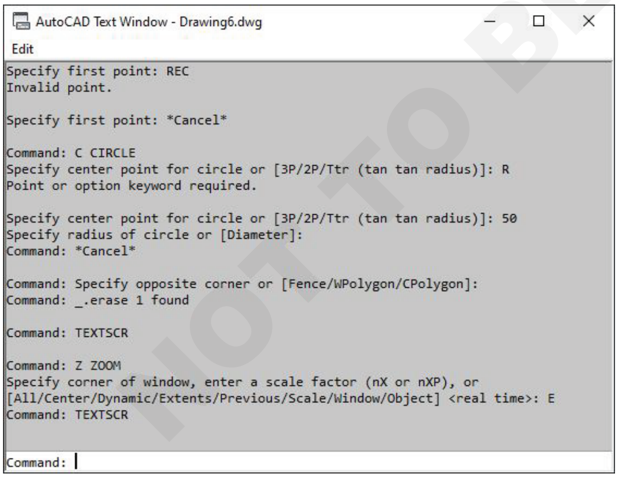
Fig 1



Graphics Screen/AutoCAD Text Window (F2)

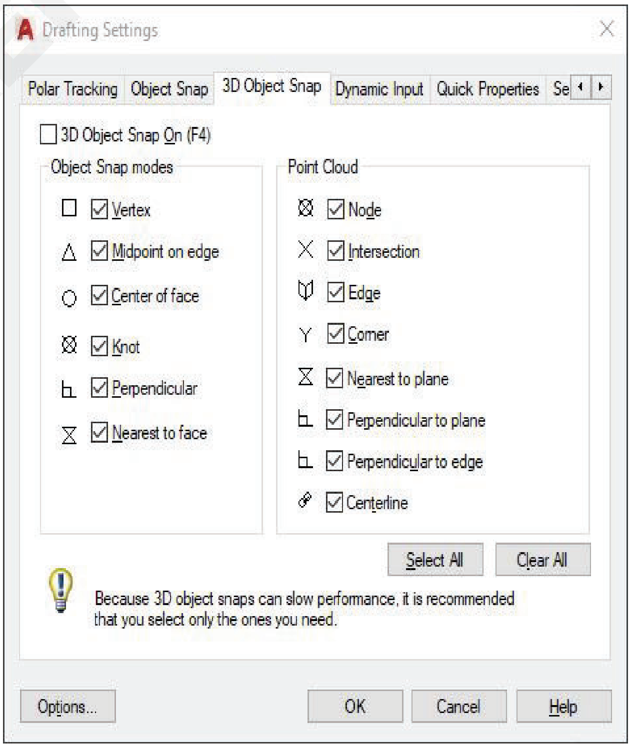
Whenever, you want to view the command inputs and its steps, press F2 to display the text window. The Text window is displayed in front of the AUTOCAD drawing area. Similarly, use the TEXTSCR command to switch from the drawing area to text window. (Fig 2)

Fig 2



Osnap On/Off (F3): Using object snaps is a quick way to locate an exact position on an object without having to know the coordinate or draw construction lines. For example, you can use an object snap to draw a line to the center of a circle or to the midpoint of a polyline segment. You can specify an object snap whenever you are prompted for a point. You can use an object snap once in the middle of a command, or you can set running object snaps.

Fig 3



3D Object Snap (F4)

Object snaps function the same way in 3D as they do in 2D with the exception that they can optionally be projected.By default, the Z-value of an object snap location is determined by the object's 3D location. However, if you work with object snaps on the plan or top view of a 3D model, a constant Z-value is more useful.

If you turn on the OSNAPZ system variable, all object snaps are projected onto the XYplane of the current UCS or, if ELEV is set to a non-zero value, onto a plane parallel to XY plane at the specified elevation.

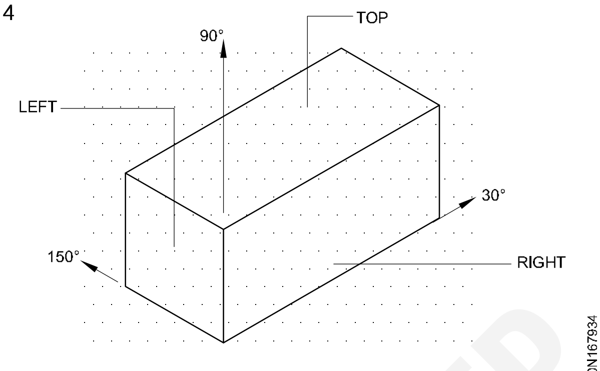
Note: When you create or modify objects, make sure that you know whether OSNAPZ is turned on or off. There is no visual reminder, and you can get unexpected results.

Isoplane top/right/left (F5) (Fig 4)

The isometric plane affects the cursor movement keys only when Snap mode is on and the snap style is Isometric. If the snap style is Isometric, Ortho mode uses the appropriate axis pair even if Snap mode is off. The current isometric plane also determines the orientation of isometric circles drawn by ELLIPSE. You can cycle through the isometric planes by pressing F5.

Isometric drawings simulate a 3D object from a particular viewpoint by aligning along three major axes. By setting the Isometric Snap/Grid, you can easily align objects along one of three isometric planes; however, although the isometric drawing appears to be 3D, it is actually a 2D representation. Therefore, you cannot expect to extract 3D distances and areas, display objects from different viewpoints, or remove hidden lines automatically. If the snap angle is 0, the axes of the isometric planes are 30 degrees, 90 degrees, and 150 degrees. Once you set the snap style to Isometric, you can work on any of three planes, each with an associated pair of axes:

Fig 4



ISOMETRIC - THREE PLANES

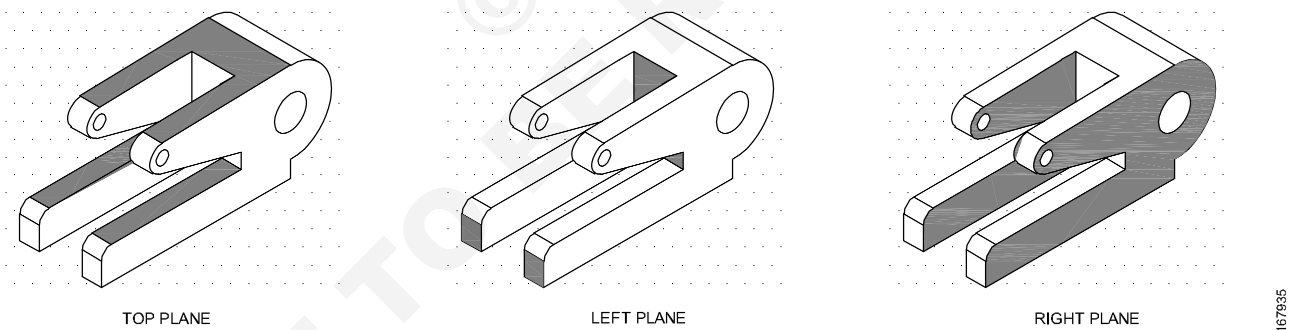
Left: Aligns snap and grid along 90- and 150-degree axes.

Top: Aligns snap and grid along 30- and 150-degree axes.

Right: Aligns snap and grid along 30- and 90-degree axes

Choosing one of the three isometric planes causes Ortho and the crosshairs to be aligned along the corresponding isometric axes. For example, when Ortho is on, the points you specify align along the simulated plane you are drawing on. Therefore, you can draw the top plane, switch to she left plane to draw another side, and switch to the right plane to complete the drawing. (Fig 5)

Fig 5



ISOMETRIC - THREE PLANES

Dynamic UCS On/Off (F6)

With the dynamic UCS turned on, you can temporarily and automatically align the XYplane of the UCS with a plane on a 3D solid while creating objects.During a command, you can align the UCS dynamically by moving your cursor over a planar face of a 3D solid rather than using the UCS command. After you finish the command, the UCS returns to its previous location and orientation.

Grid (F7)

The grid is a rectangular pattern of dots that extends over the area you specify as the grid limits. Using the grid is similar to placing a sheet of grid paper under a drawing. The grid helps you align objects and visualize the distances between them. The grid is not plotted. If you zoom in or out of your drawing, you may need to adjust grid spacing to be more appropriate for the new magnification. You can turn on or off the grids by pressing F7 key.

To display a grid and set grid spacing (Fig 6)

1 From the Tools menu, choose Drafting Settings.

2 In the Drafting Settings dialog box, Snap & Grid tab, select Grid On to display the grid.

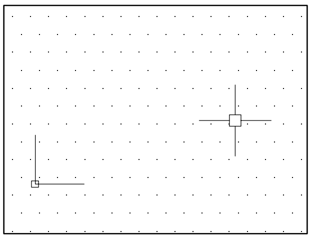
3 Under Snap Type and Style, make sure Grid Snap and Rectangular Snap are selected.

4 For grid X Spacing, enter the horizontal grid spacing in units.

5 To use the same value for vertical grid spacing, press ENTER. Otherwise, enter a new value for grid Y Spacing.

6 Choose OK.

Fig 6



GRID DISPLAY

Orthomode (F8)

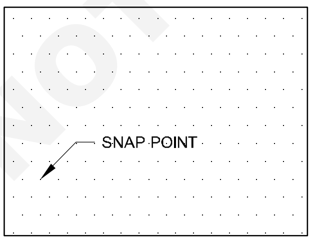
You can use Ortho mode to restrict the cursor to the horizontal or vertical axis. The orthogonal alignment depends on the current snap angle, UCS, or isometric grid and snap setting. Ortho works with commands that require you to specify a second point. You can turn Ortho on and off at any time during drawing and editing by pressing F8 key. You can use Ortho not only to establish vertical or horizontal alignments but also to enforce parallelism or to create regular offsets from existing objects. By imposing orthogonal constraints, you can draw more quickly. For example, you can create a series of perpendicular lines by turning on Ortho mode before you start. Because the lines are constrained to be parallel to the horizontal and vertical axes, you know that the lines are perpendicular.

Setting Snap (F9)

The snap is used to set increments for the cursor movement. While moving the cursor, it is sometimes difficult to position a point accurately. The SNAP command allows you to set up an invisible grid, Fig 7 that allows the cursor to move in fixed increments from one snap point to another. The snap points are the points where the invisible snap lines intersect.

The snap spacing is independent of the grid spacing, and so the two can have equal or different values. You generally set snap to an increment of the grid setting, for example, Snap=5 and Grid=10.

Fig 7

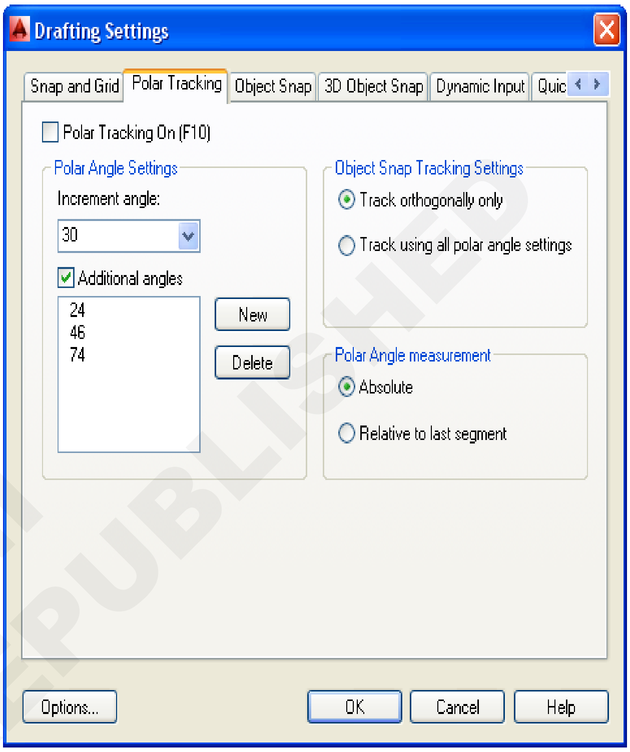


SNAP POINT

Polar Tracking (F10)

Polar tracking is used to locate points on an angular alignment path. Polar tracking can be selected by choosing the POLAR button on the status bar, by using the function key F10, or by selecting the Polar Tracking On (F10) check box in the Polar Tracking tab of the Drafting Settings dialog box (Fig 8). Polar Tracking constrains the movement of the cursor along a path that is based on the polar angle settings. For example, if the Increment angle list box value is set to 15-degree in the Polar Angle Settings area, the cursor will move along the alignment paths that are multiples of 15-degree (0, 15, 30, 45, 60, and so on) and a tooltip will display a distance and angle.

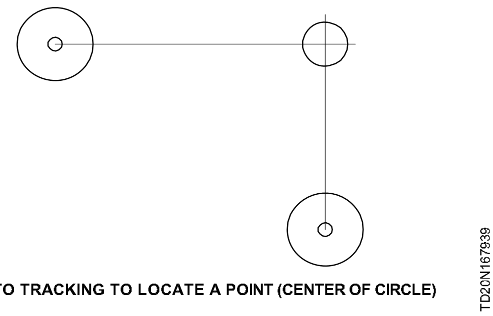
Fig 8



Object Snap Tracking (F11)

Object Snap Tracking tracks the movement of the cursor along the alignment paths based on the Object Snap points (running osnaps) that are selected in the Object Snap tab of the Drafting Settings dialog box. Selecting the Object Snap Tracking On (F11) check box in the Object Snap tab of the Drafting Settings dialog box, choosing the OTRACK button on the status bar, or using the function key FI 1 sets the object snap tracking on.The direction of the path is determined by the motion of the cursor or the point you select on an object. For example, if you want to draw a circle whose center is located in the line with the center of two existing circles (Fig 9), you can use AutoTracking. First select Center in the Object Snap tab of the Drafting Settings dialog box to set Center as the running osnap. Also choose the Osnap button on the status bar to the On position. Now, you can activate the object tracking by choosing the OTRACK button on the status bar.

Fig 9

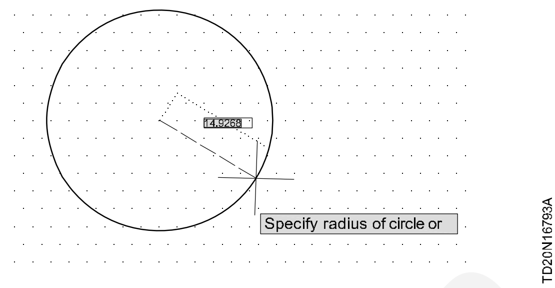


USING AUTO TRACKING TO LOCATE A POINT (CENTER OF CIRCLE)

Dynamic input mode (F12) (Fig 10)

As mentioned earlier, turning the Dynamic Input mode on allows you to enter the commands through a pointer input and the dimensions using the dimensional input. When this mode is turned on, all the prompts are available at the tooltip as dynamic prompts and you can select the command options through dynamic prompt. The settings for Dynamic Input are done through the Dynamic Input tab of Drafting Settings dialog box. Invoke the Drafting Settings dialog box by selecting the Drafting Settings option from the Tools drop-down list.

Fig 10



Object properties

Objectives: At the end of this lesson you shall be able to
- **brief the methods of applying color in AutoCAD**
- **describe the various line types in in AutoCAD.**

Color

You can enter the color number (1 through 255) or the color name (the names for the first seven colors). For example, you can specify the color red by entering the ACI number 1 or the ACI name red.You can also enter bylayer or byblock. If you enter byblock, all new objects are drawn in the default color (white or black, depending on your background color) until they are grouped into a block.

You can set the color of an object either by its layer or by specifying its color explicitly, independent of its layer.

- Assigning colors by layer makes it easy to identify each layer within your drawing.
- Assigning colors explicitly provides additional distinctions between objects on the same layer.
- You can use a variety of color palettes when assigning color to objects, including
- AutoCAD Color Index (ACI)
- True Color
- PANTONE ® Colors
- RALTM Classic and RAL Design color books
- DIC ® Color Guide
- Colors from imported color books.

AutoCAD Color Index (ACI) Palettes

Specifies a color from the AutoCAD Color Index. If you hover over a color, the number of the color and its red, green, blue value are displayed below the palette. Click a color to select it, or enter the color number or name in the Color box.

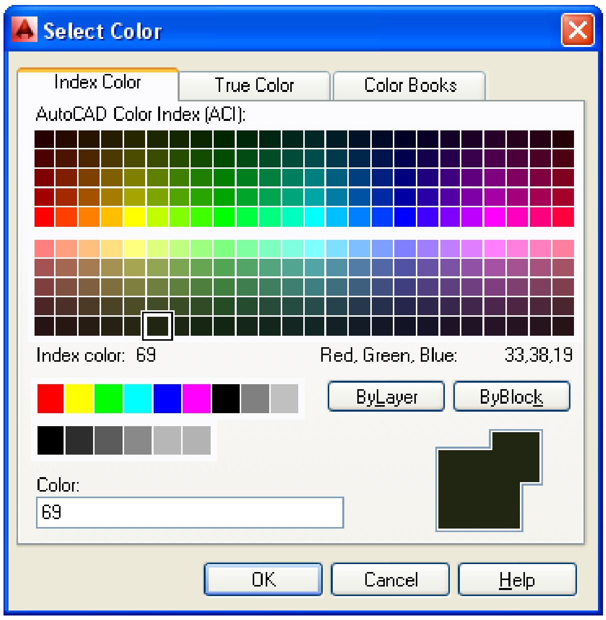
To set ACI Color for all new objects

1 Click Home tab > Properties panel > Object Color list.

2 From the list, choose a color, or choose Select Colors to display the Select Color dialog box, and do one of the following:

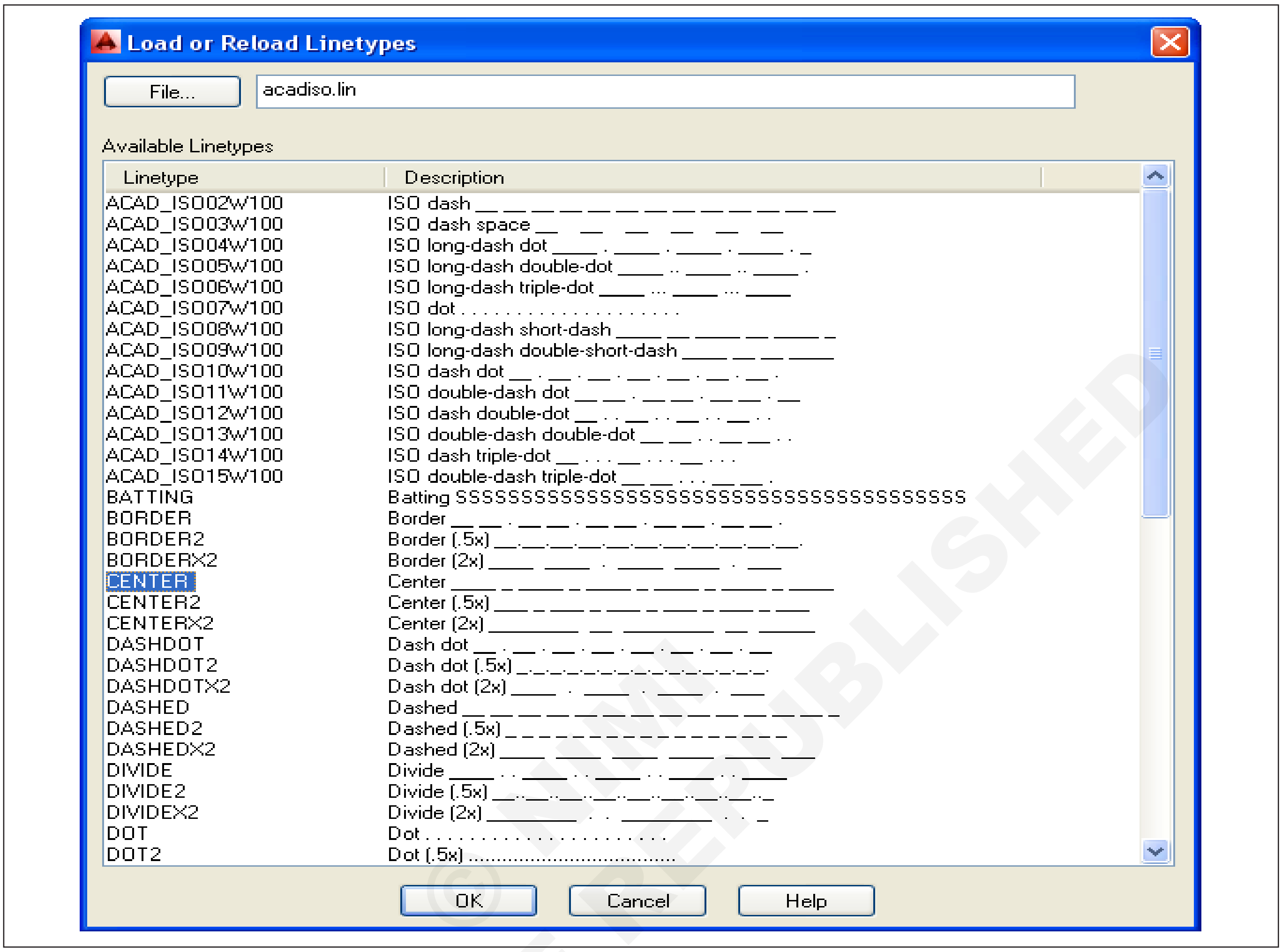
  - On the Index Color tab, click a color, or in the Color box, enter a color name or number.
  - On the Index Color tab, click ByLayer to draw new objects in the color assigned to the current layer.
  - For creating blocks only, on the Index Color tab, click ByBlock to draw objects that, when they are combined into a block and inserted into the drawing, display the current color.

Specifies color settings using the 255 AutoCAD Color Index (ACI) colors.



Line type

A linetype is a repeating pattern of dashes, dots, and blank spaces displayed in a line or a curve. You assign linetypes to objects either by layer or by specifying the linetype explicitly, independent of layers.



All objects are created using the current linetype, which is displayed in the Properties palette and the Home tab, Properties panel of the ribbon when no objects are selected.

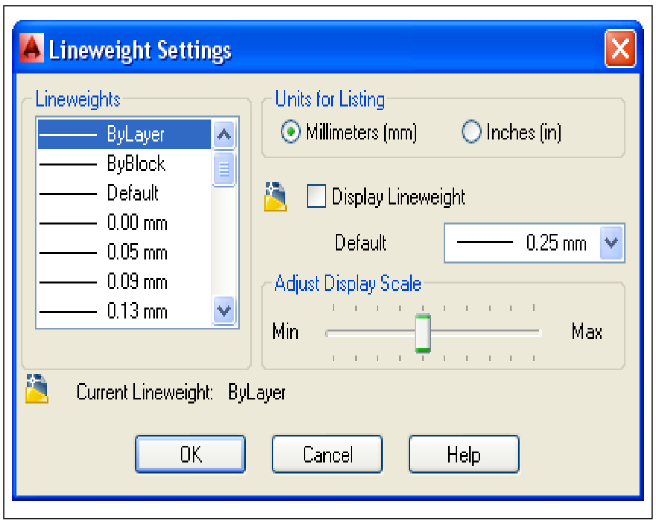
- If the current linetype is set to ByLayer, objects are created with the linetype assigned to the current layer.
- If the current linetype is set to ByBlock, objects are created using the Continuous linetype, a solid linetype without embedded spaces, until the objects are combined into a block definition. When the block is inserted into the drawing, it displays the current linetype for those objects.

Line weight

Lineweight is a property assigned to graphical objects, hatches, leader lines, and dimension geometry that results in thicker, darker lines.

Lineweights can be turned on and off in a drawing, and are displayed differently in model space than in a paper space layout

- In model space, a lineweight of 0 is displayed as one pixel wide, and other lineweights use a proportional pixel width. Lineweight display in model space does not change with the zoom factor. For example, a lineweight value that is represented by a width of four pixels is always displayed using four pixels regardless of how far you zoom in.
- In a paper space layout, lineweights are displayed using real-world units, and lineweight display changes with the zoom factor.



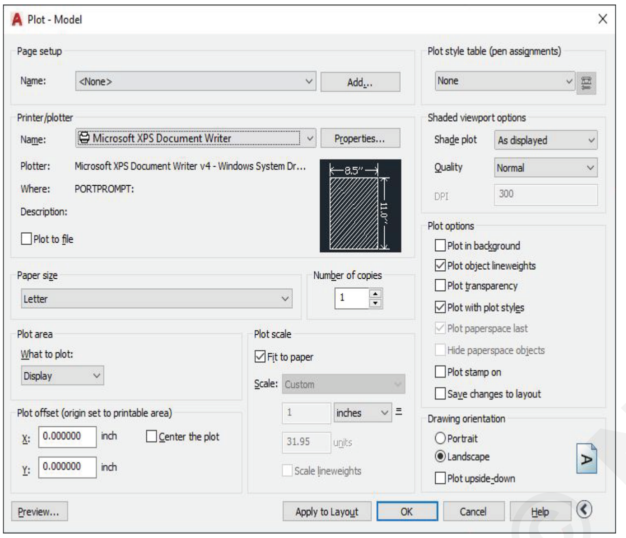
Plotting & Export

Objectives: At the end of this lesson you shall be able to
- **explain the methods of setting plotter specifications and plot drawings**
- **describe the method of creating layouts.**

Plotting Drawings

After creating a drawing of an architectural plan or a mechanical component, you may need to send it to the client or have a hard copy for reference. To do so, you need to plot the drawing. To plot the drawing, choose Plot from the Quick Access Toolbar; the Plot-Model dialog box will be displayed. If the dialog box is not expanded by default, choose the More Options button at the lower right corner of the dialog box to expand it, as shown in Fig 1.

Fig 1



Page Setup

Displays a list of any named and saved page setups in the drawing. You can base the current page setup on a named page setup saved in the drawing, or you can create a new named page setup based on the current settings in the Plot dialog box by clicking Add.

Printer/Plotter

Specifies a configured plotting device to use when plotting layouts. If the selected plotter doesn't support the layout's selected paper size, a warning is displayed and you can select the plotter's default paper size or a custom paper size.

Paper Size

Displays standard paper sizes that are available for the selected plotting device. If no plotter is selected, the full standard paper size list is displayed and available for selection.

Plot Area

Specifies the portion of the drawing to be plotted. Under What to Plot, you can select an area of the drawing to be plotted.

Window

Plots any portion of the drawing that you specify. When you select Window, the Window button becomes available. Click the Window button to use the pointing device to specify the two corners of the area to be plotted, or enter coordinate values.

See Preview and you can see the drawing. Click Plot and the drawing will be plotted.

Viewports

The options available depend on whether you are configuring model space viewports (on the Model layout) or layout viewports (on a named (paper space) layout).

Tabs for Model space viewports: New Viewports-Model Space, Named Viewports-Model Space.

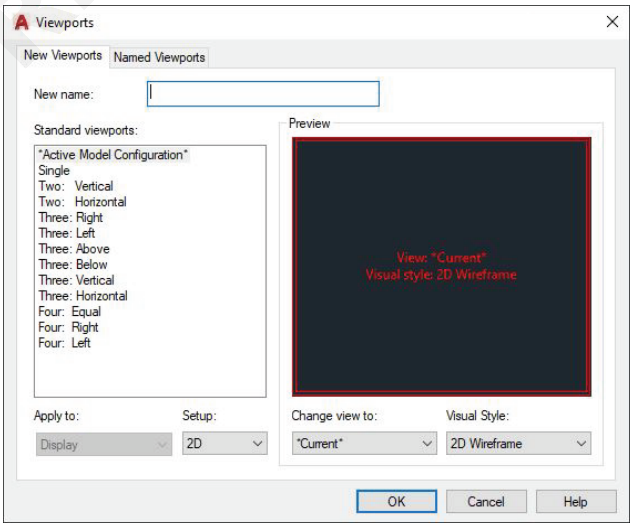
Tabs for layout viewports: New Viewports-Layouts, Named Viewports-Layouts

It is important to create layout viewports on their own layer. When you are ready to output your drawing, you can turn off the layer and output the layout without the boundaries of the layout viewports.

Viewports for 3D object viewing (Fig 2)

Command : VIEWPORTS Following screen appear

Fig 2



New Name

Specifies a name for the new model space viewport configuration. If you do not enter a name, the viewport configuration is applied but not saved. If a viewport configuration is not saved, it cannot be used in a layout.

Standard Viewports

Lists and sets the standard viewport configurations, including CURRENT, which is the current configuration.

Preview

Displays a preview of the viewport configuration you select and the default views assigned to each individual viewport in the configuration.

Apply To

Applies the model space viewport configuration to the entire display or to the current viewport.

Display: Applies the viewport configuration to the entire Model tab display.

Current Viewport: Applies the viewport configuration to the current viewport only.

Setup

Specifies either a 2D or a 3D setup. When you select 2D, the new viewport configuration is initially created with the current view in all.of the viewports. When you select 3D, a set of standard orthogonal - 3D views is applied to the viewports in the configuration.

Change View To

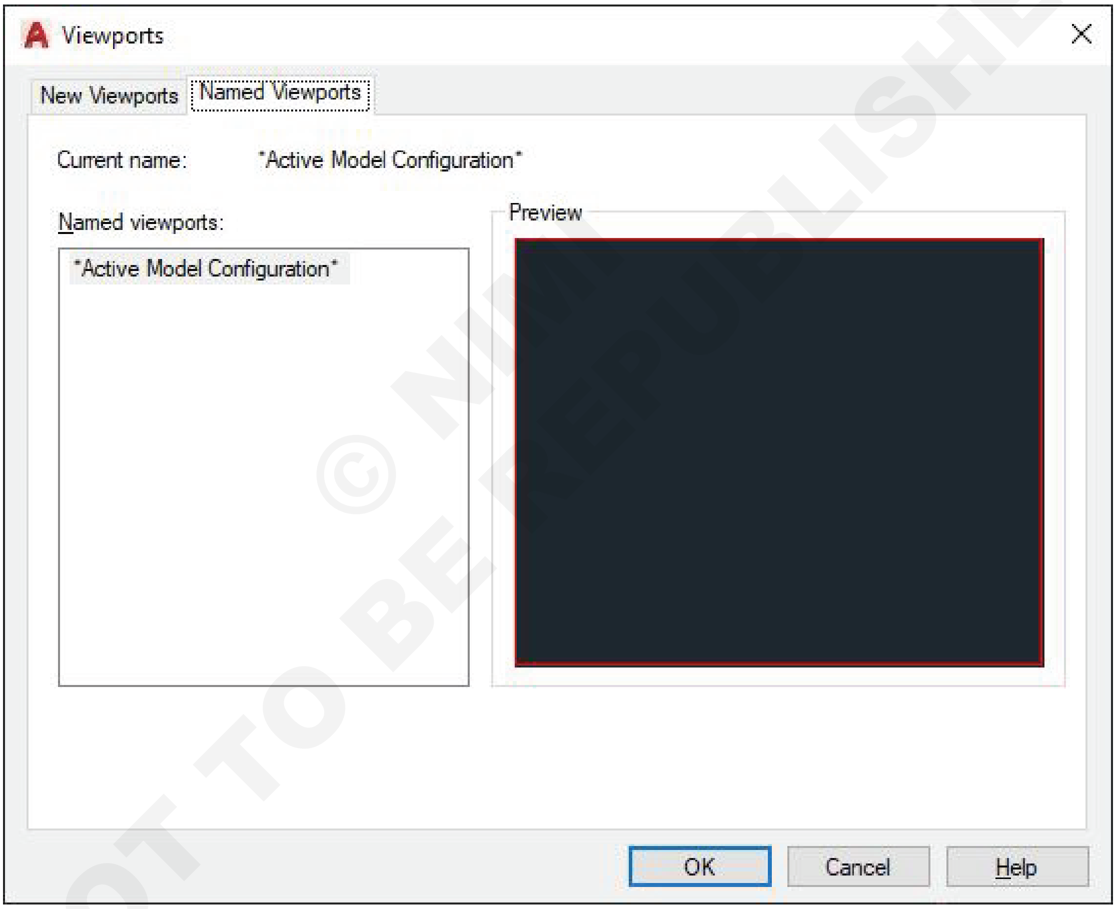
Replaces the view in the selected viewport with the view you select from the list. You can choose a named view, or if you have selected 3D setup, you can select from the list of standard views. Use the Visual Style (not in Auto CAD LT)

Applies a visual style to the viewport. All available visual styles are displayed.

New Viewports Tab-Layouts (viewports Dialog Box)

Lists all saved model viewport configurations in the drawing.

Fig 3



Current Name

Displays the name of the current viewport configuration,

New Viewports Tab-Layouts (viewports Dialog Box)

Standard Viewports

Displays a list of standard viewport configurations and configures layout viewports.

Preview

Displays a preview of the viewport configuration you select and the default views assigned to each individual viewport in the configuration.

Viewport Spacing

Specifies the spacing you want to apply between the layout viewports you are configuring.

Setup

Specifies either a 2D or a 3D setup. When you select 2D, the new viewport configuration is initially created with the current view in all of the viewports. When you select 3D, a set of standard orthogonal 3D views is applied to the viewports in the configuration.

Change View To: Replaces the view in the selected viewport with the view you select from the list. You can choose a named view, or if you have selected 3D setup, you can select from the list of standard views.

Named Viewports Tab-Layouts (viewports Dialog Box) (Fig 3)

Displays any saved and named model space viewport configurations for you to use in the current layout. You cannot save and name a layout viewport configuration.

-VPORTS - Layout Viewports

Creates multiple viewports in layouts (paper space).

The number and arrangement of layout viewports and their associated settings are called viewport configurations.

The following prompts are displayed.

Corner of viewport

Specifies the location of the one corner of the rectangular viewport.

Opposite corner. Specifies the location of the opposite, diagonal corner of the rectangular viewport.

On

Turns on a viewport, making it active and making its objects visible.

Off

Turns off a viewport. When a viewport is off, its objects are not displayed, and you cannot make that viewport current.

Fit

Creates one viewport that fills the available display area. The actual size of the viewport depends on the dimensions of the paper space view.

Shadeplot

Specifies how viewports in layouts are plotted.

As Displayed: Plots the same way it is displayed

Wireframe: Plots wireframe regardless of display

Hidden: Plots with hidden lines removed regardless of display

Visual Styles: Plots using the specified visual style; all visual styles in the drawing are listed as options whether in use or not (not available in AutoCAD LT)

Render Presets: Plots using the specified render preset; all render presets are listed as options (not available in AutoCAD LT)

Lock

Locks the current viewport. Prevents changes from being made to the associated view.

Object

Creates a nonrectangular layout viewport from a closed polyline, ellipse, spline, region, or circle. The polyline you specify must be closed and contain at least three vertices. It can be self-intersecting, and it can contain arcs as well as line segments.

Polygonal

Creates a nonrectangular layout viewport defined by a series of line and arc segments.

The descriptions of the Next Point, Arc, Close, Length, and Undo options match the descriptions of the corresponding options in the PLINE command.

Restore

Restores a previously saved viewport configuration.

Layer (Fig 4)

Resets layer property overrides for the selected viewport to their global layer properties.

2 Divides the current viewport in half.

3 Divides the current viewport into three viewports.

Horizontal and Vertical split the area into thirds. The other options create one large viewport in half the available area and two smaller ones in the other half Above, Below, Left, and Right specify where the larger viewport is placed.

4 Divides the current viewport into four viewports of equal size.

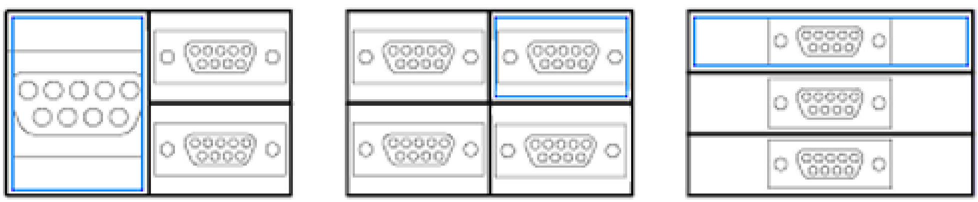
About Model Space Viewports

In model space, you can split the drawing area into one or more rectangular areas called model space viewports.

Viewports are areas that display different views of your model. In large or complex drawings, displaying different views reduces the time needed to zoom or pan in a single view. Errors that you might miss in one view might be visible in another.

The Fig 4 show several model space viewport configurations. You can save and restore viewport configurations by name with the VPORTS command.

Fig 4



When you display multiple viewports, the one that is highlighted with a blue rectangle is called the current viewport.

- Commands that control the view, such as panning and zooming, apply only to the current viewport.
- Commands that create or modify an object are started in the current viewport, but the results apply to the model and can be visible in other viewports.
- You can start a command in one viewport and finish it in a different viewport.
- You can make any viewport the current one by clicking in It.

Model space viewports should not be confused with layout viewports, which are available only in paper space and are used to arrange views of your drawing on a sheet.

Modify Model Space Viewports (Fig 5)

You can modify the size, shape, and number of model space viewports in a viewport configuration:

- Choose from several viewport configurations by clicking the [+] or [-] control in the top-left corner of a viewport.
- Drag the boundaries of viewports to adjust their size.
- Press CTRL while dragging viewport boundaries to display the green splitter bar and create new viewports. Alternatively, you can drag the outermost splitter controls.
- Drag a viewport boundary onto another boundary to remove a viewport.

After you create a new viewport, you might want to maximize and center the view by double-clicking the mouse wheel to perform a zoom extents.

To Create a New Layout Viewport

1 Click View tab > Viewports panel > New.

2 In the Viewports dialog box, New Viewports tab, under Standard Viewports, select Single.

3 Click to specify one corner of the new layout viewport

4 Click to specify the opposite corner.

A new layout viewport object is available and displays a default view. To adjust the view, double-click within the layout viewport to access model space.

To Restore a Saved Viewport Arrangement

Viewport arrangements can be saved only in model space, but you can restore saved viewport arrangements either in model space, as model space viewports, or in paper space, as layout viewports.

1 Click View tab > Viewports panel > Named.

2 In the Viewports dialog box, Named Viewports tab, select the name of the viewport configuration from the list.

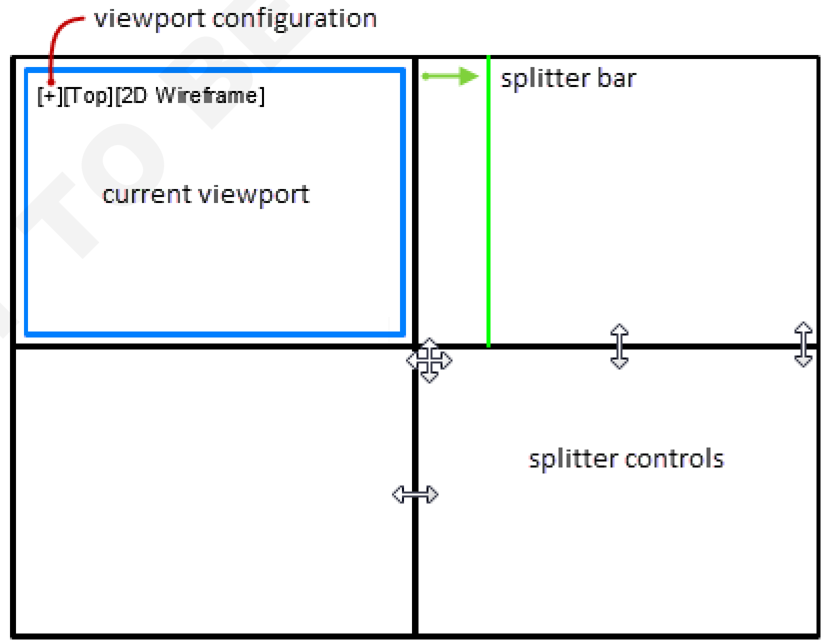
To Delete a Saved Viewport Arrangement

1 Click View tab > Viewports panel > Named.

2 In the Viewports dialog box, Named Viewports tab, select the name of the viewport configuration you want to delete.

3 Press the DELETE key.

Fig 5



ACAD Shorcuts

Arc	A
Area	AA
Array	AR
Break	BR
Chamfer	CHA
Circle	C
Copy	CP , CO
Ddmodify	MO
Dimangular	DAN, DIMANG
Dimbaseline	DBA
Dimdiameter	DD1 , DIMDIA
Dimcontinue	DCO , DIMCONT
Dimradius	DRA , DIMRAD
Dist	DI
Donut	DO
Dsviewer	AV
Erase	E
Extend	EX
Fillet	F
Hatch	H
Layer	LA
Line	L

Linetype	LT
List	LI , LS
Ltscale	LTS
Matchprop	MA
Mirror	MI
Move	M
Offset	O
Osnap	OS
Pline	PL
Point	PO
Polygon	POL
Purge	PU
Rectangle	REC
Regen	RE
Rotate	RO
Scale	SC
Setvar	SET
Snap	SN
Solid	SO
Strech	S
Trim	TR
Zoom	Z

ACAD System Variables

APERTURE	SETS OBJECT SNAP TARGET HEIGHT, IN PIXELS
AREA	STORES THE LAST AREA COMPUTED BY AREA, LIST, DBLIST
AUNITS	SETS UNIT FOR ANGLES
AUTOSNAP	CONTROLS DISPLAY OF AUTOSNAP MARKER , AUTOSNAP MAGNET ON/OF
BLIPMODE	CONTROLS WETHER MARKER BLIPS ARE VISIBLE
CECOLOR	SETS COLOR OF NEW OBJECTS
CELTSCALE	SETS CURRENT GLOBAL LINE TYPE SCALE FOR OBJECTS.
CELTYPE	SETS LINETYPE FOR NEW OBJECTS
CELWEIGHT	SETS THE LINEWEIGHT FOR NEW OBJECTS
CHAMFERA	SETS THE CHAMFER DISTANCE
CHAMFERS	SETS THE SECOND CHAMFER DISTANCE
CHAMFERC	SETS THE CHAMFER LENGTH
CHAMFERD	SETS THE CHAMFER ANGLE
CHAMMODE	SETS THE INPUT METHOD BY WHICH ACAD CREATES CHAMFER
CIRCLERAD	SETS THE DEFAULT CIRCLE RADIUS
CLAYER	SETS THE CURRENT LAYER
COORDS	CONTRLS WHEN COORDINATES ARE UPDATED ON STATUS LINE
CURSORSIZE	DETERMINES THE SIZE OF CROSSHAIR AS A PERCENTAGE OF SCREEN SIZE
DIMASZ	CONTROLS THE SIZE OF DIMENSION LINE AND LEADER LINE ARROW HEADS
DIMEXE	SPECIFY HOW FAR TO EXTEND EXTENSION LINE BEYOND DIMENSION LINE

DIMEXO	SPECIFY HOW FAR EXTENSION LINES ARE OFFSET FROM ORGIN POINT
DIMGAP	SETS THE DISTANCE AROUND DIMENSION TEXT FOR DIMENSION LINE
DIMTIH	CONTROL POSITION OF TEXT INSIDE EXTENSION LINE
DIMTSZ	SPECIFY THE SIZE OF OBLIQUE STROKE DRAWN INSTEAD OF ARROW HEAD
DISTANCE	STORES DISTANCE COMPUTED BY DIST
DONUTID	SETS THE DEFAULT INSIDE DIAMETER FOR DONUT
DONUTOD	SETS THE DEFAULT OUTSIDE DIAMETER FOR DONUT
EDGEMODE	CONRTOLS HOW TRIM AND EXTEND DETERMINE CUTTING/BOUNDARY EDGE
FILEDIA	SUPRESSES DISPLAY OF FILE DIALOGUE BOXES
FILLETRAD	STORES THE CURRENT FILLET RADIUS
GRIDMODE	SPECIFY WHETHER THE GRID MODE IS ON OR OF
GRIDUNIT	SPECIFY GRID SPACING FOR CURRENT VIEW PORT
HIGHLIGHT	CONTROLS OBJECT LIGHTING
HPANG	SPECIFY HATCH PATTERN ANGLE
HPDOUBLE	SPECIFY HATCH PATTERN DOUBLING FOR USER DEFINED PATTERNS
HPNAME	SETS A DEFAULT HATCH PATTERN NAME
HPSCALE	SPECIFY THE HATCH PATTERN SCALE FACTOR
HPSPACE	SPECIFY THE HATCH PATTERN LINE SPACING FOR USER DEFINED PATTERN
LASTANGLE	STORES THE END ANGLE OF THE LAST ARC ENTERED
LASTPOINT	STORES THE COORDINATE OF THE LAST POINT ENTERED
LIMCHECK	CONTROLS CREATION OF OBJECTS BEYOND DRAWING LIMITS
LIMMAX	STORES THE UPPER RIGHT DRAWING LIMIT FOR CURRENT DRAWING SPACE
LIMMIN	STORES THE LOWER LEFT DRAWING LIMIT FOR CURRENT DRAWING SPACE
LTSCALE	STORES THE GLOBAL LINETYPE SCALE FACTOR
LUNITS	SETS LINEAR UNITS
LUPREC	SETS NUMBER OF DECIMALS FOR LINEAR UNITS
MIRRTEXT	CONTROLS HOW MIRROR REFLECTS TEXT
OFFSETDIST	SETS THE DEFAULT OFFSET DISTANCE
ORTHOMODE	CONSTRAINS CURSOR MOVEMENT TO THE PERPENDICULAR
OSMODE	SETS RUNNING OBJECT SNAPMODE USING BIT-CODES
PDMODE	CONTROLS HOW POINT OBJECTS ARE DISPLAYED
PDSIZE	SETS TEH DISPLAY SIZE FOR POINT OBJECTS
PLINEWID	STORES THE DEFAULT POLYLINE WIDTH
POLYSIDES	SETS THE DEFAULT NUMBER OF SIDES FOR POLYGON
QTEXTMODE	CONTROL HOW TEXT IS DISPLAYED
REGENMODE	CONTROLS AUOMATIC REGENERATION OF DRAWING
SAVENAME	SAVES FILE NAME
SNAPANG	SETS SNAP AND GRID ROTATION ANGLE FOR THE CURRENT VIEW PORT
SNAPBASE	SETS THE SNAP AND GRID ORGIN POINT FOR THE CURRENT VIEW PORT
SNAPMODE	TURNS SNAPMODE ON AND OF
SNAPUNIT	SETS SPANSPACING FOR CURRENT VIEW PORT
TEXTSIZE	SETS DEFAULT TEXT HEIGHT FOR NEW TEXT OBJECTS WITH CURRENT STYLE
TRACEWID	SETS DEFAULT TRACE WIDTH
TRIMMODE	CONTROLS WHETHER ACAD TRIMS SELECTED EDGES FOR CHAMFER/FILLETS
UNITMODE	CONTROLS UNIT FORMAT FOR UNITS

Introduction to creo

Objectives: At the end of this lesson you shall be able to
- **list the hardware requirements to install creo software**
- **Install Creo software**
- **launch Creo and interface the main components;**
- **set up a Working Directory, file names and conventions;**
- **brief the File menu, Manage File options, and different file types;**
- **open a Creo object and manage files;**
- **brief what is a Session is and how to Manage Session.**

Introduction

CreoTM Parametric is one of the most powerful Computer-Aided Design (CAD), Computer-Aided Analysis and Computer-Aided Manufacture(CAM) software packages available in the world today. It is the flagship of a family of other software products, developed by PTC Corporation, for engineering design and product development, also including CreoTMDirect, CreoTM Simulate, CreoTM Layout and others. The main applications are in mechanical, product design, aerospace, construction,shipbuilding and other industries.

CreoTM Parametric (or Creo) was previously known as Pro/EngineerTM and WildfireTM. The core of the software contains a variety of tools for the creation, validation and communication of complex three-dimensional (3D) objects as parts and assemblies. In addition, there are integrated applications that associate directly with the 3D model geometry and support the development of engineering drawings, mould design, NC machine simulation, sheet metal design, piping and wiring, harness design, structural strength, thermal and CFD analyses, kinematic and dynamic analyses, feasibility and optimisation studies, and others.

Creo Installation

To install the software following hardware is required

Hardware requirements

The minimum computer system requirements are listed below:

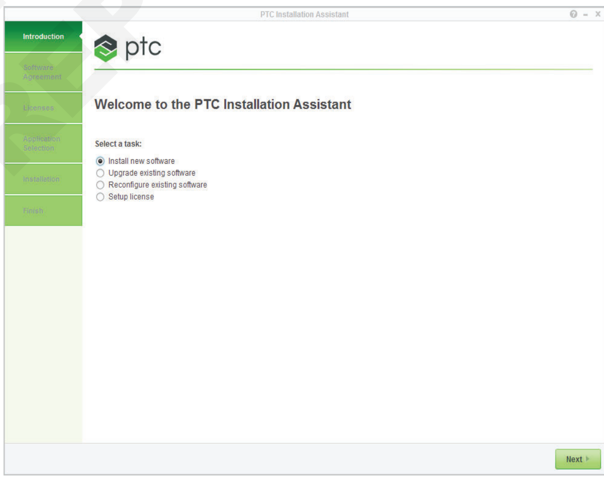
- Operating system: Windows 7 or Windows 10 64-bit editions;
- Processor: Core i5 or higher;
- RAM: More than 4Gb;
- Monitor: Minimum resolution — 1280 × 1024 pixels with 32-bit colour;
- Hard or solid state disk memory available: More than 10Gb;
- Dedicated Graphics card: 1Gb minimum;
- Microsoft compatible 3 button (2 button and a wheel) mouse;
- Internet connection.

Creo installation software package has an automated SETUP.EXE executable file that guides the installation process of all main applications and defaults.

Run SETUP.EXE file with administrative privileges to start the installation process. The PTC Installation Assistant appears as shown in Fig 1. The Introduction, Software Agreement and Licenses are all self-explanatory sections relevant to a specific user.

By pressing the Install icon (Fig 2) the installation starts and in the end a shortcut Creo icon is created automatically on the PC desktop.

Fig 1



By pressing the Customize icon, the user can access a submenu with Options (Fig 3). Make sure that Modelcheck and Mold Analysis are selected. These options are needed for the Mould Design chapter.

> **Note: Depending on the license agreement some of the applications are automatically selected (ticked) and those with no license are unavailable (dimmed) for installation. By pressing the Install icon (Fig 2), the installation starts and in the end a shortcut Creo icon is created automatically on the PC desktop.**

Fig 2

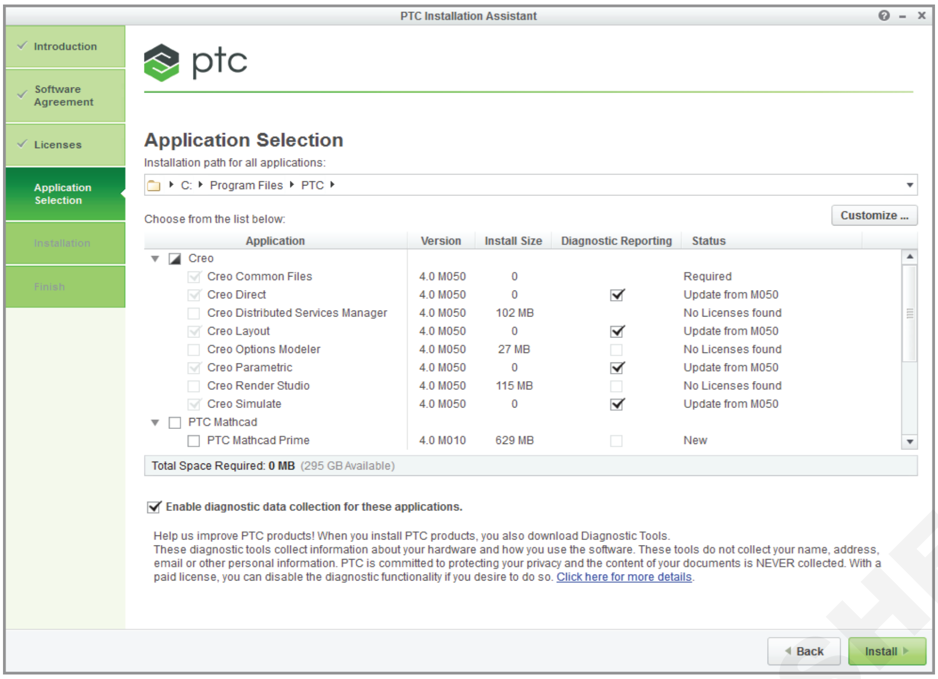
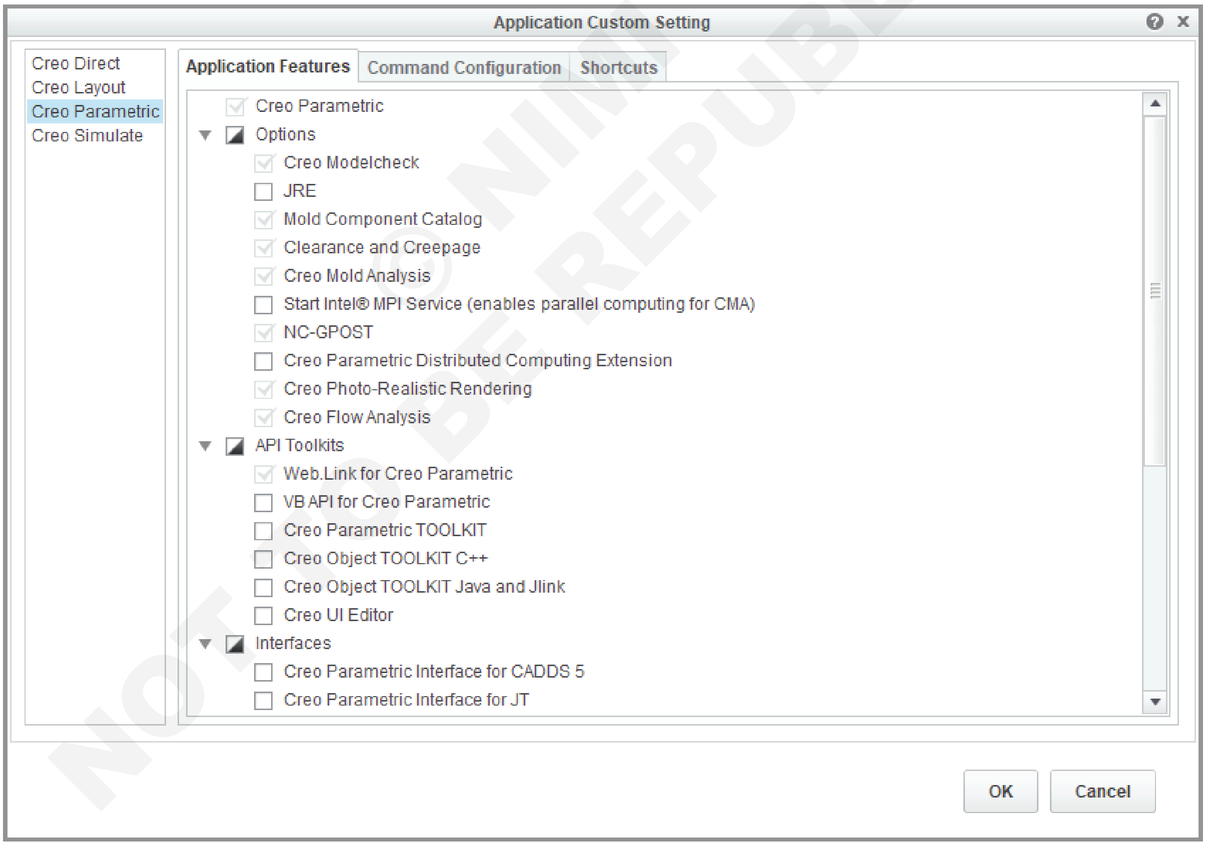


Fig 3



Getting Started and Creo Interface

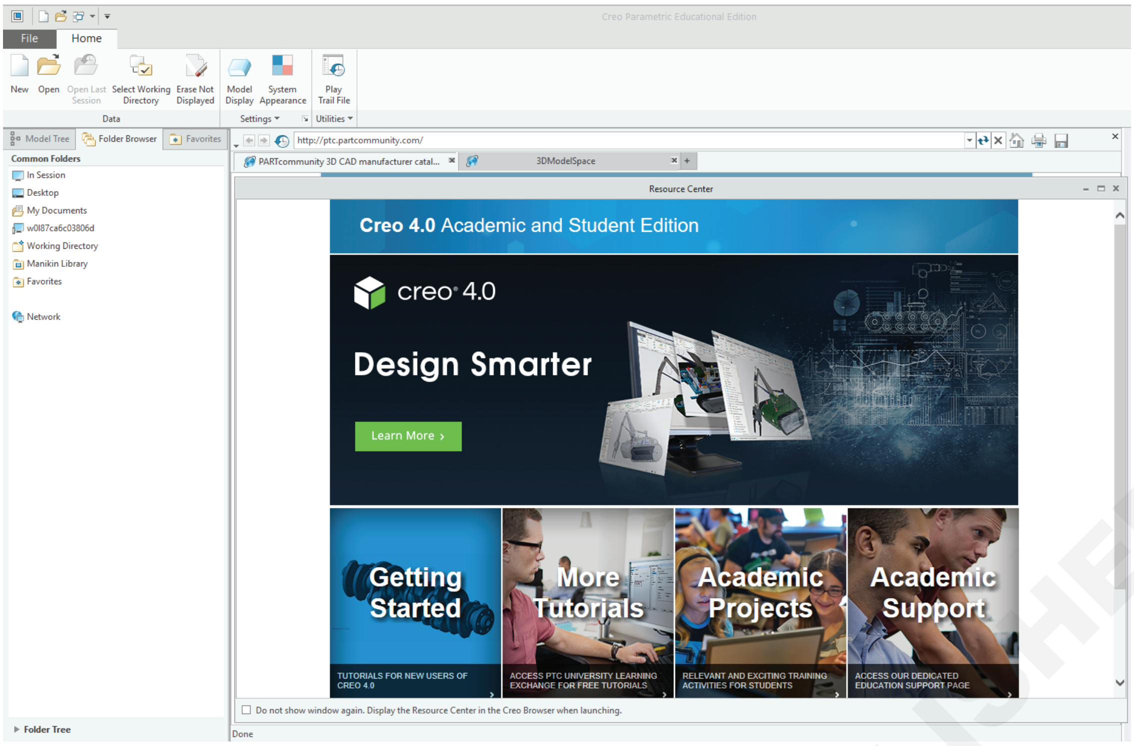
Launching the software

Launch Creo by LMB (left mouse button) double-clicking on the shortcut Creo Parametric icon in the PC desktop. Another way to launch the software is to press on the Windows Start icon, and then to activate Creo Parametric from the (Creo user interface DOC) PTC applications list.

The initial screen after starting the software is shown in Figure 4. It consists of File and Home ribbon menu options, Folder Browser tab, and Favourites tab to run a Web browser. Close the

Resource Centre window and the Web browser link http://ptc.partcommunity.com/.

Fig 4

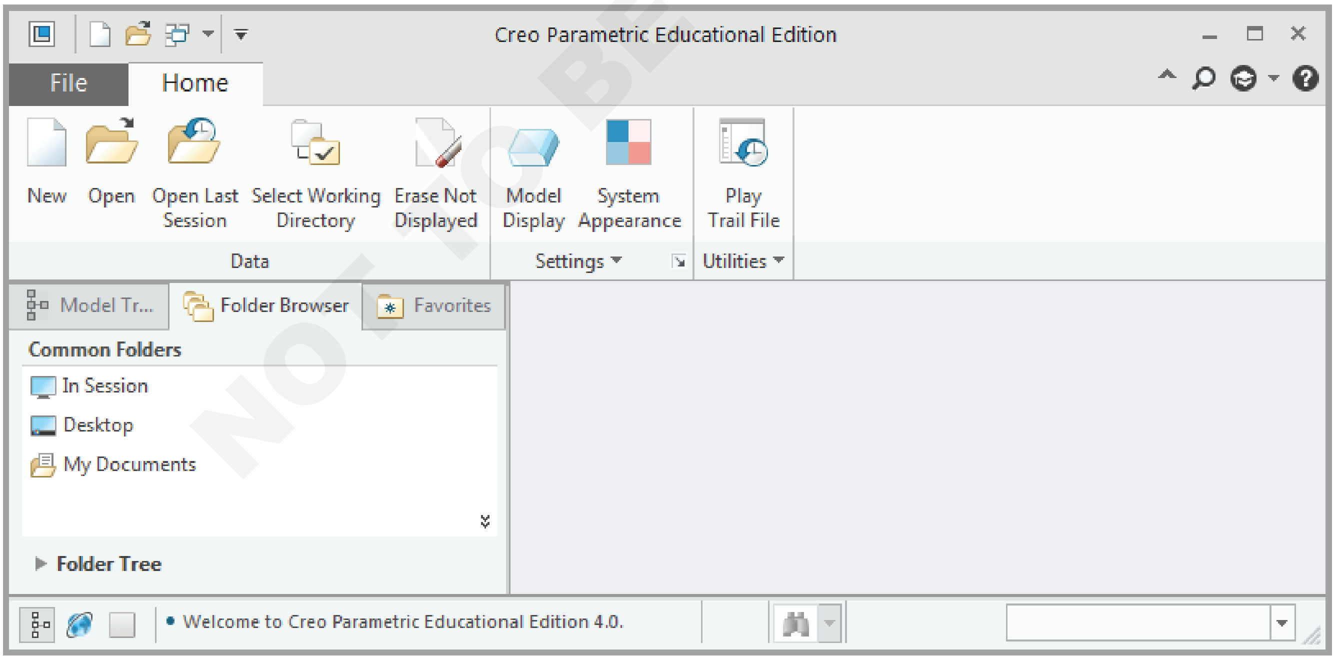


The main interface at Start Up looks as shown in Figure 5. The main areas and functions of this interface are as follows: The Folder Browser tab (Fig 5), located on the left of the screen next to the Model Tree tab, lists the folders on the computer or network.Browse the folders and view their contents in the Folder Browser.

The Favourites tab is a multi-functional Web browser embedded in Creo. It displays models and tutorials from PTC.com and other websites. A preview window appears in the centre of the screen after the launch.

The Folder Browser icons ( ) are located at the bottom left corner of the main window. Click (LMB) on the icons to either show or hide the current folder and Web browsers.

Fig 5



The Graphics area is a large grey area where the 3D model geometry is visible when a model is opened This is the main area where the model geometry is developed and displayed. The user can rotate, move and zoom the model.

Select working directory

The Working Directory is a directory or a folder in the PC disc memory (hard drive, solid state disc memory, USB memory stick, network, etc.) where all models (objects) belonging to a project are stored. The user can set any existing folder with Read/Write permissions as a Working Directory. It is very important to do this immediately after starting Creo or switching to a new project. If a Working Directory is not selected, the software will save or open an object from the default directory (usually in C: drive). For example, an assembly model containing several parts will not open those models that are located outside the Working Directory. The missing part name will appear in red in the assembly Model Tree.

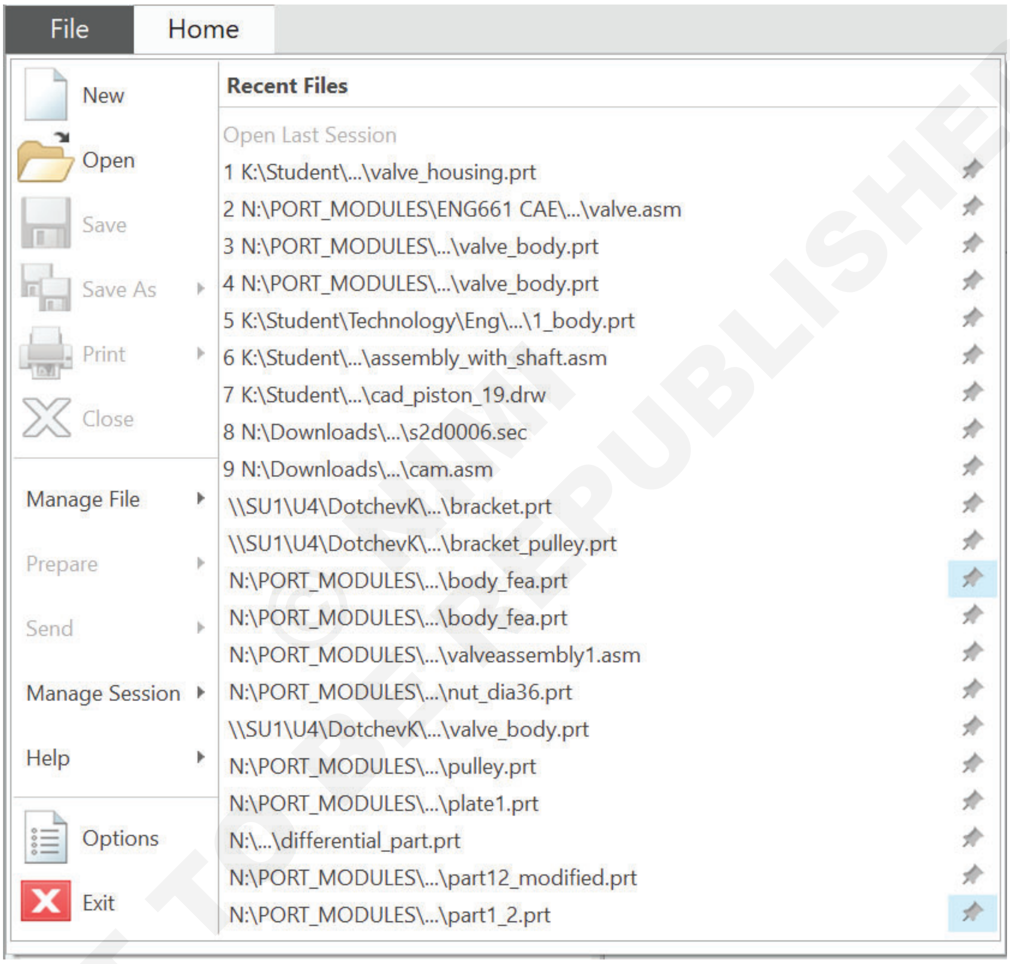
Rule number one is to set up a Working Directory immediately after starting the software! This ensures that all existing and new models (files) are opened or saved in one place.

Click on Select Working Directory icon ( ) from the Home menu at Start Up (Figure .5) and set up a Working Directory. Alternatively, select File > Manage Session > Select Working Director

File menu options

File menu options appear when selecting the File tab from the menu bar as shown in Figure 6. Some commands might be dimmed at Start up **.**

Fig 6



The main commands are:

New

Click on New ( ), (or File > New) to create a new model.

Open

Click on Open ( ), (or File > Open) to open a model.

New and Open commands create a new model or open an existing model from the Working Directory. They can be activated directly from the Home menu after Start Up .

Save

Click on Save icon ( ), (or File > Save) to save the model in the current Working Directory, thus creating another version of the object. Creo does not overwrite the existing file after Save. It creates anew file version containing the latest modifications.

Print

Print ( ) icon has a sub-menu with several commands to print the object from the active window to a printer as either a rendered image from the screen, a 2D drawing, a drawing in PDF format or a tessellated model in STL Stereo lithography format.

Close

Click on Close icon ( ) to close the active window. Note that the object is still in the computer RAM, which is called Session. If the current model is closed accidentally, it can be opened back from the Session and then saved.

Manage file

Manage File has a sub-menu with the following commands:

Rename (File > Manage File > Rename)

This command renames the current object and its versions in the Session (computer RAM) and/or in the Working Directory.

Delete Old Versions (File > Manage File > Delete Old Versions)

This command deletes all versions except the latest (with the highest variant number) of the current object from the Working Directory.

Delete All Versions (File > Manage File > Delete All Versions)

THIS COMMAND DELETES ALL VERSIONS of the current object from the Working Directory (from the computer disc) and the Session. The software asks the user to confirm this action. One should

be very careful when using this command.

Manage session

Session is the computer RAM allocated for all Creo-opened models (objects). Several objects (parts, assemblies, drawings, etc.) can be opened simultaneously and displayed in different windows within a single Session. Typically, the user works simultaneously on different models, belonging to a project, and switches from one window to another. Sometimes, a new Working Directory needs to be set up without stopping Creo. In this case, all opened objects in the Session need to be closed and removed from it to avoid memory conflicts. This is done with the following tools from the Manage Session menu

Erase Current (File > Manage Session > Erase Current)

This command removes an object (model) from the active window.

Erase Not Displayed (File > Manage Session > Erase Not Displayed)

Close

Click on Close icon ( ) to close the active window. Note that the object is still in the computer RAM, which is called Session. If the current model is closed accidentally, it can be opened back from the

Session and then saved.

File names and conventions

An object name can consist of letters and or numbers but cannot contain spaces or special symbols. The use of underscore symbol is permitted and it can also be used to separate some sections within the name. For example, MY_FIRST_MODEL is a correct name. Also, the software does not distinguish between upper and lower case letters. These rules apply to all Creo names, including the names of files, folders, features, parameters, etc.

File Management in Creo

In Creo, the Save command creates a new file with the same name but with a higher variant number and does not overwrite the previous file. After saving a model many times, the user might notice that there are

several files with the same name in the Working Directory. This is a specific feature that allows the user to retrieve an older version of the same model. The question is how to distinguish between different versions. By default, the file extensions are hidden in the Windows File Explorer. To reveal them, open the File Explorer, find the View tab and tick the option File name extensions (Windows 10) as shown in Figure .7. The file type and extension number are now visible. For example, a Creo file PART_MODEL.PRT.2 has the following convention: PART_MODEL is the object name, PRT is the object type *CAD/CAM with Creo Parametric* used by the software to associate it with an application (i.e. Part mode, or Assembly mode) and 2 is the model variant. After selecting Save, the software creates a new file in the Working Directory with the same name and type but a higher variant number 3 (i.e. PART_MODEL.PRT.3). The command Open always opens the object (file) with the highest extension (variant) number.

The main object types are as follows:

· PRT — This object is created in Part mode. It contains a record of the model geometry as a sequence of features and their parameters.

· ASM — This object is created in Assembly mode. It contains the assembly model history including the names of parts, sub-assemblies and their relative locations. It may contain additional data and surface

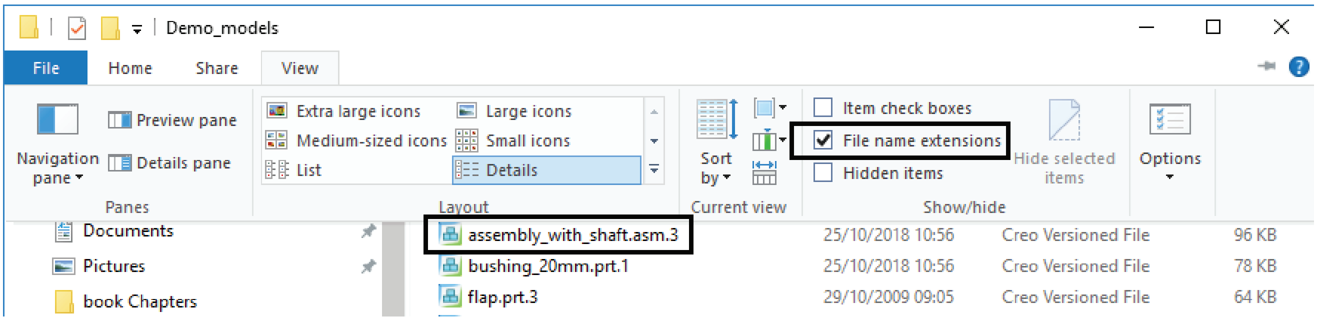
features created within the assembly. The assembly itself does not have the actual part(s) geometry. If a part model is missing from the Working Directory, then the assembly is not able to open and display

the part. The part name will appear in red in the assembly Model Tree.

· DRW — This object is created in Drawing mode. Every drawing refers to a part or assembly model. Similar to the assembly model, the drawing does not include the model geometry and will not display it

if the associated model file is missing.

Fig 7



Functions of the Mouse Buttons

The PC mouse is an essential input device in Creo 3D modelling. Every click of a mouse button generates an input for the software. It is important to understand and learn the functions of each mouse button when clicking on an icon (command) from the user interface or clicking on a model geometry item. In the beginning, it is difficult to remember all mouse button functions, but after some practice it will become second nature to use the correct button and key combinations. Usually, the software does not react if an incorrect button is clicked or if a wrong item selected. However, sometimes it may stop working or crash if the user repeatedly clicks a (wrong) button. Therefore, one should be careful and use the mouse buttons correctly. The main mouse button functions and combinations are explained in the following

Left Mouse Button (LMB)

The LMB is used to: click and initiate an action, such as click on interface icon, tab, tool, pull down menu, open a menu and other functions. For example to click on Open, Save, Extrude, Rotate and other

icons (commands). In addition, the LMB is used to click and select (pick) a geometry item as input for a command sequence. For example to click on datum plane, axis, point, edge, feature, etc.

Typically, the user performs a click and release action with the LMB to click and select (pick) an item.

In order to make multiple selections and add (or remove) more than one geometrical item(s), the user must press and hold the CTRL key down (CTRL + Hold) and then continue selecting the items with the LMB.

Middle Mouse Button or the Wheel (MMB)

This is used to accept the current selection or input. In 2D (Sketch) mode, the MMB is used to place dimensions and also to abort (stop) the current command.MMB is also used to rotate the model in the Graphics area and to viewit at various angles. To rotate, place the cursor on a point inside the

Graphics area, press and hold the MMB (MMB + Hold) and drag the mouse slowly to rotate the model. Some computer mice have a wheel instead of a middle button. The wheel rotation zooms-In or zoomsOut the model. Rotating and zooming the model are constantly used during the modelling process.

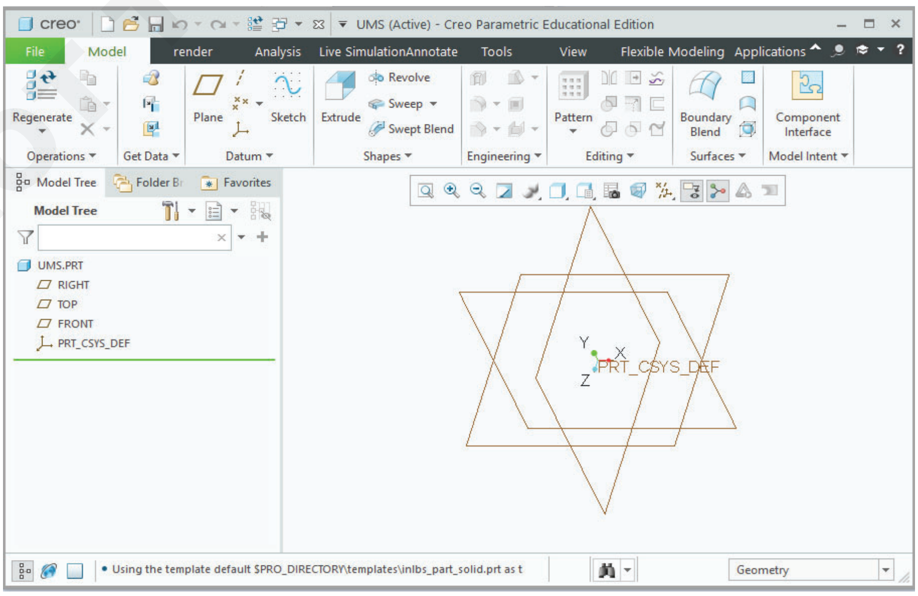
Another purpose of the MMB (or the wheel) is to translate the model on the screen. Press SHIFT and hold (SHIFT + Hold) with one finger, click the MMB, and then drag to translate the model.

Right Mouse Button (RMB)

It is used to initiate shortcut and mini menus by clicking once or by pressing and holding for a few seconds (RMB + Hold)

when you open a new file in Creo the window opens as shown in Fig 8.

Fig 8

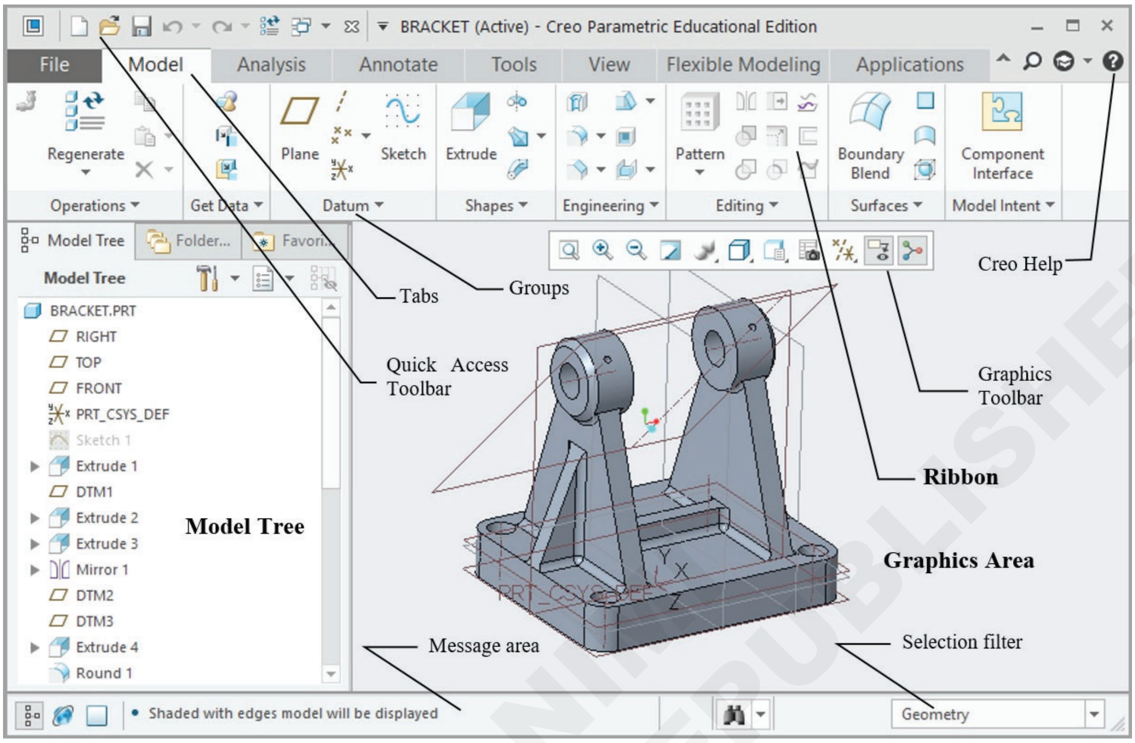


Familiarization of interface / windows

Objectives: At the end of this lesson you shall be able to
- **familiarize the main window components of the Creo**
- **state the functions of each components in Creo windows.**

Creo Modelling Interface The default, Part mode (part modelling) interface is shown in Fig 1.

Fig 1



The main components of the interface are as follows:

Main Menu — It is located on the top of the screen. Commands are grouped into tabs depending on their functions, such as File, Model, Analysis, Annotate, Tools, View, Flexible Modelling and Applications. Each of these tabs except File activates a specific

ribbon interface. Each ribbon contains a number of icons, organized into groups, which activate commands or tools with specific actions.

The commands are context dependent, and some of them might be disabled (dimmed) if correct prerequisites are not available or if geometrical entities not selected.

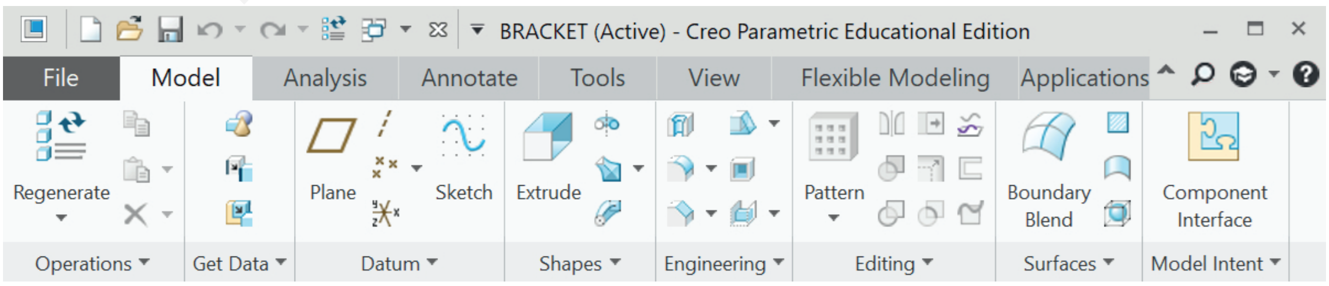
The ribbon interface is active when a new or an existing Creo object is opened in the Graphics area. Fig 1 shows Part mode ribbon interface with a part opened and shown in the Graphics area. Every Creo application, i.e. part, assembly, mould design, NC assembly, and others, has a specific interface and commands.

· Graphics Area — This is the screen area in which the model geometry is displayed. By using the mouse, the user interacts with the model, selecting geometrical entities such as plane, point, surface,

edge, etc., in order to create a new model feature or modify existing model features.

· Ribbon — The Model ribbon (Fig 2) is a context-sensitive menu that contains the main commands (icons and drop-down menus). These are arranged in groups from left to right following the modelling workflow.

Fig 2



- Message Area - A bar at the bottom of the screen under the Model Tree and Graphics area (Fig 1) providing prompts and help. Every command has a specific sequence and type of expected inputs. It is a good practice to read the messages because the system is telling what has been done and what is required next. It is recommended, especially for beginners, to read the messages during the learning process.

- Graphics Toolbar (Fig 3) - It is a small bar under the ribbon used to control the view of the model or sketch.

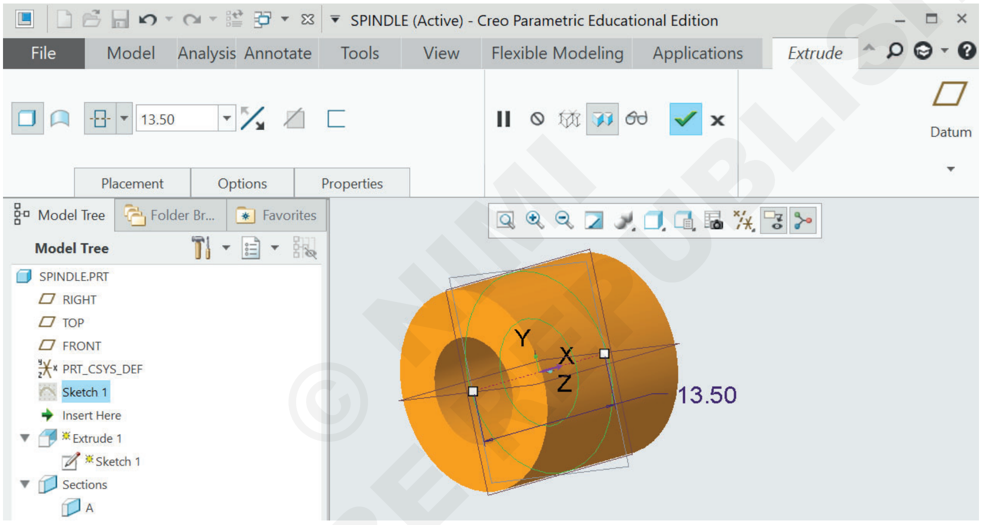
Fig 3

- Quick Access Toolbar — **It is located on the top left of the screen. It contains the following commands:** New, Open, Save, Undo, Redo, Regenerate **and** Windows **(shown in Fig 4).**

Fig 4

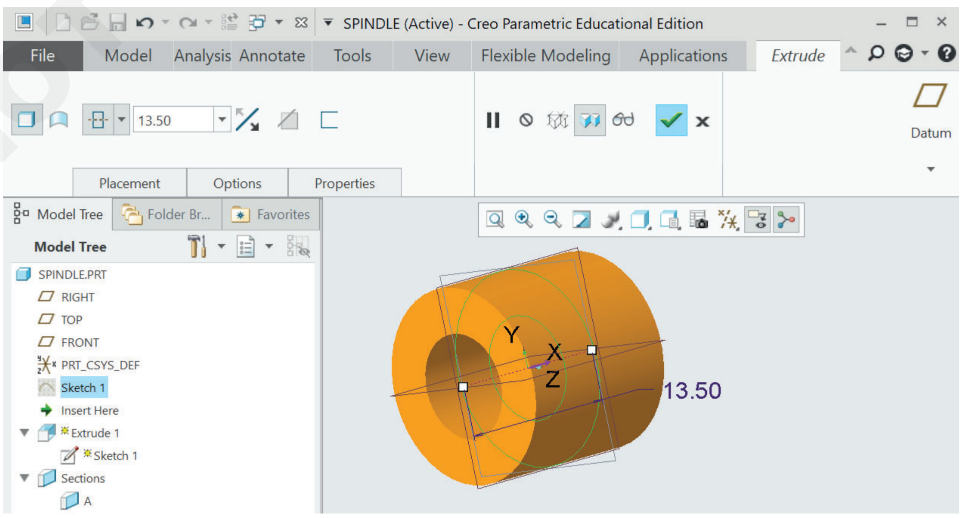
- The Regenerate command icon ( ) is frequently used in Part or Assembly mode to initiate the model recalculation after a change of a dimension or another parameter.
- Dashboard (Fig 5) — This is a submenu that will open after activation of a top-level command. For example, commands for creating Extrude, Rotate, Blend, Sweep, or other features have their dashboards. Each dashboard has command-specific tools and options for creating variations of the activated feature. For instance, the Extrude command ( ) opens a dashboard that has options to create a protrusion (adding material) or cut (removing material). Also, the dashboard can switch to create a solid or surface feature.

Fig 5



Click (LMB) on Extrude (i.e. Model > Shapes > Extrude) to activate the Extrude dashboard (Fig 6). If all necessary inputs are provided into the dashboard, for example, a 2D sketch (two concentric circles) and extrude depth (Fig 6), then a green tick ( ) is available to save the changes and close the Extrude command. Creo cannot continue if the command dashboard is still open. Click (LMB) on the OK green tick ( ) or Cancel ( ) icon on the right-hand side of the dashboard to close and continue.

Fig 6



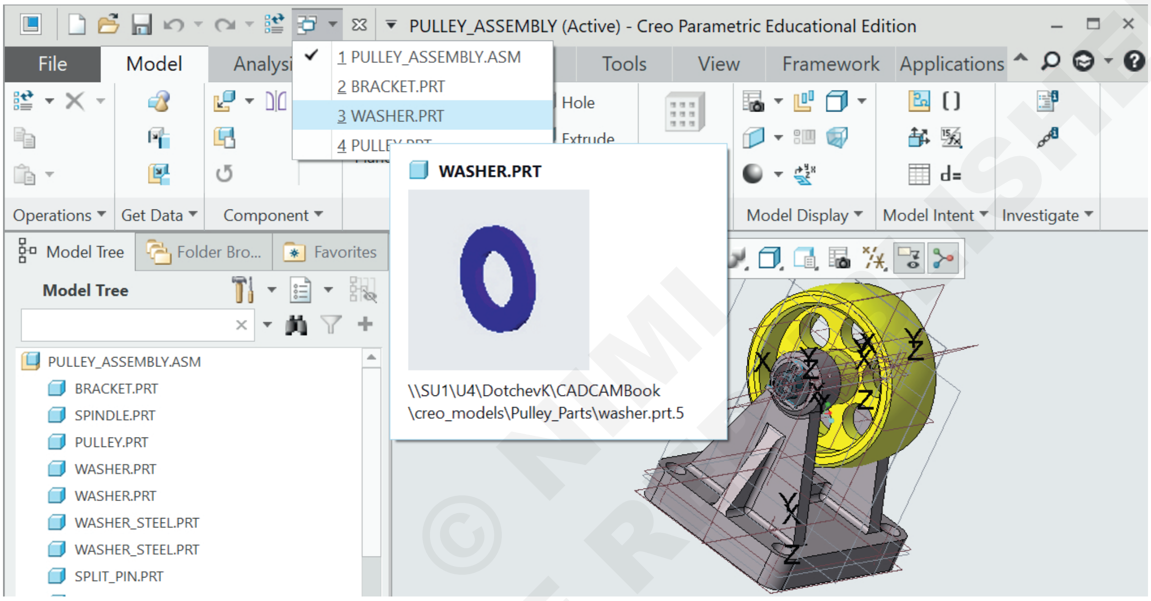
- Model Tree (Figure 1) — It is a record of all features and parameters following the order of their creation. In fact, the model is a software programme containing commands and input values that Creo executes to create the model geometry. The Model Tree shows the model history. It updates every time new features are added or deleted.
- Creo Help — To access Creo Help Centre, click (LMB) on Help ( ) icon at the top right corner of the main screen (Figure 1).
- Use the Search tool ( ) and type into the search bar to quickly find anything related to Creo, e.g. ‘extrude’. The result activates the command or highlights it on the screen in real time.
- PTC Learning Connector ( ) — It is located next to Creo Help. It allows the user to log into the PTC University e-Learning database and access tutorials and videos. The user needs a Login name and Password. Most help however can be quickly sourced from the Creo Help Centre tab.

Working with Multiple Windows

The user can open several windows, each containing a different object, within the same Session. However, only one of these windows, the Active window, can be accessed for modelling. A common practice is to switch from one active window to another and work on several opened models.

This can be done by accessing the Active window ( ) icon from the Quick Access Toolbar. Click on the icon, scroll down and select the model that is to become active, as shown in (Fig 7). The current Active

Fig 7



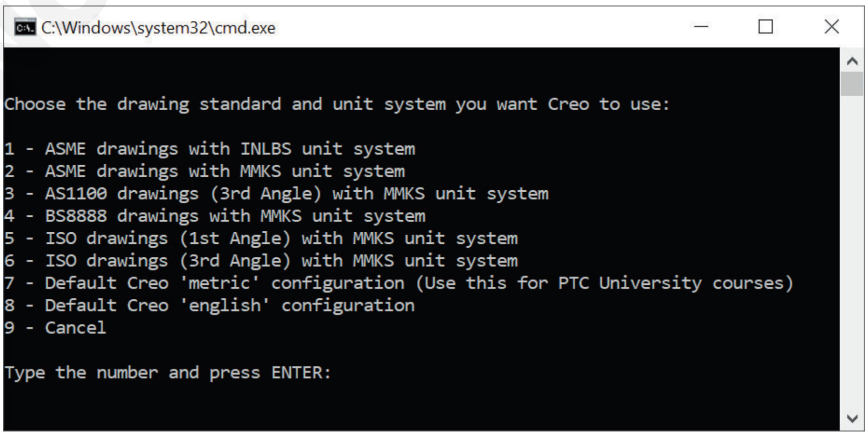
Window is marked with a black tick in front of the model name. After the Active window (object) selection, the system opens the model in the specific to the model environment, i.e. Part mode, Assembly mode, etc.

System of Units and Drawing Configuration (Fig 8)

By default, the software will be installed and configured to the ASME Inch/Pound (Imperial) system of units. In order to reconfigure it to Metric units (mm, kg), the user should run the configuration *.BAT file as follows:

- Go to C:\Program Files\PTC\Creo 4.0\M050 (or another version)\ Common Files\creo_standards.
- Select CONFIGURE.BAT file and Run as administrator.
- The CMD black window (Fig 8) opens. · Select the desired system from the list, type the corresponding number and press ENTER. For example, for the Metric units (mm, KG, second) select 4, 5 or 6. Select 7 for the Metric units (mm, N, second).
- Every option corresponds to a specific engineering drawing standard.

Fig 8



Sketching in creo

Objectives: At the end of this lesson you shall be able to
- **state what is 2D sketch**
- **brief all basic tools of sketcher**
- **explain the methods to modify all dimensions at once (scaling)**
- **brief method of sketching according to design intent.**

Sketcher is the main creation tool of Creo Parametric.

The sketcher toolbar ribbon is located at the top of the window.This section explores the many options of sketcher.

Its basic icons are shown in Figure 1. The LMB (Left Mouse Button) is used to select geometry or to select a location when creating geometry.

Fig 1 Sketcher Ribbon

Entering Sketcher Directly from Creo Parametric

Select the NEW FILE icon from the ribbon at the top of the screen. Save as >File>New. When the New file window opens, select Sketch as the type, then enter a valid filename without the extension. Pick OK. (Fig 2).

Fig 1

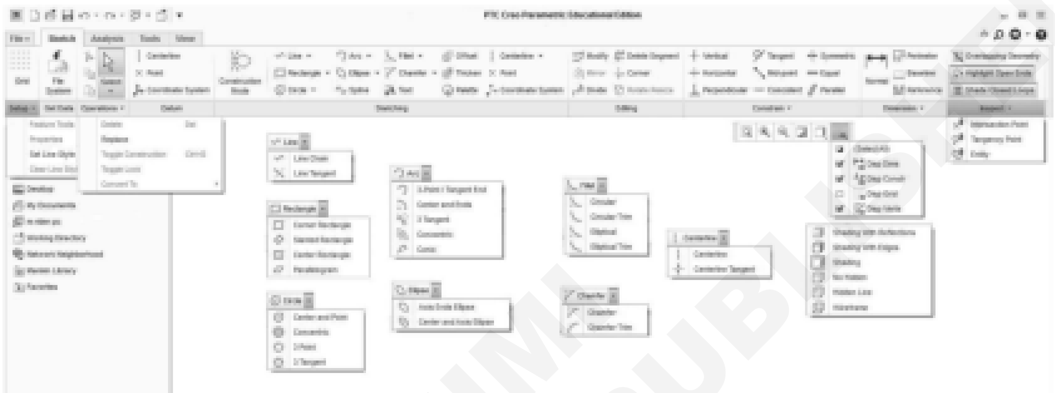


Fig 2

Entering Sketcher from Part Mode within Creo Parametric: This section assumes We are in part creation mode already. and To enter sketcher mode from part mode, *select* the SKETCHER icon at the top of the screen. *Pick* the plane that We wish to sketch on followed by orienting another plane toward the right, top, left, or bottom of the screen. *Pick* the Sketch button.

If necessary in the upper left corner of the window, *select* the SKETCH VIEW icon to orient the sketching plane parallel to the display screen.

The sketcher ribbon is slightly different at the left and right ends.

Sketching: Sketched entity can be designated as geometry or construction. Sketching tools create geometric entities by default. To create construction entities, click Sketch >  Construction Mode.We can convert geometric entities into construction entities or vice versa:

- To convert geometric entities to construction entities, select an entity then click Construction on the mini toolbar.

Construction entities are displayed with a dashed line style.

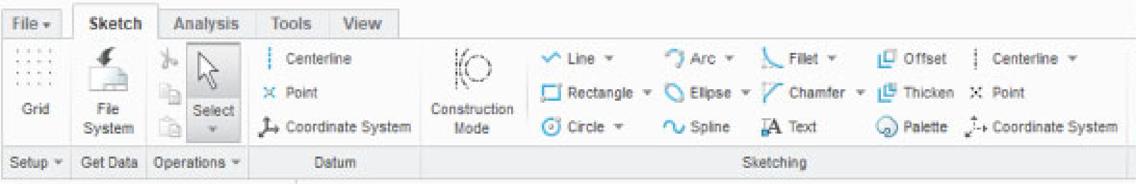
- To convert construction entities to geometric entities, select an entity then click Solid on the mini toolbar.

Geometric entities are displayed with a solid line style.

Points, centerlines, and coordinate systems have separate tools for creating datum or construction instances. We can use the Search tool to search for a datum created using the Point, Centerline, or Coordinate System Datum tools.

Dynamic Constraining: As We move the pointer, applicable constraints are offered and the geometry snaps to these constraints. The relevant existing geometry is highlighted. For example, a tangent constraint is offered when We sketch a line near the circumference of a circle. We can accept, lock, disable, or disregard the offered constraints. Selecting Sketching Tools. (Fig 3).

Fig 3



The Sketching group contains tools to create the following entities

- Lines
- Quadrilaterals
- Circles
- Arcs
- Ellipses
- Splines
- Fillets
- Chamfers
- Text
- Offset
- Thicken
- Project
- Centerline
- Point
- Coordinate system

Using the Mouse or Keyboard in Sketcher

We can use the mouse or keyboard to perform the following operations:

- Click an entity to see the available options on the mini toolbar.
- Middle-click or press ESC to abort the current operation. Middle-click or press ESC again to exit the active tool.
- Right-click to lock an offered constraint while We are sketching. Right-click again to disable the constraint and right-click a third time to enable it again.
- Right-click the sketch window to access the shortcut menu. Commands change when an entity is selected.

Points and Centerlines

We can create construction points, construction centerlines, datum points, and datum centerlines.

Points and centerlines are represented in the graphics window in different default colors and line styles. We can customize these settings in the System Appearance area of the Creo Parametric Options dialog box.

The Difference Between Construction and Datum Entities

Construction points and centerlines are sketching aids and cannot be referenced outside Sketcher. Datum entities convey feature-level information outside of Sketcher. Use datum entities to add information to sketched curve features and sketch-based features in 2D and 3D Sketcher.

To change a construction entity to a datum entity, click the entity and choose Solid on the mini toolbar. To change a datum entity to a construction entity, click the entity then choose Construction on the mini toolbar.

Using Datum Points

We can create datum points and centerline in a sketch and then use it when we create entities in Part or Assembly mode. The following table provides information about the connections between the features and the datum entities.

Feature	Entity Created by a Datum Point	Entity Created by a Datum Centerline
Sketched curve	Datum point	Datum axis
Extrude	Axis through the point and normal to the sketch plane	
Sketched pattern	Pattern instance	
Revolve feature		Axis of rotation

All other sketch-based features	No geometry is created	No geometry is created
Other features that use Sketcher points or centerlines to convey feature-level geometry	No geometry is created	No geometry is created

Using Datum Entities and Features

Keep in mind the following points when sketching an axis of rotation for a revolve feature or when placing a hole feature:

- When a sketch for a Revolve feature includes several geometry centerlines, the first centerline We create is automatically designated as the axis of revolution.
- To designate a datum or construction centerline as the axis of revolution or to select a different datum or construction centerline as the axis of rotation, select the centerline and right-click and choose Designate Axis of Revolution on the shortcut menu.

Coordinate Systems

We can use the coordinate system tool in Sketcher to create three types of coordinate systems: construction, datum, and horizontal-vertical. When horizontal-vertical coordinate systems are available for creation, we cannot create construction and datum coordinate systems, and vice versa.

There are separate sketching tools for creating construction coordinate systems and datum coordinate systems. However, we can toggle between construction and datum status using the shortcut menu. When We create construction and datum coordinate systems, construction and datum centerlines are also created along the x- and y-axes.

Construction Coordinate Systems

Construction coordinate systems are sketching aids and do not convey any information outside Sketcher. Use a construction coordinate system to accomplish the following tasks:

- Dimension a spline—Modify the spline points by specifying the x-, y-, and z-axis coordinates with respect to the coordinate system.
- Create a reference—Add coordinate systems to any section to aid dimensioning.

Datum and Horizontal-Vertical Coordinate Systems

A datum coordinate system conveys feature-level information outside of Sketcher; We can use it to add information to sketched curve features and sketch-based features in 2–D and 3–D Sketcher. Datum coordinate systems create entities in Part or Assembly mode as follows:

Type	Availability	Behaviour Outside Sketcher
Datum	2–D Sketcher Sketched curves Sketched patterns Toroidal bends Wrap features	In a sketched curve—Creates a datum coordinate system In a sketched pattern—Places pattern members In a toroidal bend—Defines the neutral plane In a wrap feature—Defines the origin
Horizontal-vertical	3–D Sketcher for features that have not been converted to Wildfire UI and that require a coordinate system, such as Graph and Blend.	Conveys feature information, such as graph coordinates or the relative origin for each of the sections used for blends.

If We retrieve a section that contains a coordinate system type unavailable in the current sketch, it is converted automatically.

Automatic Coordinate System Conversions

When We retrieve a section from the Sketcher Palette or from a legacy file, it may include a coordinate system type that is unavailable in the current Sketcher environment. Under these circumstances, the coordinate system is automatically converted as follows:

- Horizontal-vertical—Becomes a geometry coordinate system, in instances where geometry coordinate systems are available. In other instances, a construction coordinate system is substituted.

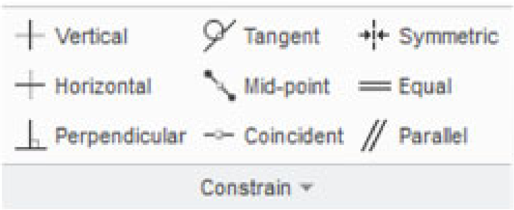
For Sketched Curve features, the geometry coordinate system generates a datum coordinate system outside Sketcher. Make sure to toggle the system to Construction if We do not want to create the datum.

- Construction or Datum—Becomes two perpendicular construction centerlines with a construction point at the intersection, in instances where horizontal-vertical coordinate systems are available.

All other coordinate systems created before Pro/ ENGINEER Wildfire 5.0, that are not needed for feature-level information, are converted to construction coordinate systems.

Sketch Constraints (Relations). (Fig 4)

Fig 4



Constraint	Geometric entities to select	Resulting Constraint
Horizontal or Vertical	One or more lines or two or more points.	The lines become horizontal or vertical (as defined by the current sketch space). Points arealigned horizontally or vertically.
Collinear	Two or more lines.	The items lie on the same infinite line.
Perpendicular	Two lines.	The two items are perpendicular to each other.
Parallel	Two or more lines. A line and a plane (or a planar face) in a 3D sketch. The items are parallel to each other.	The line is parallel to the selected plane.
Tangent	An arc, ellipse, or spline, and a line or arc.	The two items remain tangent.
Concentric	Two or more arcs, or a point and an arc.	The arcs share the same centerpoint.
Midpoint	Two lines or a point and a line.	The point remains at the midpoint of the line.
Coincident	A point and a line, arc, or ellipse.	The point lies on the line, arc, or ellipse.
Equal	Two or more lines or two or more arcs.	The line lengths or radii remain equal.
Symmetric	A centerline and two points, lines, arcs, or ellipses.	The items remain equidistant from the centerline, on a line perpendicular to the centerline.

About the Dimension Tools

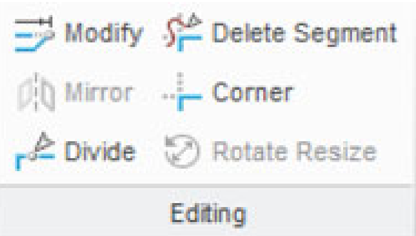
Use these dimension tools to dimension Sketcher entities:

- Normal—Creates a linear, radial, diameter, angular, conic, or ordinate dimension. Works in Continue mode.
- Perimeter—Dimensions the total length of a chain or a loop of entities.
- Baseline—Creates linear dimension entities based on the position of the first selection. All subsequent selections are measured relative to the first selection.
- Reference—Creates a symbolic dimension for the entity.

To Modify Dimension Values (Fig 5)

To modify one dimension, double-click it. To modify several dimensions at once, use the Modify Dimensions dialog box.

Fig 5



To Modify One Dimension

1 Double-click the dimension.

2 Type a new value and press ENTER. The dimension is changed.

To Trim and Extend Entities

1 Bring the pointer over the entity that we want to trim. The entity is highlighted.

2 Holding down the CONTROL key, drag the endpoint of the entity that we want to trim. The entity is trimmed or extended in the direction in which we drag it. At every intersection, the constraint that it created is displayed.

To Trim Entities to Each Other

1 Click Sketch > Corner.

2 Click any two entities (they do not have to intersect) on the portion of the entity that we want to keep. The two entities are trimmed.

To Delete a Segment

1 Click Sketch > Delete Segment.

2 Click the segment that we want to delete. The segment is deleted.

To Divide Entities

We can divide a section entity into two or more new entities. If the entity is dimensioned, delete the dimension before using the Divide command.

1 Click Sketch > Divide.

2 Click the entity at the locations where we want to divide. The entity is divided at the specified location.

To create a divide at an intersection, click near the intersection. The curser automatically snaps to the intersection and the divide is created.

About Dividing and Trimming Entities

We can divide and trim entities by using the Divide and Trim tools or by holding down the CONTROL key and dragging the endpoint of the entity to the desired location.

Mirroring Geometry

Use the Mirror command to mirror Sketcher geometry about a sketched centerline. For example, we can create half of the section and then mirror it.

When we mirror geometry, constraints are also mirrored.

We can mirror only geometric entities. We cannot mirror dimensions, text entities, centerlines, and reference entities.

About Cut, Copy, and Paste Operations

We can use the cut and copy operations to remove or duplicate, respectively, a portion of a section or the complete section.

The sketch entities that we cut or copy are placed on the clipboard

Click Sketch > Cut or press CTRL+X to cut the selected Sketcher geometry entity or entities.

Click Sketch > Copy or press CTRL+C to copy the selected sketch geometry entity or entities.

We can use the paste operation to place the cut or copied entities at a required location within an active section.

Click Sketch > Paste or press CTRL+V to paste the copied entities into the active section.

we can use the copied or cut sketch geometry multiple times. We can also use the cut, copy, and paste operations to move the contents of a section across multiple sections.

About Translating, Rotating, and Scaling a Section

We can translate, rotate, or scale an existing sketcher geometry or an imported or pasted sketch geometry within an active sketch. The geometry we select to translate, rotate, or scale appears within a bounding box with the following handles:

- The translation handle appears at the center of the bounding box.
- The rotation handle appears at the upper right corner of the bounding box.
- The scale handle appears at the lower right corner of the bounding box.

We can move the translation, rotation, or scale handles from their original locations. To move these handles, right-click a handle and drag the handle to a new location.

Consider the following points when we translate, rotate, or scale the sketcher geometry:

- The original position of the translation, rotation, and scale handles appear in dimmed color when we perform a translation, rotation, or scale operation.
- The translation, rotation, or scaling dimensions that appear during the translation, rotation, or scaling operations, are removed after we complete these operations.
- Dimensions and constraints are created when we complete the translation, rotation, or scaling operation. These dimensions and constraints are independent of the changes to the geometry as a result of the translation, rotation, or scaling operation.
- We can undo or redo the translation, rotation, or scaling operation.
- While performing a translation, rotation, or scaling operation, we can undo or redo any of these operations to attain an intermediate state of the geometry. After we complete the translation operation we cannot replay the intermediate state of the geometry.
- After we complete the translation, rotation, or scaling operation by quitting the Rotate Resize tab, the undo operation displays the geometry in the state before the translation operations.
- Toaccess the Rotate Resize tab, click Sketch > Rotate Resize. In addition, the Rotate Resize tab opens automatically when we perform one of the following operations:
- Paste a copied or cut geometry.
- Import a shape from the Sketcher palette.
- Import an existing section from the file system.

We cannot scale a section if we use Paste Special to paste a copied or cut section.

Basic modeling

Objectives: At the end of this lesson you shall be able to
- **state the basic features in Creo**
- **brief the features like extrude, extrude cut, revolve, revolve cut.**

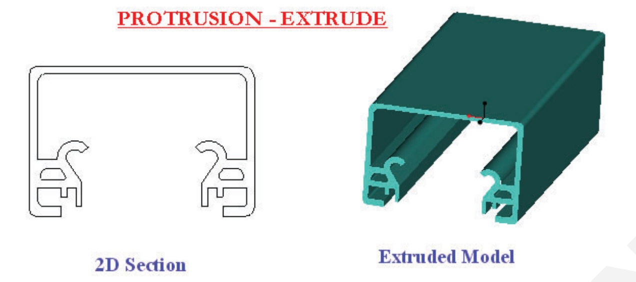
Creating Features

There are many kinds of features that you can create on a part. There are solid features and surface features, and features specific to applications. Part modeling refers to the creation of solid features and some user-defined features. Some features add material and some remove material. The most basic way to add material is through a protrusion. The most basic way to remove material is through a cut.

Extrude (Fig 1)

Extrusion is a method of defining three-dimensional geometry by projecting a two-dimensional section at a specified distance normal to the sketching plane.

Fig 1



Use the Extrude tool as one of the basic creation methods that allows you to create a solid or surface, and to add or remove material.

You can create the following extrusion types with the Extrude tool:

- Protrusion—Solid, Thickened
- Cut—Solid, Thickened
- Extruded surface
- Surface trim—Regular, Thickened

Accessing the Extrude Tool

To access the Extrude tool, click pick the Extrude button in the Shapes group of the ribbon

- (Preferred) Select an existing sketched datum curve and click. This method is referred to as object-action.
- Click and create a sketch to extrude. This method is referred to as action-object.
- Select a datum plane or planar surface to use as the sketching plane and then click .

Dash Board

The Dash board consists of the following elements:

Common Extrusion options

- —Creates a solid.
- —Creates a surface.
- **Depth options**—Constrains the depth of the feature. (Fig 2)

Fig 2

- Depth box and Reference collector—Specifies a depth value for the extrusion that is controlled by the depth dimension. If a depth reference is required, the text box acts as a collector and lists the reference summary.
- —Flips the direction of feature creation with respect to the sketching plane.
- Options used for creating a cut
- —Creates a cut using the extruded volume.
- —Changes the side to be removed when creating a cut.
- Options used with the Thicken Sketch option
- —Creates a feature by assigning a thickness to the section outline.
- —Changes the side where a thickness is added, or adds a thickness to both sides.
- Thickness box—Specifies a thickness value to apply to the section outline.
- Options used for creating a Surface Trim
- —Trims a surface using a projected section.
- —Changes the side of the quilt to be removed, or keeps both sides.
- Quilt collector—If both sides of the quilt are kept, specifies the side to retain the quilt id of the original quilt.

Slide-up Panels

- The Extrude tool provides the following slide-up panels:
- Placement—Use this slide-up panel to redefine the feature section. Click Define to create or change the section. Click Unlink to make the section independent of the sketched datum curve.
- Options—Use this slide-up panel to do the following:
- Redefine the depth of the feature for each side of the sketching plane.
- Create a surface feature with capped ends by selecting the Capped Ends option.
- Properties—Use this slide-up panel to edit the feature name and open feature information in the Pro/ENGINEER browser.

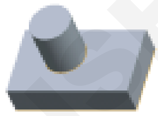
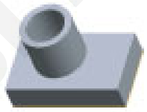
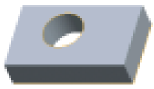
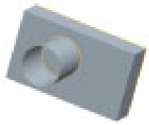
To create an extruded feature

1. Create the sketch profile.
2. Select the Extrude tool 
3. Select / sketch the section.
4. Specify the direction of feature creation.
5. Define depth attributes.
6. To verify the feature, click .

If you are satisfied with the geometry you created, click

Different Types of Extrude Feature

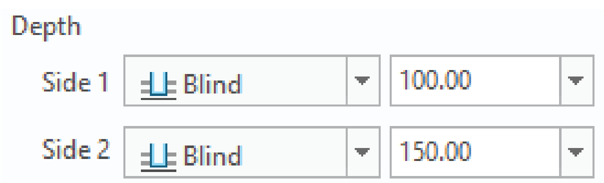
The next table shows different types of geometry that you can create with the Extrude tool.

Extruded Solid Protrusion	
Extruded Solid Protrusion with an assigned thickness (Thickened)	
Extruded Cut, created with Through Next	
Extruded Surface	
Extruded Surface Trim A section is projected onto quilt to cut out a hole in the quilt.	
Surface trim with an open section A section is projected on the quilt to create a trim line and cut the quilt.	

Symmetrical and Asymmetrical depth options

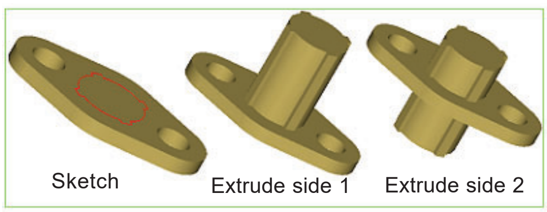
The system creates a default extrusion using the Blind depth option. (Fig 3)

Fig 3



- Side1—To specify the depth of feature on one side of the sketching plane.
- Side2—Specify the depth to feature to the other side of the sketching plane separately. (Fig 4)

Fig 4



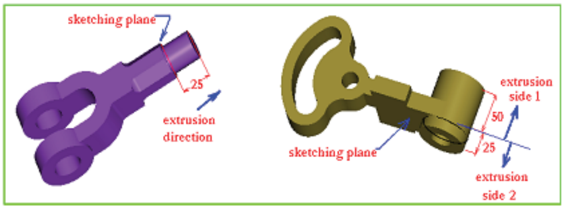
Specifying the Depth Attributes

You can specify the depth of an extruded feature by selecting one of the following depth options:

Blind—Extrudes a section from the sketching plane by the specified depth value. (Fig 5)

Note: Specifying a negative depth value flips the depth direction

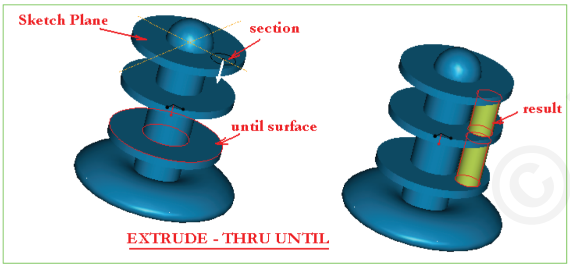
Fig 5



Symmetric—**Extrudes a section on each side of the sketching plane by half of the specified depth value.**

Through Until—Extrudes a section to intersect with a selected surface or plane. For a terminating surface, you can select the following (Fig 6)

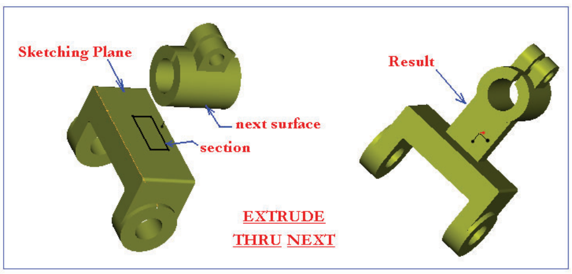
Fig 6



- A part surface, which is not required to be planar
- A datum plane, which is not required to be parallel to the sketching plane
- A quilt composed of one or several surfaces
- In an assembly, you can select geometry of another component.

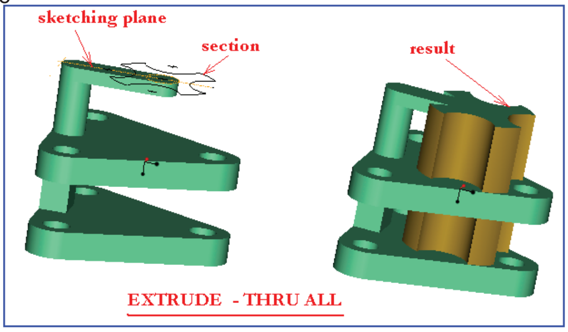
To Next—Extrudes a section to the next surface. Use this option to terminate a feature at the first surface it reaches.Note: You cannot use a datum plane as a terminating surface. (Fig 7)

Fig 7



Through All—Extrudes a section to intersect with all surfaces. Use this option to terminate a feature at the last surface it reaches. (Fig 8)

Fig 8



To Selected—Extrudes a section to a selected point, curve, plane, or surface. (Fig 9, 10 & 11)

Fig 9

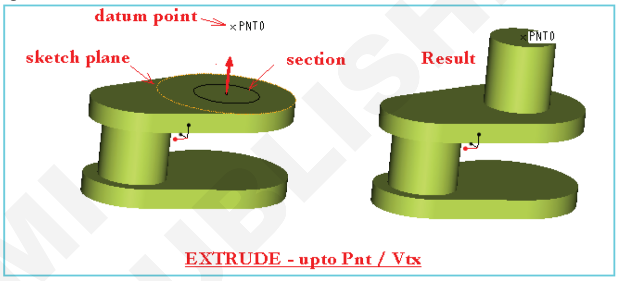


Fig 10

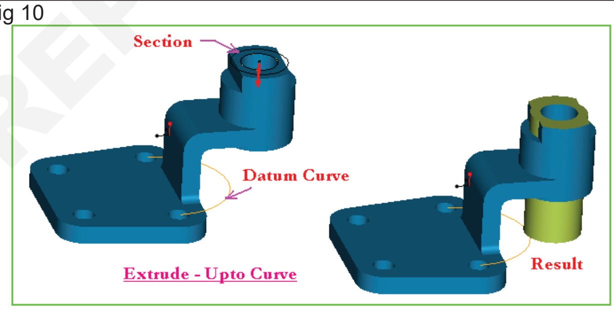
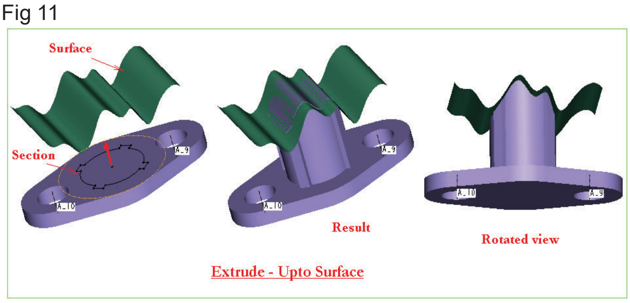
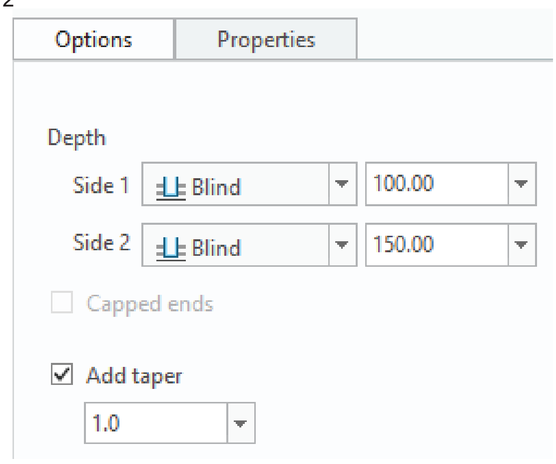


Fig 11



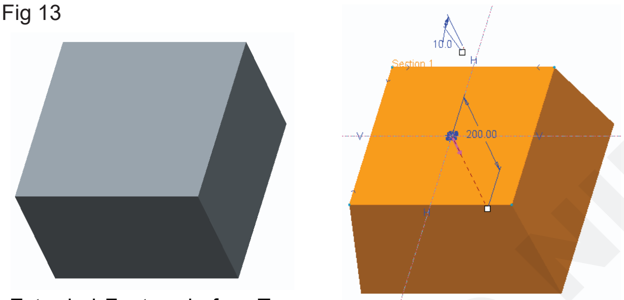
To Add Taper to a Extruded Feature (Fig 12)

Fig 12



You can add a taper to an extrude if the section of the feature is a closed loop. You cannot taper an extrude if is selected to add a thickness. The Extrude tab must be open to perform this procedure. (Fig 13

Fig 13



Extruded Feature before Taper   Extruded Feature after Taper

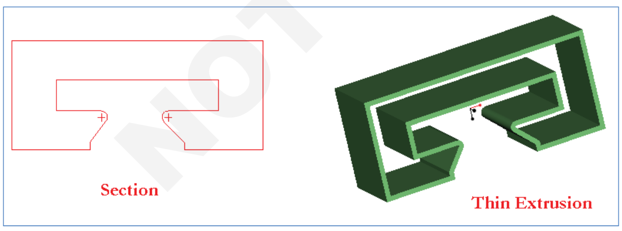
1 Click the Options tab.

2 Select the Add taper check box

3 Type a value for the taper angle from -89° to +89° in the box.

To Create a Thin Extruded Feature (Fig 14)

1 Click Model Extrude. The Extrude tab opens.



2 Select a sketch to extrude, or to create a sketch, click the Placement tab, click Define, sketch a section, and click OK. Note :You could also select a sketch first, or select a datum plane or planar surface first, and then click Model Extrude.

3 Select a depth option from the menu.

4 To flip the direction of feature creation in relation to the sketching plane, click .

5 To add thickness to the sketch, click or right-click the feature and choose Thicken Sketch on the shortcut menu.

1 Type a value for thickness in the box to the right of

2 To change the side of the sketch where the thickness is added, click to the right of the thickness box. You can switch between three modes: Side 1, Side 2 or Both sides.

3 (Optional) The section used for the extrusion is associative with the sketched datum curve you selected. To break this associativity and copy the section into the extrusion, click the Placement tab, and then click Unlink.

4 To create a double-sided feature, click the Options tab and define the depth for the second side.

Click

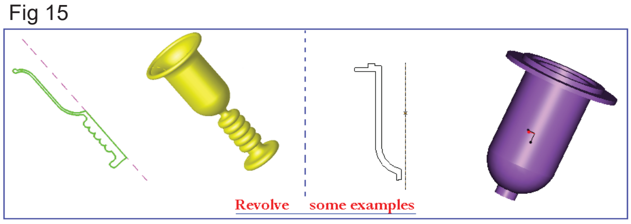
Revolve (Fig 15)

The Revolve tool creates a feature by revolving a sketched section around a centerline. Use the Revolve tool as one of the basic creation methods that allows you to create a revolved geometry as a solid or surface, and to add or remove material.

You can create different types of revolved feature with the Revolve tool:

- **Revolved protrusion—Solid, Thickened**
- **Revolved cut—Solid, Thickened**
- **Revolved surface**
- **Revolved surface trim—Regular, Thickened.**

Fig 15

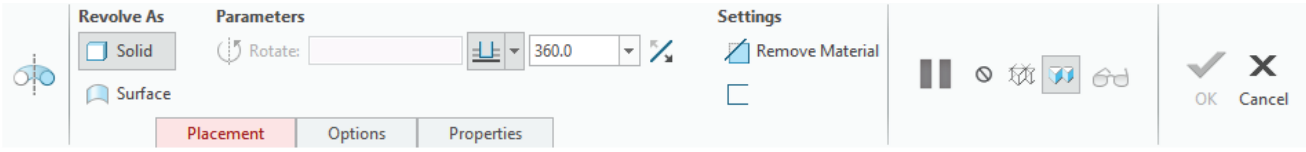


To Create a Revolved Feature

To access the Revolve tool, pick the Revolve the Extrude button in the Shapes group of the ribbon.

The dash board consists of the following elements: Common Revolve options. (Fig 16)

Fig 16



- —Creates a solid feature.
- —Creates a surface feature.
- Angle options—Lists options to constrain the angle of revolution for the feature. Choose one of these options: Variable, Symmetric, or To Selected.
- Angle box/Reference collector—Specifies an angle value for the revolved feature. If a reference is required, the text box acts as a collector and lists the reference summary.
- —Flips the direction of feature creation with respect to the sketching plane.
- Options used for creating a cut
- —Creates a cut using the revolved feature volume.
- —Changes the side to be removed when creating a cut.
- Options used with the Thicken Sketch option
- —Creates a feature by assigning a thickness to the section outline.
- —Changes the side where a thickness is added or adds a thickness to both sides.
- Thickness box—Specifies a thickness value to apply to the section outline.
- Options used for creating a Revolved Surface Trim
- —Trims a surface using a revolved section.
- —Changes the side of the quilt to be removed, or keeps both sides.
- Quilt collector—If both sides of the quilt are kept, select the side to retain the quilt id of the original quilt.

About Sections Used for Revolved Features

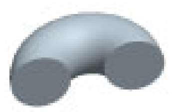
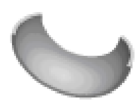
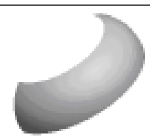
Creating a revolved feature requires you to define a section that you want to revolve and the axis of revolution. The axis can be a linear reference or a Sketcher centerline.

Consider the following rules for defining revolved sections:

- You can use open or closed sections to create revolved surfaces.
- Geometry must be sketched only on one side of the axis of revolution.

Different Types of Revolved Feature

The next table shows different types of geometry that you can create with the Revolve tool.

Revolved Solid Protrusion	
Revolved Protrusion with an assigned thickness **(created using a closed section)**	
Revolved Protrusion with an assigned thickness **(created using an open section)**	
Revolved Cut	
Revolved Surface	

About the Angle of Revolution (Fig 17)

In a revolved feature, a section is revolved around an axis of revolution to a specified angle. You can define the angle of revolution by selecting one of the following angle options:

Variable—Revolves a section from the sketching plane by the specified angle value. Type the angle value in the box, or select one of predefined angles (90,180,270, 360).

Symmetric—Revolves a section on each side of the sketching plane by half of the specified angle value.

To Selected—Revolves a section up to a selected datum point, vertex, plane, or surface.

Note: The terminating plane or surface must contain the axis of revolution.

Tips for Changing the Angle Option Using Snapping

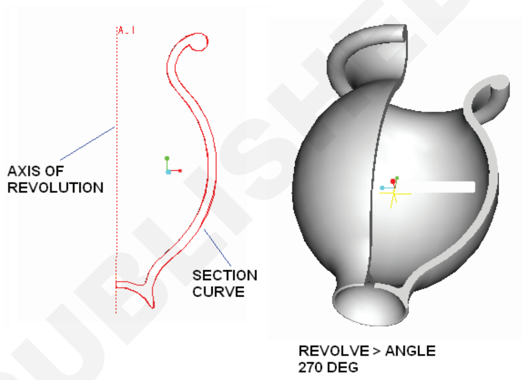
You can use snapping to the nearest reference to change the angle option from Variable to 'To Selected'. Hold down SHIFT and drag the handle to a reference that you want to use to terminate the feature.You can change the angle option back to Variable by holding down SHIFT and dragging the handle. As you drag the handle, the angle dimension is displayed.

About a Revolved Cut

You can use the Revolve tool to remove material by revolving a sketched section about a centerline.To create a cut, use the same angle options as for protrusions. For solid cuts, use closed sections. For cuts created with Thicken Sketch, you can use both closed and open sections.While defining a cut, you can switch between the following feature attributes:

- Cut and protrusion by clicking Remove Material
- The side where material is removed by clicking Flip Material Side
- Solid and thin cut by clicking Thicken Sketch.

Fig 17



Advanced Modelling

Objectives : At the end of this lesson you shall be to
- **explain the blend, sweep and shell features in creo**
- **explain profile, pattern and round features in creo**
- **state the features of chamfer, hole and draft in creo.**

The Advanced modelling using Creo Parametric, teaches you how to use advanced part modelling techniques to improve your product designs. You will learn various tools used in Creo parametric,

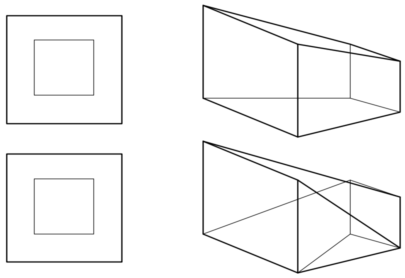
Blends

A blended feature consists of a series of at least two planar sections that have been joined at their vertices with transitional surfaces to form a continuous feature.

Example: Starting Points and Blend Shape

The starting points of the sections influence the shape of the blend. Fig 1.

Fig 1



TD20N168051

Swept blends

A swept blend can have two trajectories: an origin trajectory (required) and a secondary trajectory (optional). Each Swept Blend feature must have at least two sections, and sections may be added between these two sections. To define a trajectory of the swept blend, you can select a sketched curve, a chain of datum curves, or edges. Only one trajectory is active at a time.

Example: Creating a Swept Blend Fig.2

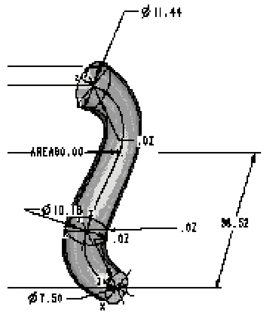
Completed Swept Blend

Sweeps

Create a Sweep feature by sweeping a cross-sectional sketch along one or more trajectories. You can control the object's orientation, rotation, and geometry.

Material can be added or removed as the sketch sweeps along the trajectory. You can add a thickness to the sketch. The geometric representation of the sweep can be solid or surface.

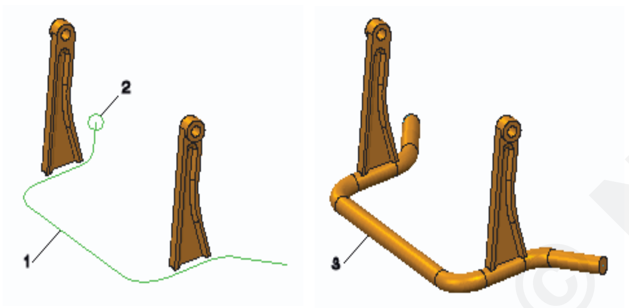
Fig 2



You can merge the end of a solid sweep to a nearby solid surface without leaving a gap. You can close each end of a surface sweep if the sketch forms a closed loop, and the trajectory is open. When you create cut, trim, or thin features, use the arrows in the graphics window to indicate the direction of the tool operation. About Constant Section Sweeps

You create a sweep feature with a constant section by specifying a trajectory and then sketching a section to follow along it. Fig 3.

Fig 3



1 Trajectory 2 Section 3 Swept feature.

Shell Feature

The Shell feature hollows out the inside of the solid, leaving a shell of a specified wall thickness.

It lets you specify a surface or surfaces that you want to remove from the shell (Fig.4). If you do not select a surface to remove, a "closed" shell is created (Fig.5), with the whole inside of the part hollowed out and no access to the hollow.

If you flip the thickness side (for example, by entering a negative value, or by clicking on the Shell tab), the shell thickness is added to the outside of the part.

When defining a shell, you can also select surfaces where you want to assign a different thickness. You can specify independent thickness values for each such surface.

The Shell feature allows you to select the adjacent tangent surfaces. This enables you to remove or offset (independently or with different thickness) the surfaces which are tangent to their neighbouring surface at one or more boundaries.

At the tangent edge, where the separation of the shell offset occurs, a normal capping surface is constructed to close the gap.

Fig 4

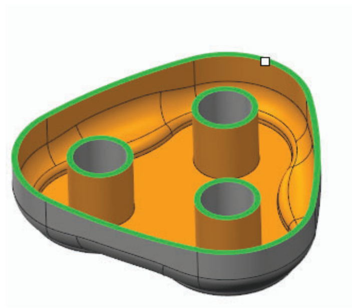


Fig 5

Profile Rib Feature

A Profile Rib feature is a thin fin or web protrusion that attaches to solid surfaces in your design. Typically, these ribs are designed to strengthen parts in your design and are often used to prevent unwanted bending. You can create a profile rib by defining the feature cross section between two perpendicular surfaces.

Profile Rib features are only available in Part mode, and they are subject to normal feature operations, including patterning, modifying, editing references, and redefining.

Designing a Profile rib feature requires you to:

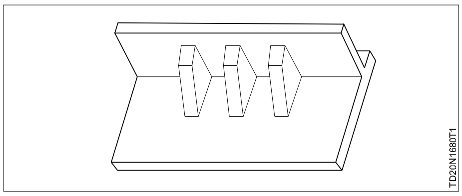
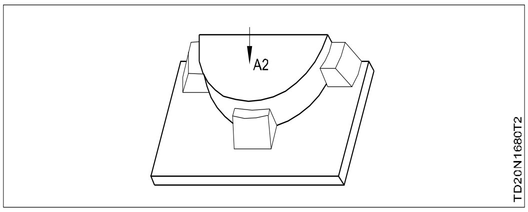
- Create a dependent section by selecting a Sketch feature from the Model Tree, or to sketch a new independent section.
- Determine the rib material side with respect to the sketching plane and desired rib geometry.

You can enter the Profile Rib tool and begin designing your rib feature under the following conditions:

- Sketch Not Selected—Entering the Rib tool and then selecting an existing sketch or creating a new sketch for the Rib feature.
- Sketch Selected—Selecting an existing sketch for the Rib feature and then entering the Rib tool.

In either case, after you designate a sketch for the rib, the validity of your sketch is examined and, if valid, it is placed in the collector. The reference collector only accepts one valid rib sketch at a time.

There are two types of profile rib features available. However, the type is automatically set according to the attaching geometry.

Straight	Attaches to straight surfaces.Extrudes either to one side or symmetrically about the sketching plane.	
Rotational	Attaches to surfaces of revolution. The angled surface of the rib is conical, not planar.Revolves the section about the axis of the parent, making a wedge either to one side or symmetrically about the sketching plane. The wedge is then trimmed with two planes parallel to the sketching surface. The distance between the planes corresponds to the thickness of the rib and attaching geometry.	

The Pattern Feature

A pattern consists of multiple instances of a feature. Select a pattern type and define dimensions, placement points, or a fill area and shape to place the pattern members. The result of the operation is a feature pattern. When you pattern this feature pattern, the result is a feature pattern pattern. You cannot pattern either a group pattern or a feature pattern pattern.

For most pattern types, the feature or feature pattern selected for patterning is the pattern leader. After you pattern the selected feature or feature pattern, the pattern leader that you selected is the pattern header while the instances are pattern members. To copy, mirror, and move patterns, you must select the pattern header instead of the pattern members.

You can create a Reference Pattern of a feature that references the geometry of any pattern member. The feature selected for patterning does not need to be on the pattern leader.

You can mirror a pattern, group pattern, or pattern of a pattern. You can also apply move or rotational transformations to a pattern, group pattern, or pattern of a pattern.

Patterns offer the following benefits:

- Creating a pattern is a quick way to reproduce a feature.
- A pattern is parametrically controlled. Therefore, you can modify a pattern by changing pattern parameters, such as the number of instances, spacing between instances, and original feature dimensions.
- Modifying patterns is more efficient than modifying individual features. In a pattern, when you change dimensions of the original feature, the whole pattern is updated.
- It may be easier or more effective to perform operations once on the multiple features contained in a pattern, rather than on the individual features. For example, suppressing a pattern or adding it to a layer.

You can only pattern a single feature. To pattern several features, create a local group and pattern this group. You can choose Unpattern from the shortcut menu when a group pattern or pattern pattern of the following types is created:

- Dimension
- Table

The result is a set of groups or feature patterns.

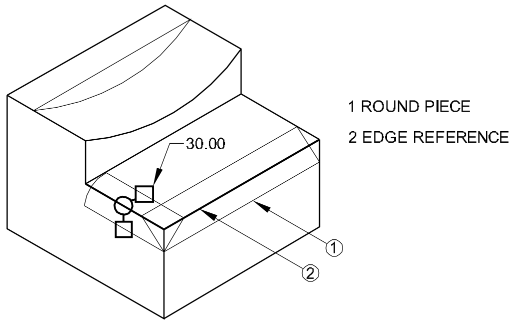
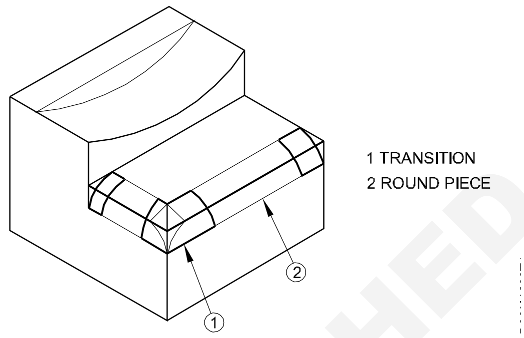
- The line style attributes of a datum curve are not transferred to its pattern members.• A thin feature "remembers" the surface to which it is attached and patterns to this surface.

The Round Feature

A round is an edge treatment feature in which a radius or chord is added to one or more edges, an edge chain, or the space between surfaces. Surfaces can be solid model surfaces, or they can be zero-thickness quilts or surfaces.

To create rounds, you must define one or more round sets. A round set is an organizational unit containing one or more pieces of round geometry. After you specify round placement references, default attributes, radius or chord values, and transitions are used to create a round that best fits the selected geometry. When you select multiple references, the round propagates across tangent neighbors until it encounters a break in tangency.

Round Sets and Transitions

Set Mode Display	Transition Mode Display
Two edge references are selected for the round set. Preview geometry of the two round pieces and the radius or chord value are displayed.	All transitions for the round feature are displayed. Two round pieces are displayed for context.
	

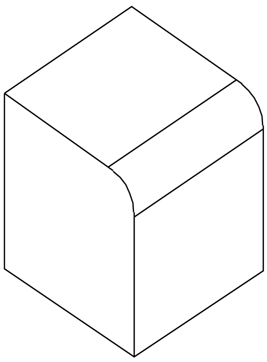
About Round Types

You can create the following round types:

- Constant

The round piece has a constant radius (Fig 6).

Fig 6

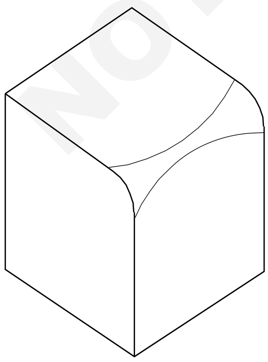


TD20N168056

- Variable

The round piece has multiple radii (Fig 7).

Fig 7

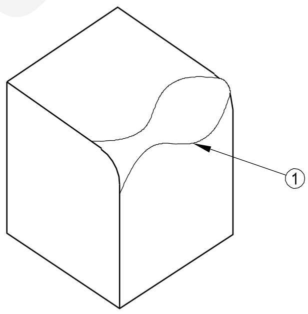


TD20N168057

- Round driven by a curve

The radius of the round is driven by the datum curve (Fig 8).

Fig 8



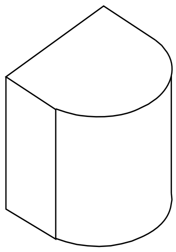
TD20N168059

1 Datum curve

- Full

The full round replaces the selected surface (Fig 9).

Fig 9



TD20N168059

The chamfer feature

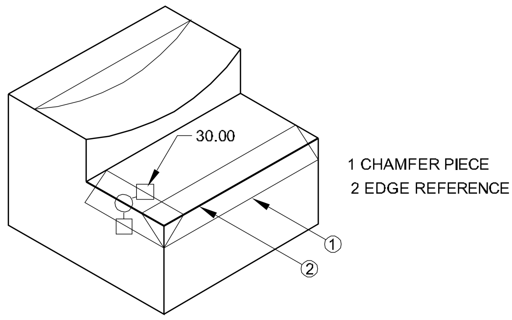
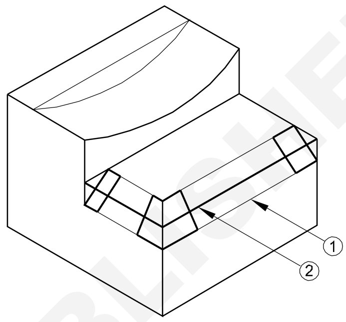
You can create and modify chamfers. Chamfers are a type of feature where an edge or corner is beveled. Surfaces can be solid model surfaces or traditional Creo zero-thickness quilts and surfaces. You can create two types of chamfers: corner chamfers and edge chamfers.

Corner Chamfers

You create corner chamfers using the **Corner Chamfer** tool. When you create a corner chamfer, you select a vertex defined by three edges, and then you set length values along each chamfer direction edge.

Edge Chamfers

You create edge chamfers using the **Edge Chamfer** tool. To create edge chamfers, you define one or more chamfer sets. A chamfer set is an organizational unit containing one or more chamfer pieces (chamfer geometry).

Set Mode Display	**Transition Mode Display**
Two edge references are selected for the chamfer set. Preview geometry of the two chamfer pieces and the distance value is shown.	All transitions for the entire edge chamfer feature are displayed. The two chamfer pieces are shown for context.
	

The hole feature

The Hole tool enables you to add simple, custom, and industry-standard holes to your models. You add holes by defining a placement reference, offset references, optional hole orientation references, and the specific characteristics of the hole.

As you work, the system displays preview geometry of the hole. Notice that a hole always begins at the placement reference and extends to the specified depth. You can directly manipulate and define the hole in the graphics window and on the Hole tab.

You can create the following hole types:

- Simple—Consists of an extruded or revolved cut that is not directly associated with any industry standard. You can create the following Simple holes types:
- Predefined rectangle profile—Uses (straight) geometry predefined by the system. By default, the system creates one-sided Simple holes. However, you can create two-sided Simple Straight holes by using the Shape tab. Two-sided Simple holes are typically used in assemblies and enable you to simultaneously format both hole sides.
- Standard hole profile—Uses standard hole profile as drill hole profile. You can specify the countersink, counterbore, and tip angle for the holes.
- Sketched—Uses a sketch profile that you create in Sketcher.
- Standard—Consists of a revolved cut based on industry-standard fastener tables. Creo provides industry-standard hole charts and tapped or clearance diameters for the selected fastener. You can also create your own hole charts. The thread notes are automatically created for Standard holes. You can separate the hole axis from the hole thread surface and place the thread in a specified layer. You can create the following types of Standard holes:

- — Tapped hole
- — Tapered hole
- — Drilled hole
- — Clearance hole

Hole Features vs Cut Features

Hole features are different from cut features in the following ways:

- Hole features use a predefined placement scheme that can be more desirable than the dimensioning scheme of the cut.
- Simple Straight holes and Standard holes do not require a sketch, unlike cut features.

The Draft Feature

The Draft feature adds a draft angle from -89.9° to +89.9° to individual surfaces or to a series of surfaces. You can draft only the surfaces that are formed by tabulated cylinders or planes. To access the Draft feature, click Model > Draft.

You can draft either solid surfaces or quilt surfaces, but not a combination of both. When you select surfaces to be drafted, the first selected surface determines the type of additional surfaces, solid or quilt, that can be selected for this feature.

For drafts, the system uses the following terminology:

- Draft surfaces—The surfaces of the model that are being drafted.
- Draft hinges—Lines or curves on the draft surfaces that the surfaces are pivoted about (also called neutral curves). Draft hinges can be defined by selecting a plane or quilt, in which case the draft surfaces are pivoted about their intersection with this plane, or by selecting individual curve chains on the draft surfaces.
- Pull direction (also called draft direction)—Direction that is used to measure the draft angle. This is usually the direction of mold opening. You can define it by selecting a plane (in which case the pull direction is normal to this plane), a straight edge, a datum axis, or an axis of a coordinate system.
- Draft angle —The angle between the pull direction and the resulting drafted surfaces. If the draft surfaces are split, you can define two independent angles for each side of the drafted surface. Draft angles must be within the range of -89.9° to +89.9°.

Draft surfaces can be split either by the draft hinge or by a different curve on the draft surface, such as an intersection with a quilt, or a sketched curve. If you are splitting by a sketch, the system extrudes it to a quilt within the draft feature, in the direction normal to the sketching plane. If the draft surfaces are split, you can:

- Specify two independent draft angles for each side of the drafted surface.
- Specify a single draft angle, with the second side drafted in the opposite direction.
- Draft only one side of the surface (either one), with the other side remaining in the neutral position.

When the surfaces to be drafted include rounds, you can either preserve the rounds, and they remain rounds, or you can draft the rounds, and then they become conic.

If you propagate a draft, the system expands the selection of surfaces to be drafted. The draft adds surfaces until a sharp edge or a surface that cannot be drafted is reached. The draft_tan_propagation_default configuration option defines the default of whether to propagate a draft or not. The command defines whether to propagate a draft f or the draft feature.

Assembly

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

- **describe the process of assembly in creo.**

To create a subassembly or an assembly, you must first create datum features or a base component. You can then create or assemble additional components to the existing component(s) and datum features.

Assembling Components

You can add components to an assembly in the following ways:

- Assemble a component parametrically by specifying its position relative to the base component or other components and/or datum features in the assembly.
- Assemble components manually or automatically using predefined component interfaces. Refer to About Automatic Placement of Components for more information.
- Assemble a component nonparametrically using the Package command in the Assembly group. Use packaging as a temporary means to include the component in the assembly; then finalize its location with assembly instructions.
- Create a part or subassembly directly in Assembly.
- You can use notebooks and specify declarations to assemble components automatically. You create these assemblies by automatically aligning datum planes and axes of different parts in accordance with the declarations previously made in Notebookt and Part modes. You can specify declarations, and after a component has a declaration, it can be automatically assembled.
- You can include a component as a member of an assembly without actually placing it in the assembly window. This technique allows you to list the component as a member of the assembly even if the component is not ready to be assembled (for example, it does not have geometry). The system lists included components in the Model Tree and BOM, but does not display them on the screen or include them in mass property calculations. To add constraints later, you can redefine the placement of the component.

You can remove a component from an assembly by deleting it or replacing it with another component. In addition, you can also redefine the placement constraints for assembled components.

To place a base component or feature, you must either create three orthogonal datum planes as the first feature, assemble an existing component (part, subassembly, or skeleton model), or create a base component.

A component that is added into an assembly is saved in the assembly directory.

Using Datum Planes as the First Feature: When you create three orthogonal datum planes as the first feature in an assembly, you can assemble a component with respect to these planes, or create a part in Assembly as the first component. Using datum planes as the first feature has the following advantages:

- You can redefine the placement constraints of the first assembled component.
- You can pattern the first component you add, creating a flexible design.
- You can reorder subsequent components to come before the first one (if the components are not children of the first component).

Assembling a Component Parametrically: Using the Component Placement tab, you can assemble components parametrically by establishing constraints that define the component's position in the assembly. The component's position changes according to changes in components or assembly features to which it is constrained.

Creating a Base Component: If you do not create three orthogonal datum planes, the base component is the first part, subassembly, or skeleton model placed into an assembly. In many ways it is like the base feature of a part. The initial assembly units are the same as the units of the base component. When a base component is the first object in an assembly (before any assembly features), no placement constraints are defined. You place the component with the Default constraint. If you replace a base component with interchangeable components, the replacing components will always be placed by default as well.

When you create the first component of an assembly, you can either create an empty component or copy from an existing component. As with an assembled base component, the initial assembly units are the same as the base component, and interchange components that replace the created base component will always be in the default orientation.

Creo parametric drawing

Objectives : At the end of this lesson you shall be to
- **list the functionally of engineering drawing modes in creo parametric**
- **brief the steps involved in creating a drawing**
- **explain the methods of adding models and opening a existing models in drawing.**

About the Creo Parametric Drawing Mode: Creo Parametric offers functionality to work with engineering drawings in Drawing mode with the Detail module. Use this module to perform the following operations:
- Create drawings of all Creo Parametric models.
- Export drawing files to other systems and import files into Drawing mode.
- View and annotate models and drawings.
- Manipulate dimensions, and use layers to manage the display of different items. Creo Parametric associates drawings with their parent models. The model automatically reflects any dimensional changes that you make to a drawing. In addition, corresponding drawings also reflect any changes that you make to a model (such as the addition or deletion of features and dimensional changes) in Part, Sheetmetal, Assembly, or Manufacturing modes.
- Work with various view types. All views in the drawing are associative. If you change a dimensional value in one view, other drawing views are updated accordingly.
- Work with multiple sheets in the drawing.
- Add and modify different kinds of textual and symbolic information.
- Define mapkeys for drawing operations. Execute the mapkeys in any of the tabs irrespective of whether the tab is active or not.
- Customize engineering drawings with sketched geometry, create custom drawing formats, and make multiple cosmetic changes to drawings.
- Use shortcut menus to modify an object in a drawing from anywhere in the Model Tree, Drawing Tree, or the graphics window. At any time when a drawing window is active, you can interrupt your current process and activate a drawing object for modification.
- Create 3D parametric, feature-based models from 2D drawing data using AutobuildZ. The 2D drawing data can be imported to Creo Parametric from any of the supported file formats, such as DXF, DWG, and IGES. Additionally you can use the following capabilities of AutobuildZ:
- Clean up 2D data by moving unnecessary and unwanted data to blanked layers.
- Associate the parts with the active drawing in session.
- Create features in the active part by selecting geometric entities on the drawing in session.
- Map the views of the drawing to datum planes in the parts.
- Automatically regenerate the drawing views of the active part on a separate drawing sheet when features are added to the active part.

Creating a Drawing: When you start a new drawing, you specify a 3D model file in which to place drafting views.

1 Click File > New. The New dialog box opens.

2 Click Drawing and type a name in the File name box or use the default. Click OK. The New Drawing dialog box opens.

3 In the Default Model box, type the name of a model in the working directory. If you started the new file from an open 3D file, the 3D file name appears by default. The selected model is set as the current drawing model.

4 Under Specify Template, do one of the following:

- To use a drawing template, click Use template and select a template from the list.
- To create a drawing without a template but with an existing format, click Empty with format. Under Format, specify the format you want to use.
- Specify the drawing size or retrieve a format. To specify the size, do one of the following:

Click Portrait or Landscape in the Orientation box and select a standard size from the Standard Size list.

Alternatively,

Click Variable in the Orientation box to define both the height and width dimensions. Select Inches or Millimeters and type values in the Width and Height boxes.

To retrieve a format, select Retrieve Format and select a name from the Name list in the Format box. You can also type [?] or click Browse to select a name from the Open dialog box.

5 Click OK. The new drawing opens.

- If the part that you are using to create the drawing has simplified representations, the Open Rep dialog box opens. Select the required representation and click OK. The new drawing is created with the selected representation set as the current representation of the drawing model.
- If you are using the default template to create the drawing of a part that has simplified representations, the default representation is used to create the drawing. The Open Rep dialog box opens only when you are creating a new empty drawing.
- If a model has multiple instances defined with a Family Table, you are prompted to choose an instance from the Select Instance dialog box before you select the representation using the Open Rep dialog box.

Adding Models to the Drawing: Before you can place a view of a 3D model, you must associate the 3D model with the drawing. This is called adding the model to the drawing.

When you start creating a new drawing file you are prompted in the New File dialog box for a 3D model to reference. Once the drawing is in progress, use the following procedure to add other models to the drawing file:

1 Open a drawing to add a drawing model.

2 Perform one of the following operations:

- Click Drawing Models in the Model Views group of the Layout tab.
- Right-click and click Drawing Models on the shortcut menu.

The DWG MODELS menu appears on the Menu Manager.

3 Click Add Model. The Open dialog box opens and lists all the files in your current working directory.

4 Select the model to add as a drawing model to the current drawing, and click Open.

- If the new model that you are adding to the drawing contains part simplified representations, the Open Rep dialog box opens. Select the required representation and click OK.
- If the model that you are adding to the drawing has different instances of a Family Table, you are first prompted to select an instance from the Select Instance dialog box, and then the simplified representation from the Open Rep dialog box.

5 The model is added to the drawing and set as the current drawing model. If you have selected a representation in the Open Rep dialog box, the selected representation is set as the current representation of the drawing model.

Adding a model to the drawing does not place a view of the model on the sheet, but lets the drawing reference the model so that you can place a view. You can insert drawing views of the newly added part as you would for any other drawing model.

Opening a Model from a Drawing

To open a model from within Drawing mode, you can use File > Open, or you can use the Model Tree.

1 Click File > Open. The File Open dialog box opens.

2 Navigate to and select the model you want to open, and then click Open.

The model is retrieved into the current session in a separate window.

Using the Model Tree

1 In the Model Tree, select the model you want to retrieve, and then right-click to display the shortcut menu.

2 Click Open from the shortcut menu. The model is retrieved into the current session.

Surface Modelling

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

- **describe the method of creativity an Extruded Surface**
- **explain the method of creativity a Revolved Surface**
- **explain method of creativity a Sweep Surface**
- **state the method of creativity a Blended Surface**
- **describe the method of creativity a Swept Blend Surface**
- **explain the method of creativity a Helical Sweep Surface.**

Surface modeling

Surface models are a type of three-dimensional (3D) models with no thickness. These models are widely used in industries like automobile, aerospace, plastic, medical, and so on. Surface models should not be confused with thick models, that is, models having mass properties. Surface models do not have thickness whereas thick or solid models have a user-defined thickness. In Pro/ENGINEER, the surface modeling techniques and feature creation tools are the same as that used in solid modeling. A solid model of any shape that is created can also be created using the surface modeling techniques. The only difference between the solid model and the surface model will be that the solid model will have mass properties but the surface model will not. Sometimes, complex shapes are difficult to create using solid modeling. Such models can be easily created using surface modeling and then the surface model can be converted into the solid model. It becomes easy for a person to learn surface modeling if he is familiar with solid modeling feature creation tools.

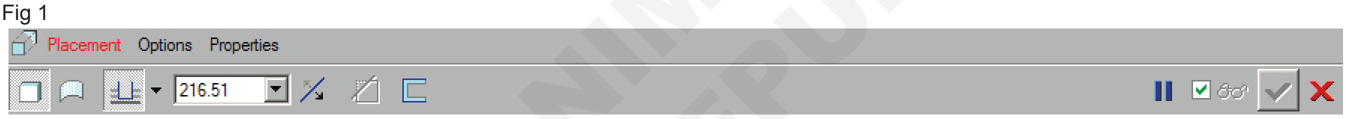
Creating surfaces in pro/engineer wildfire 2.0

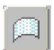
In Pro/ENGINEER Wildfire 2.0, a sketch can be toggled between a solid model and a surface model. The two tool buttons that are used to toggle between the solid feature and a surface feature are available on dashboards.

Creating an Extruded Surface

To create an extruded surface, choose the Extrude Tool button from the Base Features toolbar. The Extrude dashboard is displayed as shown in Fig 1.

Fig 1



 In this dashboard, the Extrude as solid button is selected by default. Select the Extrude as surface button to extrude the sketch and create a surface model. Attributes are sketch plane, both-side or one-side extrusion, depth of extrusion, and so on are same for the surface model as solid model.

A surface model can be extruded with capped ends or with open ends. To create the capped end surface model, the sketch should be a closed loop. Otherwise, a surface can be created with the open sketch Fig 2&3.

To create a surface with capped ends, select the Capped Ends check box in the Options slide up panel.

Creating a Revolved Surface

To create a revolved surface, choose the Revolve Tool button from the Base Features toolbar.

The Revolve dashboard is displayed as shown in Fig 4. This feature creation tool works in the same way as in the case of solid modeling.

Fig 2

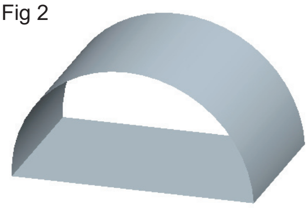


Fig 3

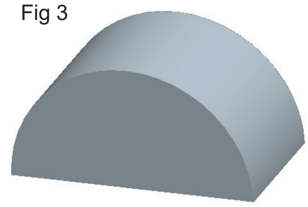
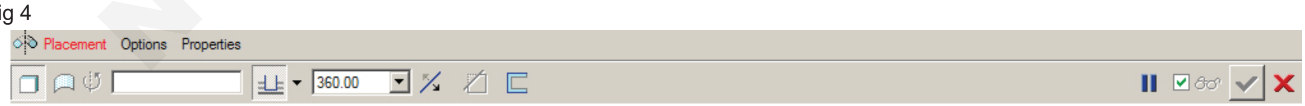


Fig 4



 The Revolve as solid button is selected by default; choose the Revolve as surface

button to create a revolve surface. You can create a revolved capped end surface or an open end surface. The Capped End check box in the Options slide-up panel is available only when the sketch is closed and the angle of revolution is less than 360-degrees. Fig 5 shows the open end revolve surface and Fig 6 shows the capped end revolve surface.

Fig 5

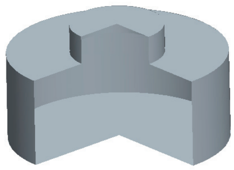
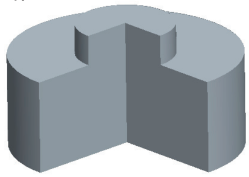


Fig 6



Creating a Sweep Surface

To create a sweep surface feature, choose Insert > Sweep > Surface from the menu bar. The SWEEP TRAJ menu is displayed. The method to create a surface sweep feature is the same as that to create a solid sweep feature. The additional option of capping the ends is also available in the Sweep option.

Fig 7 and 8 show the sweep surfaces with the open and closed ends respectively.

Fig 7

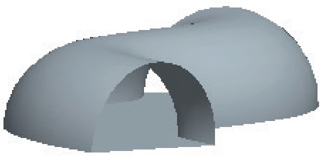
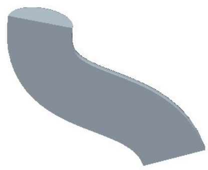


Fig 8



Creating a Blended Surface

To create a surface blend, choose Insert > Blend > Surface from the menu bar. The BLEND OPTS menu is displayed. The method to create a blended surface is the same as that to create a solid blend. To create a solid blend feature, refer to Chapter 7. Blended surfaces can be with open ends or capped ends. Fig 9 shows the blended surface with open ends and Fig 10 shows the blended surface with capped ends.

Fig 9

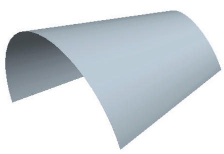
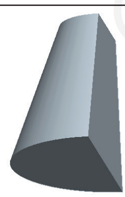


Fig 10



Creating a Swept Blend Surface

To create a swept blend surface, choose Insert > Swept Blend > Surface from the menu bar.

The BLEND OPTS menu is displayed. The method to create a swept blend surface is the same that to create a solid swept blend feature. Fig 11 shows the swept blend with open ends and Fig 12 shows the swept blend with capped ends.

Fig 11

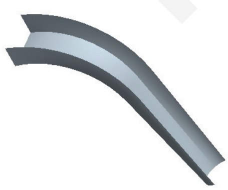
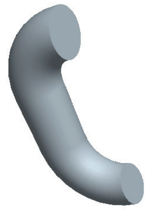


Fig 12



Creating a Helical Sweep Surface

To create a surface helical sweep, choose Insert > Helical Sweep > Surface from the menu bar.

The Surface dialog box and the ATTRIBUTES menu is displayed. The method to create a helical sweep surface feature is the same as that to create a solid helical sweep feature. Fig 13 shows the helical sweep surface with open ends and Fig 14 shows the helical sweep surface with capped ends.

Creating a Surface Using Variable Section Sweep

Fig 13

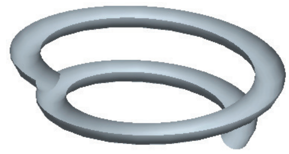
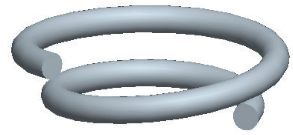
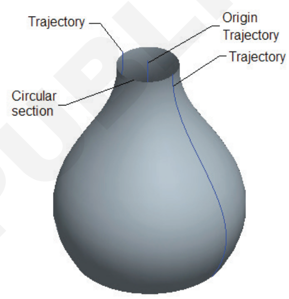


Fig 14



To create a surface by variable section sweep, choose Insert > Variable Section Sweep from the menu bar. The Variable Section Sweep dashboard is displayed. The procedure to create a variable section sweep feature or surface is the same as was discussed in Chapter 8. Fig 15 shows the trajectories and section that are used to create the variable section sweep surface. You have an option to keep the ends open or capped. This option is available in the Options slide-up panel.

Fig 15



Creating surfaces using style environment of Pro/ Engineer / CREO

Style is an environment available in Pro/ENGINEER that is used to draw free style curves and create surfaces by joining them. The surfaces created using the Style environment are called Super features. This is because these features can contain any number of curves or surfaces. The Style surfaces can be joined with the Pro/ ENGINEER surfaces. They can have the parent-child relationship among themselves and as well as with Pro/ ENGINEER features. To enter the Style environment, choose the Style Tool available in the Base Features toolbar or choose Insert > Style from the menu bar. Fig 16 shows the appearance of the Style environment.

Style Tools Toolbar

Fig 17 shows the Style Tools toolbar available in the Style environment. The tools available in this toolbar are discussed next.

Select Button

This button is used to select the surfaces, curves, planes, and so on in the Style environment. If you are in middle of a feature creation tool you can choose the Select button to exit that tool.

Fig 16

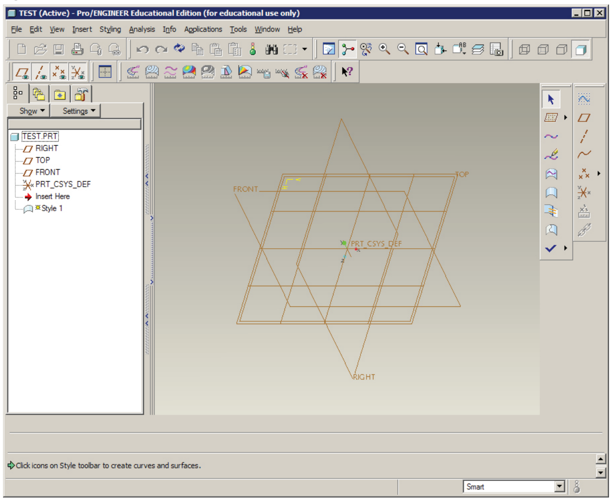
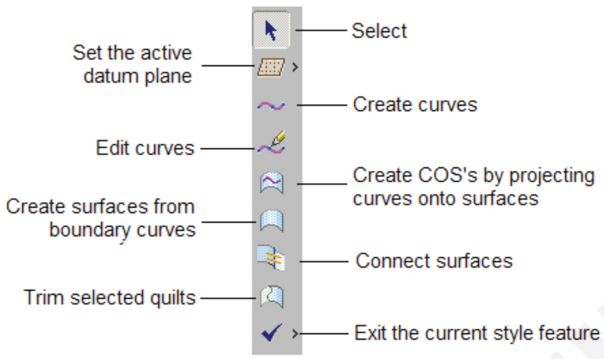


Fig 17



Set the active datum plane Button

This button is used to select the datum plane on which the drawing or the editing operation needs to be performed. The datum plane that you select is highlighted by a mesh.

Create Internal Datum Plane Button

This button is chosen by selecting the black arrow on the right of the Set the active datum plane button. When you select the arrow, the flyout is displayed. Choose the Create Internal Datum Plane button to create an internal datum plane in the Style environment. When you choose this button the DATUM PLANE dialog box is displayed. This dialog box is used to create a datum plane. The datum planes are named as DTM1, DTM2, and so on.

It should be noted that the datum planes created using this button are displayed in the graphics window only when you are in the Style environment. Once you exit the Style environment, the datum plane becomes invisible. Any feature created in the Style environment is displayed in the Model Tree as a Style feature.

Create Curves Button

This button is used to draw curves. When you choose this button, the Curve dashboard is displayed as shown in Fig 18.

The options in this dashboard are discussed next.

Fig 18

Free Radio Button

When the Curve dashboard is displayed, the Free radio button is selected by default. The

prompt in the Message Area reads "Click to define points for the curve (SHIFT to snap)".

To create curve, click on the screen. A yellow point is displayed at the location where you clicked. Now, again click to define the second point of the curve. The two points are joined. When you click to define the location of the third point, you will notice that the curve that you are drawing is defined by a spline. After defining the points, press the middle mouse button to create the curve. While specifying a point if you press the SHIFT key then the point is snapped to the entity already present on the screen.

Remember that the curve drawn using the Free option is created on the active datum plane. To draw a 3D curve you need to snap the point on the existing entity. You can also draw a 3D curve by choosing the Toggle showing all views and one view full-size button from the Style toolbar. When you choose this button, the display is turned into four windows. In Pro/ENGINEER, this type of display is called a 4-view display mode. The four views show the top, default, right-side, and front views. You can select a point in one window and then select the second point in the other window. By specifying points in different windows, the 3D curve can be drawn. To switch back to the single window display mode, choose the Toggle showing all views and one view full-size button.

Planar Radio Button

This radio button when selected allows you to create the curve on the datum plane that is highlighted by the mesh. This datum plane is called the active plane. The active plane can be selected before invoking the Curve dashboard by choosing the Set the active datum plane button.

COS Radio Button

This radio button is used to draw curves on surfaces. The points that you define on a surface are constrained to that surface. When you click to define the location of the first point of the curve, the point is placed. Now, this surface is selected and the points placed hereafter should lie on the same surface. If you click outside this surface then the point is not placed on the surface. After the curve is drawn, press the middle mouse button. The red curve is converted to a white curve indicating that the curve is completed. The curve drawn on the surface is the child of the surface.

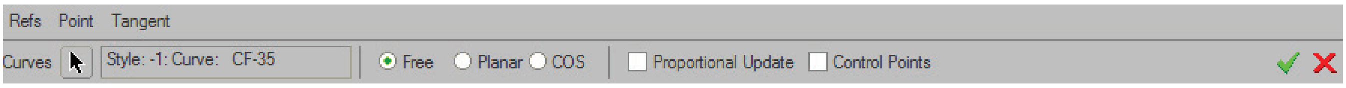
Control Points Check Box

While drawing the curve, if this check box is selected, then while editing the curve thecontrol points are displayed.

Edit curves Button

This button is used to edit the curves that are created as style features. When you choose this button the Edit curve dashboard is displayed and you are prompted to select a curve. When you select a curve to edit, the Edit curve dashboard appears as shown in Fig 19.

Fig 19



The options in the Edit curve dashboard are discussed next.

Curve collector

When you select a curve to edit, the id of the curve is displayed in this collector.

Free radio button

If the curve that is selected for editing was drawn using the Free option, then this radio

button is selected by default.

Planar radio button

If the curve that is selected for editing was drawn using the Planar option, then this radio button is selected by default.

COS radio button

If the curve that is selected for editing was drawn using the COS option, then this radio button is selected by default.

Proportional Update check box

If the curve that is selected for editing was drawn using the Proportional Update option, then the curve is edited proportionately with the points.

Control Points check box

If the curve that is selected for editing was drawn using the Control Points option, then the control points are displayed on the curve. Using these control points you can modify the shape of the curve.

By default, a curve has a natural contact with the adjacent surface. This is evident from the check mark on the left of the Natural option in the shortcut menu. Fig 20 shows the curve that is connected to the adjacent surface using the Natural option. The curve is drawn using the Free option. The point on the cylindrical surface is selected by using SHIFT+left mouse button and similarly another point is selected on the surface at the base. Fig 21 shows the curve whose contact type is changed to the Surface Tangent option by choosing it from the shortcut menu.

Creating COS's by projecting curves onto surfaces Button

Using this button, a curve created in the Style environment can be projected onto the selected surface.

Fig 20

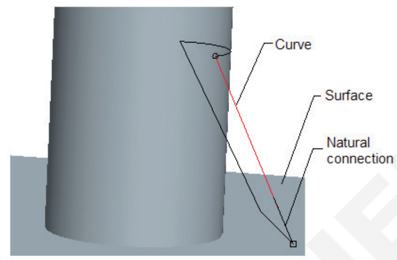
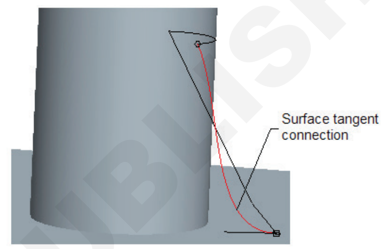


Fig 21



Create surfaces from boundary curves Button

This button is used to create a surface among a closed boundary of curves. When you choose this button the Boundary Surfaces dashboard is displayed and you are prompted to select three or four boundary curves to define a surface. Select the four curves as shown in Fig 22. After selecting the four curves, press the middle mouse button. The surface is created as shown in Fig 23.

Fig 22

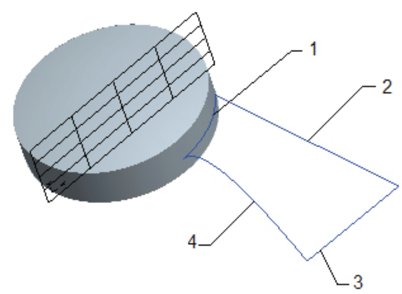
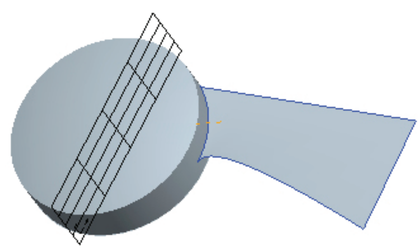


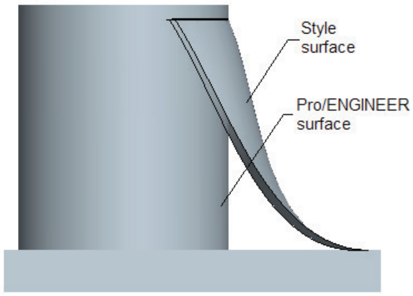
Fig 23



Connect surfaces Button

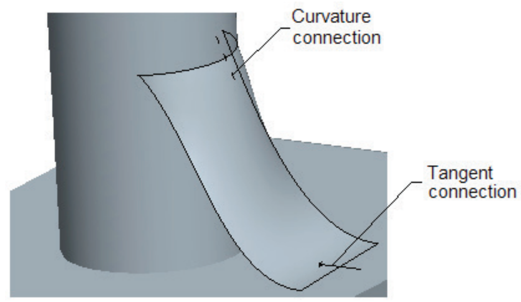
When you choose this button, the Connect surfaces dashboard is displayed and you are prompted to select the two surfaces. The Style surface can be connected to the Pro/ENGINEER surface. When you select the two surfaces shown in Fig 24 and press the middle mouse button, the connections are automatically applied to the two surfaces.

Fig 24



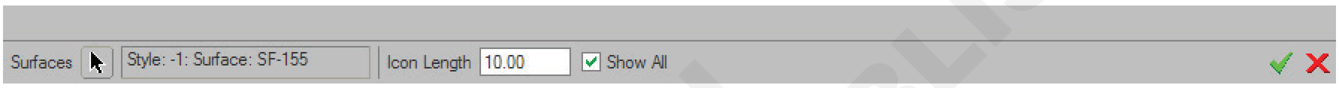
These connections may be of two types: curvature connection represented by a dashed line and the tangent connection represented by an arrow. If the tangent connection is applied then the arrow is displayed and if the curvature connection is applied then a dashed line is displayed on the surfaces. Fig 25 shows the two surfaces where the tangent connection is applied and is not applied.

Fig 25



After surfaces are selected, the Connect surfaces dashboard is displayed as shown in Fig 26. To apply the connection, click on any one end of the dashed line. The dashed line is converted to an arrow, indicating that the two surfaces are connected. To remove the connection, use SHIFT+left click on the arrow.

Fig 26



Trim selected quilts Button

This button is used to trim a surface. When you choose this button, the Trim dashboard is displayed and you are prompted to select the surface(s) to trim. Select the surface so that it turns pink in color and then press the middle mouse button. Now, you are prompted to select the curve that will be used to trim the surface. Select the curve and press the middle mouse button. The selected surface is highlighted in two portions. Select the portion to delete. Choose the green check mark to exit the trim tool.

Fig 27 shows the surface and the curve that are selected for trimming. This figure also shows the surface divided into two portions. The portion defined by the curve is selected to delete. Fig 28 shows the surface after trimming.

Fig 27

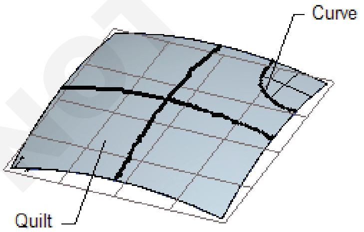
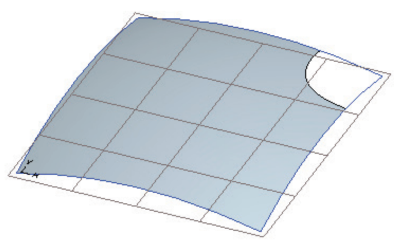


Fig 28



SURFACE EDITING TOOLS IN Pro/ENGINEER WILDFIRE

The surface editing tools help in decreasing the modeling time. They also help in creating

complex surface models. The surface editing tools that you will be learning in the next section

are as follows:

1 Mirror	2 Merge	3 Trim
4 Fill	5 Intersect	6 Offset
7 Thicken	8 Solidify	9 Vertex Round

Mirroring the Surfaces

The Mirror Tool is used to mirror the surface about a plane. This tool is available in the Edit Features toolbar only when a surface is selected. When you choose this button, the Mirror dashboard is displayed as shown in Fig 29.

Using the References tab you can choose the mirroring plane. The Copy as Dependent check box is selected by default in the Options tab. The makes sure that the Parent-child relationship is maintained between the mirrored and the original surface. Fig 30 shows the mirror plane about which the surface is mirrored as shown in Figure 31.

Merging the Surfaces

The Merge Tool is used to merge the two surfaces and make them a single surface. A surface is also known as a Quilt. To convert a surface to a solid, it is necessary that the surfaces are merged. While merging the surfaces, this tool also trims the surfaces. This tool is available in the Edit Features toolbar only when the two surfaces to be merged are selected. When you choose the Merge Tool button, the Merge dashboard is displayed as shown in Fig 32.

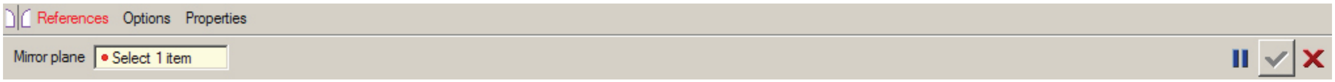


Fig 29

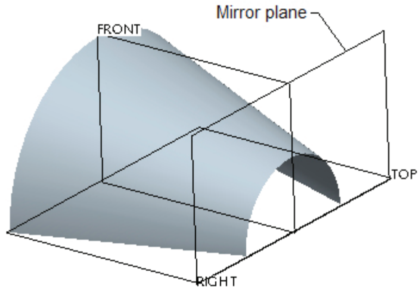


Fig 30

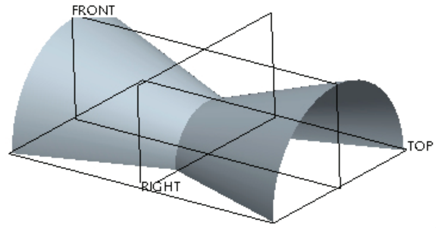


Fig 31

Fig 32

The following steps explain the procedure to merge the surfaces shown in Fig 33.

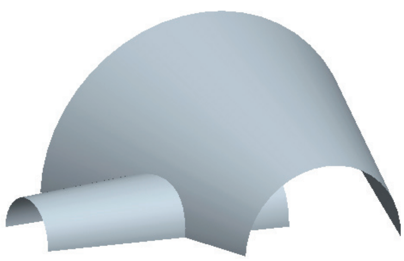


Fig 33

1 Select the Quilts option from the Filter drop-down list. Select the two surfaces and when the surfaces turn pink in color, choose the Merge Tool; the Merge dashboard is displayed. In this figure, the part of the surfaces that will be retained after the two surfaces are merged is highlighted by yellow dots on it. The yellow arrows points to show the side of the surface to keep. The direction of yellow arrow can be toggled by using the Change side of first quilt to keep and the Change side of second quilt to keep buttons available on the Merge dashboard.

2 Choose the Change side of first quilt to keep button and then choose the Change side of second quilt to keep button. Notice that the outer side of the surfaces are highlighted with yellow dots as shown in Fig 34.

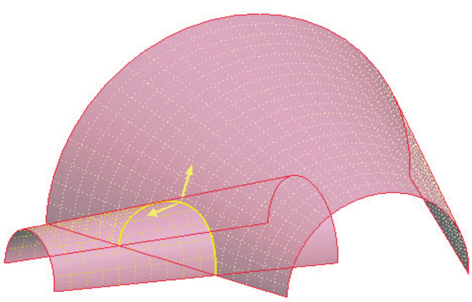


Fig 34

3 Choose the Preview button and then exit the dashboard. The resulting merged surface is shown in Fig 35. This merged surface is a single surface and now can be converted to a solid feature.

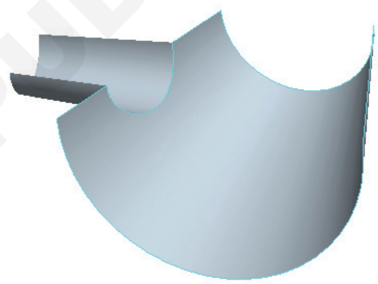


Fig 35

The Reference tab of the Merge dashboard shows the selected quilts. In the Options tab, you can select between Intersect and Join options. The Join option can be used when the edge of one quilt lies on the other quilt.

Trimming the Surfaces

As the name suggests, the Trim Tool is used to trim the selected surfaces using a trimming object. You need to select the surface that you need to trim and then choose the Trim Tool button from the Edit Features toolbar. The Trim dashboard is displayed, as shown in Fig 36. You are prompted to select the trimming object. This trimming object can be a curve, plane, edge, or a surface.

The part of the surface that is to be retained is highlighted with yellow dots. A yellow arrow points in the direction of the surface to be retained after trimming. You can choose the Flip between one side, other side, or both sides of trimmed surface to keep button to toggle the direction of yellow arrow. By default, the trimming object is deleted after the surfaces are trimmed. If you need to keep the trimming object, select the Keep trimming surface check box from the Options slide-up panel.

Fig 36

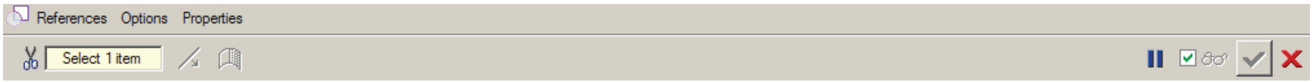


Fig 37 shows the surface selected as the trimming object, the trimming surface, and the yellow arrow. From this figure it is evident that the arrow is pointing toward the right; therefore the right portion of the surface will be retained after trimming. Fig 38 shows the surface obtained after trimming.

Fig 37

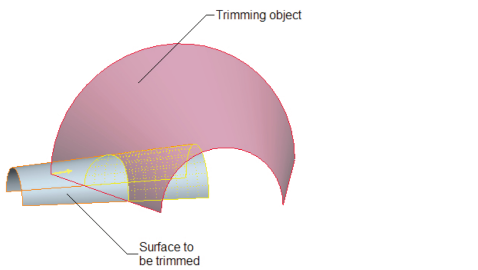
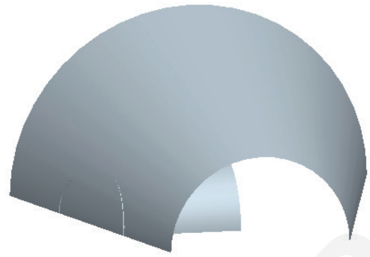


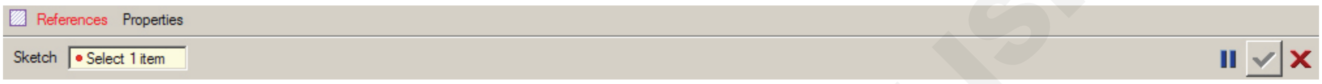
Fig 38



Creating the Fill Surfaces

The Fill option is used to create a planar surface by sketching its boundaries. When you choose this option from the Edit menu in the menu bar, the Fill dashboard is displayed as shown in Fig 39.

Fig 39



From the References slide-up panel choose the Define button to select the sketching plane and drawing the sketch. Fig 40 shows the sketch plane and Fig 41 shows the surface that is created using the Fill option.

Fig 40

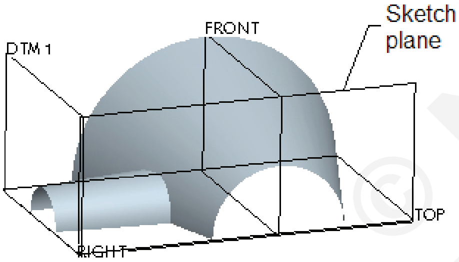
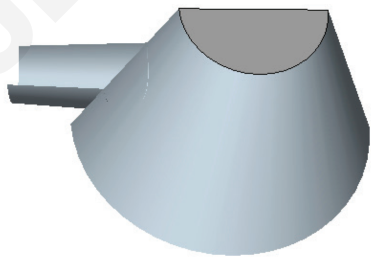


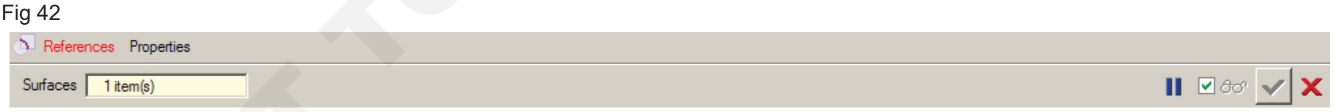
Fig 41



Creating the Intersect Curves

The Intersect option is used to create a curve at the intersection of two surfaces. The intersect curve can then be used for various purposes. The Intersect option is available in the Edit menu only when you have selected a surface. When you choose this option from the Edit menu, the Intersect dashboard is displayed as shown in Fig 42.

Fig 42



When you select the second surface, the intersecting curve is created as shown in Fig 43.

Make sure to select the second surface while holding the CTRL key down. The curve created can be copied, moved, and so on. One of the uses of the intersect curve is shown in Figu 44 and 45. In Fig 44, the intersecting curve is copied at a distance of 150. To create the surface shown in Fig 56, the Boundary Blend Tool is used. To create the boundary blend, the intersecting curve is selected and then the curve edge of the surface is selected. Both the curves are blended and the tangency is increased by dragging the handles.

Creating the Offset Surfaces: A surface can be copied to an offset distance. To offset a surface, select the surface to offset and choose Edit > Offset from the menu bar. The Offset dashboard is displayed, as shown in Fig 46. The Offset option is available only when you have selected a surface to offset.

Fig 43

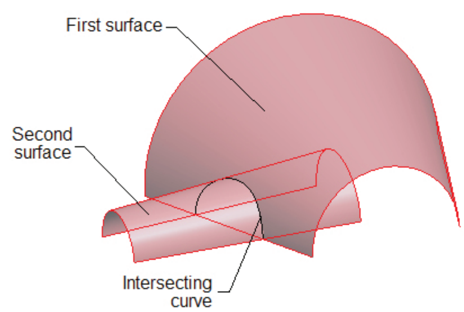


Fig 44

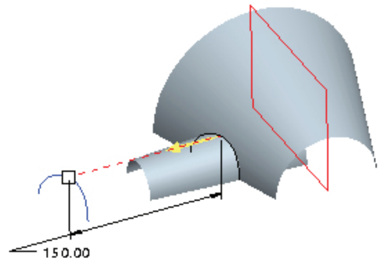


Fig 45

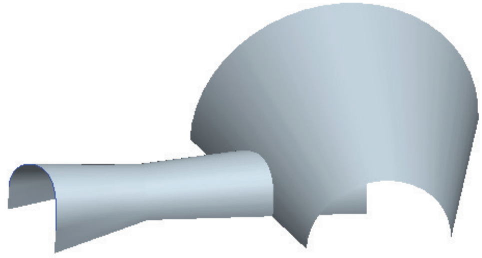
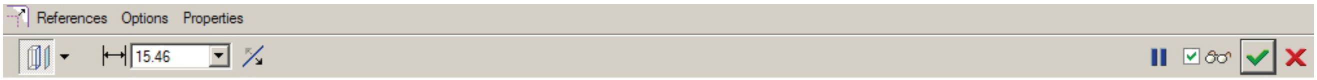


Fig 46



In Pro/ENGINEER, there are three methods to offset a surface. These methods are as follows:

1 Create the offset of the whole surface, using the Standard option.

2 Sketch a section and offset the area inside the section with the draft, using the With Draft option.

3 Sketch a section and offset the area inside or outside the section, using the Expand option.

In the Offset dashboard, first you need to specify the type of offset surface you need to create.

The types of offset that can be created in Pro/ENGINEER Wildfire 2.0 are as follows:

1 Standard Offset Feature

2 With Draft Feature

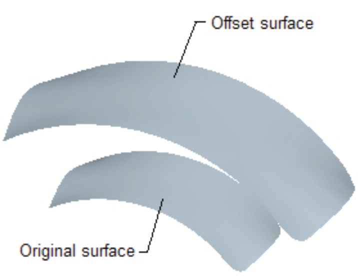
3 Expand Feature

Standard Offset Feature

The Standard Offset Feature button is present at the lower-left corner of the Offset dashboard and is selected by default. You can enter the offset value in the dimension box.

Using this option you can offset the surface as a whole. From the drop-down list present in the Options slide-up panel, you can offset a surface normal to the surface, allow Pro/ENGINEER to automatically fit the surface, or control the direction of the offset in the x, y and z-axes. If you select the Control Fit option, you need to select a coordinate system and specify the direction to offset. From the Options slide-up panel you can select the Create side surface check box to join the offseted surface with the side surfaces. Figure 47 shows the original surface and the offset surface.

Fig 47



With Draft Feature

The With Draft Feature button can be selected form the list of buttons that appear by selecting the black arrow present on the right of Standard Offset Feature button. Using this option you can sketch the section and then give a draft angle to side surfaces. Choose the Define button from the from the References slide-up panel to define a sketch plane and create the sketch. Fig 48 shows the draft offset surface with the Straight radio button selected from the Options slide-up panel. The section that was drawn on the sketch plane was circular. Similarly, Fig 49 shows the draft offset surface with the Tangent radio button selected from the Options slide-up panel.

Fig 48

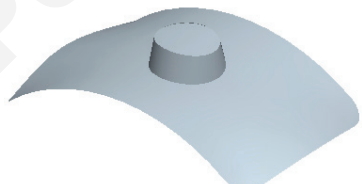
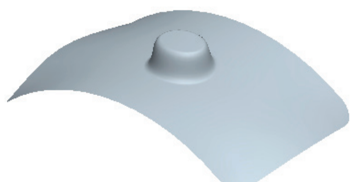


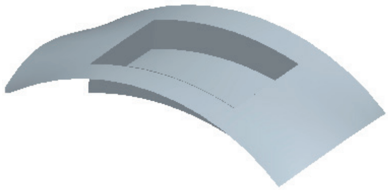
Fig 49



Expand Offset Feature

The Expand Offset Feature button can be selected form the list of buttons that appear by selecting the black arrow present on the right of Standard Offset Feature button. Using this option you can sketch the section and then choose to offset the inside of the sketch or the outside of the sketch. For this purpose you need to choose the Flip the material sides of sketch button from the dashboard. Fig 50 shows the offset surface when the inside of the sketch is selected to offset. The section that was drawn on the sketch plane was rectangular. Choose the Define button from the Options slide-panel to define the sketch plane and create the sketch. Fig 51 shows the draft offset surface when the outside of the sketch is selected to offset.

Fig 50

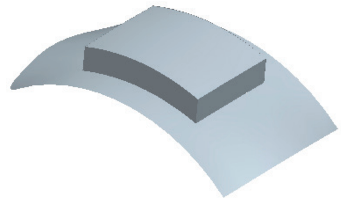


Giving Thickness to a Surface

To add thickness to a quilt or to a surface, select the quilt and choose the Thicken option from the Edit menu. The Thicken dashboard is displayed, as shown in Fig 52.

Drag the handle to set the thickness of the quilt or enter the thickness value in the dimension box. You can even remove material from the quilt by choosing the Removes material from inside thickened quilt button from the dashboard.

Fig 51



Using the drop-down list present in the Options slide-up panel, you can give thickness to the quilt normal to the surface, allow Pro/ENGINEER to automatically scale the surface along axes, or scale and fit the original surface with respect to the coordinate system. If you select the Control Fit option, you need to select a coordinate system and specify the direction to scale. Fig 53 and 54 show the surfaces after adding thickness by controlling the thickness using the Normal to surface option and the Automatic fit option respectively.

Fig 52

Fig 53

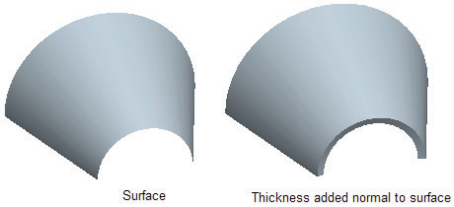
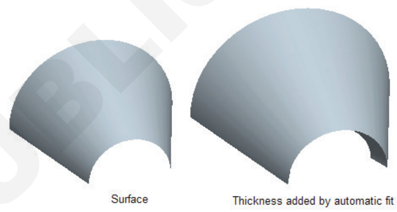


Fig 54



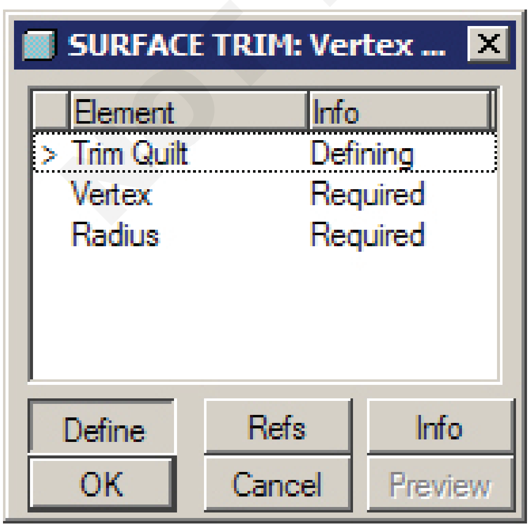
Converting a Surface to a Solid

You can convert a closed surface into a solid by choosing Edit > Solidify from the menu bar. This option is available only when a closed surface is selected. This option fills the hollow surface with material.

Creating Round at the Vertex of a Surface

The vertices of a surface or quilt can be rounded using the Vertex Round option. Choose Insert > Advanced > Vertex Round from the menu bar. The SURFACE TRIM: Vertex Round dialog box is displayed as shown in Fig 55.

Fig 55



You are prompted to select the datum quilt to intersect. Select the surface; now you are prompted to select the corner vertex(s) to be rounded. Select the first vertex and then press the CTRL key to select the other vertex as shown in Fig 56. After selecting the vertices, press the middle mouse button. The Message Input Window is displayed. Type the radius of round and press ENTER. Choose the OK button from the SURFACE TRIM: Vertex Round dialog box. The vertices are rounded as shown in Fig 57.

Fig 56

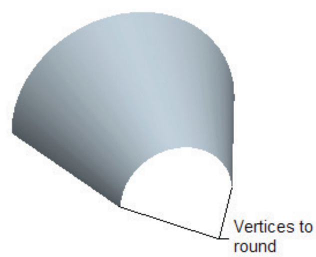
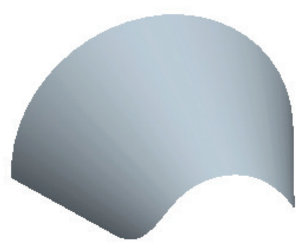


Fig 57



Creation of surface modelling

Objectives: At the end of this lesson you shall be to
- **explain the steps in creation of surface modelling**

In this lesson you will create the surface model shown in Fig 1. The orthographic views of the surface model are shown in Fig 2.

Fig 1

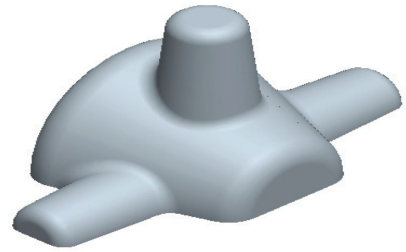
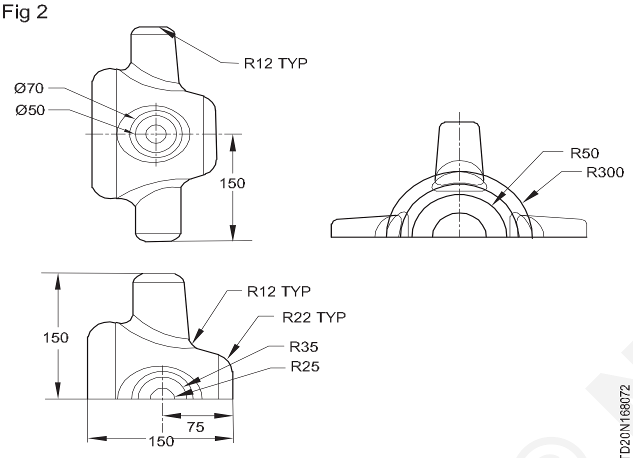


Fig 2



The following steps are required to complete this tutorial:

a Examine the model and determine the number of features in it, refer to Fig 2.

b Create the base feature, which is a blend surface, refer to Figs 3 through Fig 5.

Fig 3

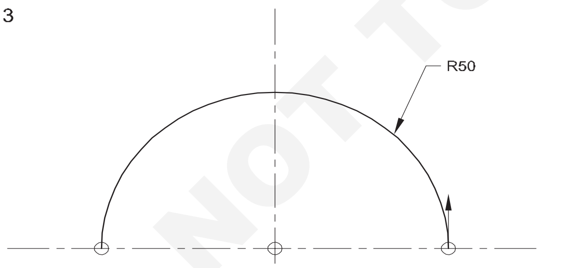


Fig 4

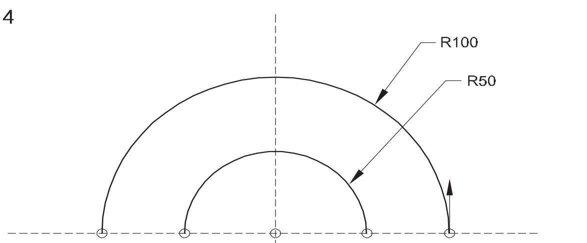
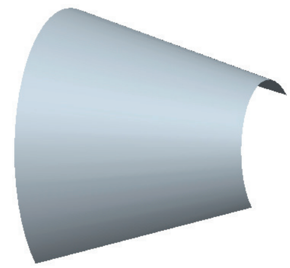


Fig 5



c Create the second feature, which is a blend feature, refer to Figs 6 & 7.

Fig 6

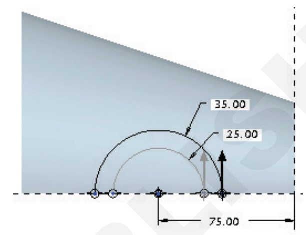
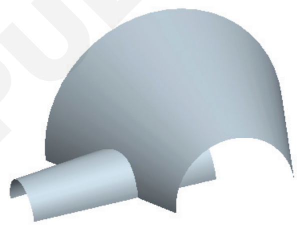
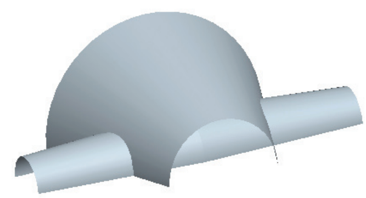


Fig 7



d The third feature is a mirror feature that will be created by mirroring the second feature about a plane passing from the center, refer to Fig 8.

Fig 8



e Create the fourth feature which is also a blend feature, refer to Figs 9 & 10.

Fig 9

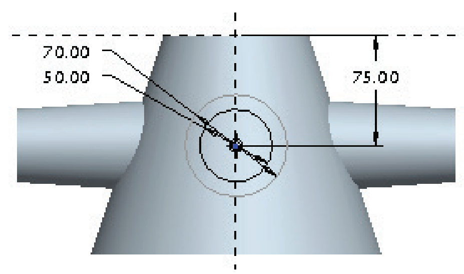
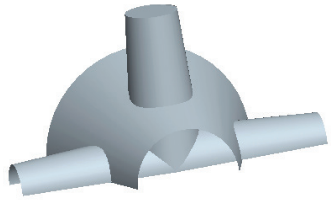


Fig 10



f Next, individually the surfaces will be selected to merge, refer to Figs 11 & 12.

Fig 11

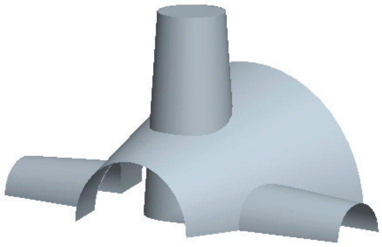
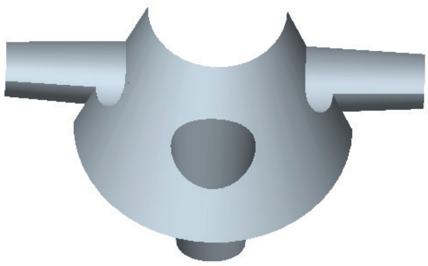


Fig 12



g Remaining features are the fill features that will create surfaces on the blend features, refer to Figs 13 through 15.

h Create rounds on the edges, refer to Fig 16.

Fig 13

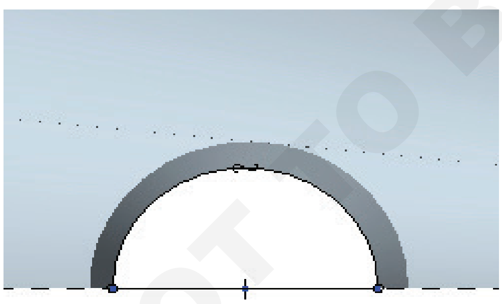


Fig 14

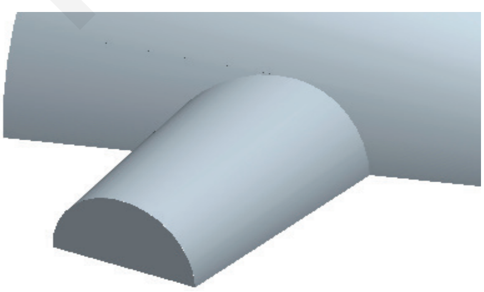


Fig 15

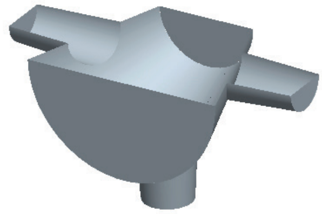
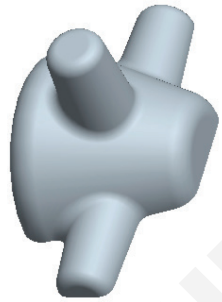


Fig 16



After understanding the procedure for creating the model, you are now ready to create it.

When the Pro/ENGINEER session is started, the first task is to set the working directory.

Since this is the first tutorial of this chapter, you need to create a folder named c13 if it does not exist and set it as the working directory.

Starting a New Object File

1 Start a new part file and name it as c13tut1.

The three default datum planes are displayed in the graphics window. The Model Tree is

also displayed in the graphics window. Close the Model Tree by clicking on the sash present on the right edge of the Model Tree.

Creating the Base Feature

You will use the menu bar present on the top of the screen to invoke the Blend option. The Blend option will be used to create the base feature.

1 Choose Insert > Blend > Surface from the menu bar. The BLEND OPTS menu is displayed.

2 Choose Parallel > Regular Sec > Sketch Sec > Done from the BLEND OPTS menu. The SURFACE dialog box and the ATTRIBUTES menu is displayed.

3 Choose Straight > Open Ends > Done from the ATTRIBUTES menu. You are prompted to select the sketch plane.

4 Select the RIGHT datum plane. The DIRECTION menu is displayed.

5 Choose Okay from the DIRECTION menu. The SKET VIEW menu is displayed.

6 Select the Top option and then choose the TOP datum plane. The References dialog box is displayed and you enter the Sketcher environment.

7 Close the References dialog box and draw the arc and dimension it as shown in Fig 3.

8 After drawing the first arc, press and hold down the right mouse button and choose the Toggle Section option from the shortcut menu.

9 Draw the second arc and dimension it as shown in Fig 4.

10 After drawing the sketch, choose the Continue with the current section button to exit the Sketcher environment. The DEPTH menu is displayed.

11 Choose Blind > Done from the DEPTH menu. The Message Input Window is displayed.

12 Enter a value of 150 and press ENTER.

13 Choose OK from the SURFACE dialog box. The base feature is created as shown in Fig 5.

Creating the Second Feature

To create the second blend feature you need to create a datum plane that is at a distance of 150 from the FRONT datum plane that is passing through the center of the base feature.

1 Choose Insert > Blend > Surface from the menu bar.

2 Choose the Parallel > Regular Sec > Sketch Sec > Done from the BLEND OPTS menu.

3 Choose Straight > Open Ends > Done from the ATTRIBUTES menu. You are prompted to select the sketch plane.

4 Choose the Make Datum option to display the DATUM PLANE menu. Select the Offset option and then select the FRONT datum plane.

5 Press the middle mouse button to confirm the selection. Now select the Enter Value option from the OFFSET menu to display the Message Input Window and enter 150 in it.

6 Select the Done option from the DATUM PLANE menu and then select Okay from DIRECTION menu.

7 Set the orientation of the sketch plane by selecting the TOP datum plane to be at the top while sketching.

8 After you enter the Sketcher environment, close the References dialog box.

9 Sketch the first arc of radius 25, and then after toggling the sketch draw the second arc as shown in Figure 6.

10 Exit the Sketcher environment; the DEPTH menu is displayed.

11 Choose Thru Until > Done from the DEPTH menu.

12 Select the FRONT datum plane. Choose OK from the SURFACE dialog box. The blend surface is extruded up to the selected datum plane as shown in Fig 7.

Creating the Mirror Copy of the Second Feature

The third blend feature is the same as the second blend feature. Therefore, a mirror copy of the second feature will be used to create the third feature.

1 Select the second feature and then choose the Mirror Tool button. The Mirror dashboard is displayed and you are prompted to select a plane to mirror about.

2 Select the FRONT datum plane and exit the Mirror dashboard by choosing the Build Feature button. The mirror copy of the second feature is created as shown in Fig 8.

Creating the Fourth Blend Feature

The fourth blend feature will be created on the top of the base feature. To create the blend feature, you will need to create a datum plane that is at a distance of 150 from the bottom of the base feature.

1 Choose Insert > Blend > Surface from the menu bar.

2 Choose Parallel > Regular Sec > Sketch Sec > Done from the BLEND OPTS menu.

3 Choose Straight > Capped Ends > Done from the ATTRIBUTES menu. You are prompted to select the sketch plane.

4 Choose the Make Datum option to display the DATUM PLANE menu. Select the Offset option and create a datum plane at a distance of 150 from the TOP datum plane.

5 Set the orientation of the sketch plane by selecting the RIGHT datum plane to be at the top while sketching.

6 After you enter the Sketcher environment, close the References dialog box.

7 Sketch the first circle of diameter 50, dimension it. Toggle the sketch and then draw the second circle of diameter 70 as shown in Fig 9.

8 Exit the Sketcher environment; the DEPTH menu is displayed.

9 Choose Thru Until > Done from the DEPTH menu.

10 Select the TOP datum plane. Choose OK from the SURFACE dialog box. The blend surface is extruded up to the selected datum plane as shown in Fig 10.

Merging the Surfaces to Create a Quilt

To create round on the edges it is necessary to create a common edge where the two surfaces are joining. For this purpose, the surfaces are merged.

Note: It is easier to select the two surfaces for merging from the Model Tree. You should remember that to select more than one surface you need to press the CTRL key. When you select the surfaces from the Model Tree the boundary of the surface is highlighted in red color indicating that the surface is selected. When you are selecting a surface directly from the graphics window, you need to select the surface thrice. The third time when you select the surface, it turns pink in color.

You can also select the Quilt option from the Filter drop-down list to select the surfaces. The Filter drop-down list is available in the status bar at the bottom right corner of the main window.

1 Select the blend surface that is at the left and then select the blend surface at the middle. When the two surfaces are highlighted, choose the Merge Tool. The Merge dashboard is displayed and the two arrows shows the portion that will be retained after merging.

Note: The Merge Tool button is available only when the two surfaces are selected for merging.

2 Choose the Change side of first quilt to keep button from the dashboard. The direction of yellow arrow changes.

3 Choose the Change side of second quilt to keep button from the dashboard. The direction of yellow arrow changes. The portion of the surface that is now highlighted will be retained after merging.

4 Exit the dashboard by selecting the Build Feature button. The model after merging the two surfaces is shown in Fig 11.

Using the same procedure, merge the blend surface at the right with the blend surface at the middle. After that, merge the top blend surface with the middle blend surface. Fig 12 shows the surface model after merging all the surfaces and forming a quilt.

Creating the Fill Surfaces

Four surfaces will be created to cap the ends of the blend surfaces. First, the left blend surface will be capped using the Fill option.

1 Choose the Fill option from the Edit menu. The Fill dashboard is displayed.

2 From the References slide-up panel choose the Define button. The Sketch dialog box is displayed and you are prompted to select the sketch plane.

3 Choose the Datum Plane Tool button from the Datum toolbar. To choose the button you need to move the Sketch dialog box because the dialog box overlaps the tool button.

4 Select the two vertices of the left blend surface. To select the second vertex hold down the CTRL key. Then holding down the CTRL key select the FRONT datum plane.

5 Select FRONT from the DATUM PLANE dialog box. The drop-down list appears in the row where you clicked. From the drop-down list, select the Parallel option. Choose the OK button from the DATUM PLANE dialog box. The datum plane is created and a yellow arrow points in the direction of viewing the sketch.

6 Choose the Sketch button to close the Sketch dialog box. Close the References dialog box and enter the Sketcher environment.

7 Choose the Create an entity from an edge button and select the smaller semicircular edge of the blend surface. Complete the sketch as shown in Fig 13.

8 Exit the Sketcher environment and then exit the Fill dashboard by choosing the Build Feature button. The Fill surface is created as shown in Fig 14.

Similarly, create the fill surfaces to cap the ends of the middle surface blend feature. Mirror the fill surface to create the fill surface at the right blend surface. Fig 15 shows the surface model after capping all the ends of the blend surfaces.

Merging the Fill Surfaces

The fill surfaces that you have created should be merged with the other surfaces in order to create round on their edges.

1 Hold down the CTRL key to select the fill surface that is at the left and then the blend surface at the middle. When the two surfaces turn pink in color, choose the Merge Tool to display the Merge dashboard.

2 Exit the dashboard by selecting the Build Feature button.

Using the same procedure, merge the remaining fill surfaces individually with the blend surface at the middle. To check whether all the surfaces are merged, select the surface model thrice. If the whole surface model is highlighted in pink color then all the surfaces are merged and form a quilt.

Creating Rounds

When all the surfaces are merged then the edges are obtained at the intersection of two surfaces. These edges can be easily rounded. In the given surface model, note that there are rounds that are having two different radius values. Therefore, you need to create two sets to define two values of rounds.

1 Choose the Round Tool from the Engineering Features toolbar.

2 Select the edges that have a radius value of 12. Remember that to select more than one edge, you need to hold down the CTRL key.

3 After creating the rounds of radii 12, select the Sets tab to display the slide-up panel.

4 Click on New set, you have added a set that is named Set2.

5 Select the two edges that are having radii of 22. After creating the rounds of radii 22, exit the Round dashboard.

The surface model after creating the rounds is as shown in Fig 16.

6 Choose the Save the active object button from the File toolbar and save the model.

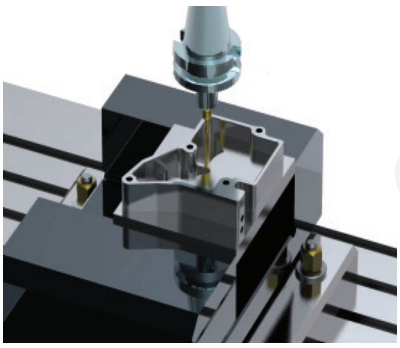
Manufacturing

Objectives: At the end of this lesson you shall be to
- **state what is manufacturing module in creo**
- **list of the sub modules in manufacturing module in creo**
- **brief the method of creating the NC code for a given part in creo.**

(Manufacturing process allows you set up and run NC machines, create assembly process sequences, create bills of material, and generate inspection programs for Coordinate Measuring Machines (CMMs).

The Manufacturing area of Help to learn about streamlining the NC programming process for production milling of prismatic parts and multisurface three-axis milling). Manufacturing Help shows you how to program and set up your NC machines, create process flows that cover the NC operations as well as other operations, and define CMM inspection programs that probe manufactured parts. (Fig 1)

Fig 1



Additive manufacturing is the process that creates three-dimensional objects from 3D digital models. Additive Manufacturing application lets you manage and print your models using a 3D printer. Use the Additive Manufacturing help to learn about creating tray assemblies, setting up printing options, and printing the models.

Manufacturing Modules consist of the following

- Creo NC
- Expert Machinist
- Manufacturing Process Planning
- Coordinate Measuring Machines (CMM)
- Manufacturing Verification
- Additive Manufacturing

Creo NC Overview

The Creo NC Help describes the logical sequence of tasks to progress from a design model to NC machine data. Creo NC creates the data necessary to drive an NC machine tool to machine a part. Refer to these topics for information on using the NC user interface, on configuring Creo Parametric for NC, and on performing manufacturing tasks.

Step on creo NC

- Using Creo NC
- Creating a Manufacturing Model
- Configuring Creo Parametric for Creo NC
- Using the Manufacturing Process
- Using the Process Manager
- Working with the Process Table
- Creating a Manufacturing Template
- Creating a Manufacturing Process
- Creating Operations
- Creating Workcells
- Defining Fixtures
- Specifying Coordinate Systems
- Creating NC Sequences
- Setting Up a Retract Surface
- Setting up the Machine Tool
- Setting Manufacturing Parameters
- Defining NC Sequences
- Working with Mill Geometries
- Defining a Milling NC Sequence
- Defining a Turning NC Sequence
- Defining a Hole Making NC Sequence
- Defining a Wire EDM NC Sequence
- Defining an Auxiliary NC Sequence
- Defining a User-Defined NC Sequence
- Defining a Mirror NC Sequence
- Using Cutter Location Data and Displaying Tool Path

- Using Cutter Location (CL) Data
- Displaying a Tool Path
- Performing Post-Processing for Creo NC

About Creo NC

Creo NC creates the data necessary to drive an NC machine tool to machine a Creo part. It does this by providing the tools to let the manufacturing engineer follow a logical sequence of steps to progress from a design model to ASCII CL data files that can be post-processed into NC machine data.

Licensing Requirements

Creo NC is a family of optional modules that can be ordered in any combination, to provide a custom fit of the available functionality to your company's needs.

The Creo Complete Machining license covers the complete Creo NC functionality as described in this Help System. Other modules provide subsets of this functionality.

To Create a Manufacturing Model

1 On the Creo Parametric Quick Access toolbar, click or click File > New. The New dialog box opens

2 Select the Manufacturing option under Type.

3 Select the NC Assembly option under Sub-type.

4 Type a name for the new manufacturing model in the Name text box, unless you want to accept the default.

5 If you do not wish to use the default template, clear the Use default template check box.

6 Click OK. The New File Options dialog box appears. Select the template on which you want to base the manufacturing drawing. If you do not wish to use a template, select Empty from the list of templates.

In the New File Options dialog box, select the Copy associated drawings check box if you want to create a new manufacturing drawing based on the drawing that is a part of the selected template.

For example, the template inlbs_mfg_nc.asm contains a corresponding drawing inlbs_mfg_nc.drw. To create a manufacturing model named mfgmodel.asm using this template, select the template and then select the Copy associated drawings check box. A drawing is created with the same name and a different extension, that is, mfgmodel.drw.

7 Click OK. The model tree of the manufacturing assembly is displayed in the Creo Parametric Navigator window. At this point, the assembly contains the components and features from the selected template. If you are not using a template, the assembly is empty. You have to define the manufacturing assembly configuration next.

About Manufacturing Process

Creo NC process consists of the following basic steps:

1 Set up the manufacturing database. It may contain such items as workcells (machine tools) available, tooling, fixture configurations, site parameters, or tool tables. This step is optional. If you do not want to set up all your database up front, you can go directly into the machining process and later define any of the items above when you actually need them.

2 Define an operation. An operation setup may contain the following elements:

- Operation name
- Work Centers (workcell or machine tool)
- Coordinate system for CL output
- Operation comments
- Operation parameters
- FROM and HOME points

You have to define a work center and a coordinate system before you can start creating NC sequences. Other setup elements are optional.

3. Create NC sequences for the specified operation. Each NC sequence is a series of tool motions with the addition of specific post-processor words that are not motion-related but required for the correct NC output.

The tool path is automatically generated by the system based on the NC sequence type (such as Volume Milling, Outside Turning), cut geometry, and manufacturing parameters. You can apply more "low level" control, if you like, by:

- Defining your own tool motions, that is, approach, exit, and connect motions. Tool motions include Automatic Cut motions.
- Inserting non-motion CL commands.

4 For each completed NC sequence, you can create a material removal feature, either by making the system automatically remove material (where applicable), or by manually constructing a regular Creo feature on the workpiece (such as Slot or Hole).

Modal Settings

Most of the machining setup elements are modal, that is, all subsequent NC sequences will use this setting until you explicitly change it. Among those are:

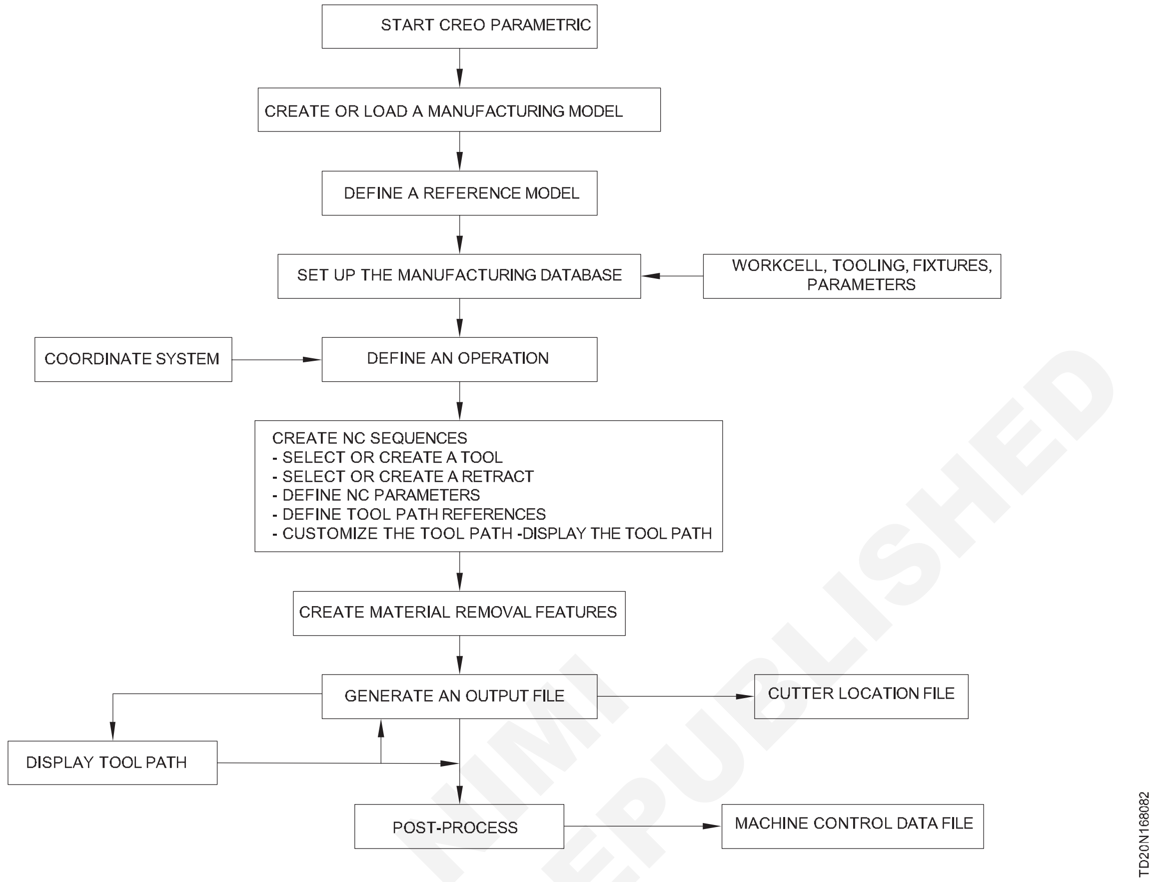
- Operation setup (including the workcell and Machine coordinate system)
- Fixture setup
- Tool (provided the tool type is compatible with the NC sequence type)
- Manufacturing parameters of an activated site
- NC Sequence coordinate system (for the first NC sequence, the Machine Coordinate system specified for the operation will be implicitly used as the NC sequence coordinate system as well, unless you explicitly specify another one)

- Retract surface.

Example: Manufacturing Process

The following flow chart gives a summary of the manufacturing process.

Fig 2



Mold Design and Casting

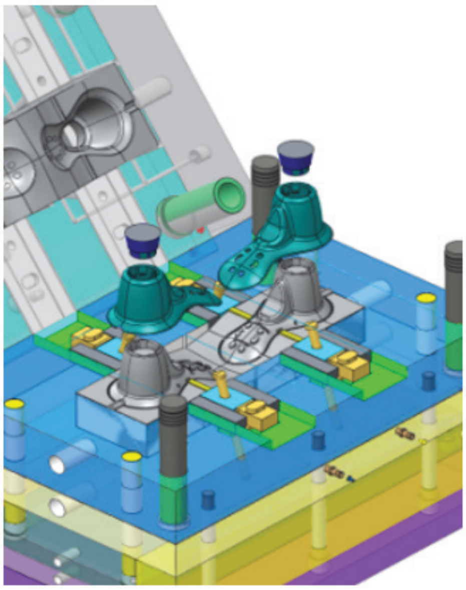
Objectives: At the end of this lesson you shall be to
- **brief the mold design and casting using creo parametric**

Introduction

Mold Design and Casting lets you simulate the mold design process, design die assemblies and components, and prepare castings for manufacturing.

Use the Mold Design and Casting area of Help to learn about creating, modifying, and analyzing mold components or assemblies. Mold Design and Casting Help shows you how to quickly update mold components to the changes in the design model or to design die assemblies and components and prepare castings for manufacturing. You will learn how to create and modify design parts, cavities, mold layouts, and drawings. (Fig 1)

Fig 1



Mold Design and Casting Modules

- Mold Design and Casting
- Creo Mold Analysis (CMA)

Mold Design and Casting

Mold Design and Casting is an optional module for Creo Parametric that provides the tools to simulate the mold and cast design process. This module lets you create, modify, and analyze the mold and cast components and assemblies, and quickly update them to the changes in the design model.

Mold Design and Casting, together with Creo Parametric, provide tools to do the following:

Design Part Creation and Modification

- Create models
- Import and repair geometry if necessary

The import functionality can be used with the following: See Data Doctor Option and Interface

- Analyze if a design part is moldable, using Draft Check and Thickness Check capabilities
- Automatically create parting lines and detect undercuts using Silhouette Curve functionality
- Fix problem areas by creating draft, rounds, and other features as needed

Cavity Creation

- Assemble and orient design model dynamically while checking draft and projected area
- Apply a shrinkage that corresponds to design part material, geometry, and molding conditions
- Automatically create the workpiece stock from which core, cavity, and inserts will be split
- Create parting geometry, including sliders, inserts, automatic parting lines, and automatic parting surfaces
- Automatically split the workpiece to create cores, cavity, and inserts as solid models
- Create and assemble sand cores for cast design

Mold Layout Creation

- Create top level mold assembly
- Placement and patterning of mold cavities to allow multi-cavity molding
- Online selection and automatic assembly of standard mold bases
- Modification of mold base plates to allow for assembly of mold cavity
- Online selection and automatic assembly of ejector pins and other Mold Catalog items
- Automated creation of runners
- Automated creation of waterlines, including 3-D waterline interference checks
- Define and simulate mold opening and check for interference between mold components

Drawing Creation

- Create complete production drawings, including dimensions, tolerances, automatic bill of materials (BOMs) with or without balloon notes
- Use of drawing templates

About Mold and Cast Features

Mold and cast features exist at the assembly level. There are two classifications of features: regular features and user-defined features.

Regular features are special features added to a model to facilitate the molding or casting process. These features include silhouette curves, ejector pin holes, runners, waterlines, draft lines, offset areas, volumes, and trimming features.

User-defined features are created in part mode and used to create commonly used structures in the workpiece or die block. A user-defined feature is created once and used multiple times by modifying the dimensions each time it is copied into an assembly.

About PTC Creo Mold Analysis

Creo Mold Analysis (CMA) is an injection molding simulation application. You can use CMA to verify the part design for manufacturing to avoid the molding defects. Some of the most common molding issues are incomplete cavity filling, undesirable weld line location, and improper wall thickness.

You can use the unique auto-meshing function to prepare the model for analysis in very less time. Additionally, CMA provides product insights in terms of potential weld line, air trap locations, and the required clamping force.