MACHINIST

NSQF LEVEL - 4

2nd Year

TRADE THEORY

SECTOR : CAPITAL GOODS & MANUFACTURING

(As per revised syllabus July 2022 - 1200Hrs)

Directorate General of Training

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

NATIONAL INSTRUCTIONAL
MEDIA INSTITUTE, CHENNAI

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

Sector : **Capital Goods and Manufacturing**

Duration : **2 Year**

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

Developed & Printed by

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

First Edition : November 2023 Copies : 1000

Rs.360/-

FOREWORD

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

The National Instructional Media Institute (NIMI), Chennai, has now come up with instructional material to suit the revised curriculum for **Machinist 2nd Year Trade Theory** in **CG & M Sector** under **Yearly Pattern.** The NSQF Level - 4 (Revised 2022) Trade Practical will help the trainees to get an international equivalency standard where their skill proficiency and competency will be duly recognized across the globe and this will also increase the scope of recognition of prior learning. NSQF Level - 4 (Revised 2022) trainees will also get the opportunities to promote life long learning and skill development. I have no doubt that with NSQF Level - 4 (Revised 2022) the trainers and trainees of ITIs, and all stakeholders will derive maximum benefits from these Instructional Media Packages IMPs and that NIMI's effort will go a long way in improving the quality of Vocational training in the country.

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

Jai Hind

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

November 2023
New Delhi - 110 001

PREFACE

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

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

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

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

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

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

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

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

Chennai - 600 032 **EXECUTIVE DIRECTOR**

ACKNOWLEDGEMENT

National Instructional Media Institute (NIMI) sincerely acknowledges with thanks for the co-operation and contribution extended by the following Media Developers and their sponsoring organisations to bring out this Instructional Material **(Trade Theory)** for the trade of **Machinist 2nd Year NSQF Level - 4 (Revised 2022)** under **Capital Goods & Manufacturing** Sector for ITIs.

MEDIA DEVELOPMENT COMMITTEE MEMBERS

Shri. A. Vijayaraghavan - Assistant Director of Training (Retd.),
ATI, Chennai - 32.

Shri. Suriya Kumar - MTAB Technology Center (P) Ltd.,
Tamilnadu.

Shri. J.D. Dinesh Kumar - MTAB Technology Center (P) Ltd.,
Tamilnadu.

Shri. H.A. Manukumar - Training Officer,
NSTI, Chennai.

Shri. M. Sampath - Training Officer (Retd.),
CTI, Chennai - 32.

NIMI CO-ORDINATORS

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

Shri. V. Gopalakrishnan - Manager,
NIMI, Chennai - 32.

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

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

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

INTRODUCTION

TRADE PRACTICAL

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

The manual is divided into Six modules

Module 1	**Tool and Cutter Grinding**
Module 2	**Milling**
Module 3	**CNC Turning**
Module 4	**CNC Milling (VMC- Vertical Milling Center)**
Module 5	**Repair & Overhauling**
Module 6	**Advanced Milling**

The skill training in the shop floor is planned through a series of practical exercises centered around some practical project. However, there are few instances where the individual exercise does not form a part of project.

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

TRADE THEORY

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

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

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

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

CONTENTS

LEARNING / ASSESSABLE OUTCOME

On completion of this book you shall be able to

S.No	Learning Outcome	Ref. Ex.No.
1	**Re-sharpen different single & multipoint cutting tool. [Different single point tools, slab milling cutter, side & face milling cutter, end mill cutter and shell end mill cutter.] CSC/N0109**	**2.1.111-2.1.115**
2	**Set different machining parameters and cutters to prepare job by different milling machine operations. [Different machining parameters - feed, speed, depth of cut, different machining operation - facing, drilling, tapping, reaming, counter boring, counter sinking, spot facing, and boring slot cutting.] CSC/N9407**	**2.2.116-2.2.121**
3	**Set the different machining parameters and cutters to prepare components by performing different milling operation and indexing. [Different machining parameters - feed, speed and depth of cut. Different components - Rack, Spur Gear, External Spline, Steel Rule, Clutch, Helical Gear] CSC/N9407**	**2.2.122-2.2.130**
4	**Set (both job and tool) CNC turning centre and produce components as per drawing by preparing part programme. CSC/NO115**	**2.3.131-2.3.166**
5	**Set CNC VMC (vertical machining center) and produce components as per drawing by preparing part program.CSC/N9408**	**2.4.167-2.4.214**
6	**Plan and perform simple repair, overhauling of different machines and check for functionality. [Different Machines - Drilling Machine, milling machine and Lathe] CSC/N9403**	**2.5.215-2.5.219**
7	**Set the different machining parameters and cutters to prepare components by performing different milling operation and indexing. [Different machining parameters - feed, speed and depth of cut. Different components - end mill, bevel gear, cam, worm & worm wheel] CSC/N9407**	**2.6.220-2.6.224**

SYLLABUS

Duration	Reference Learning Outcome	Professional Skills (Trade Practical) with Indicative hours	Professional Knowledge (Trade Theory)
Professional Skill 60 Hrs.; Professional Knowledge 15 Hrs	Re-sharpen different single & multipoint cutting tool. [Different single point tools, slab milling cutter, side & face milling cutter, end mill cutter and shell end mill cutter.] CSC/N0109	111.Demonstrate and practice of grinding of different single point tools. (18 hrs.)	Tool & cutter grinder- Introduction, parts, construction, use and specification, different types of tool rest & their application. (05 hrs.)
		112.Demonstrate and practice of grinding of slab milling cutter. (10 hrs.) 113.Re-sharpening side and face milling cutter. (12 hrs.)	Various methods of cutter grinding. (05 hrs.)
		114.Demonstrate and practice of grinding of end mill cutter. (10 hrs.) 115.Re-sharpening of shell end mill cutter. (10 hrs.)	Various cutter grinding attachments and their uses. (05 hrs.)

Duration	Reference Learning Outcome	Professional Skills (Trade Practical) with Indicative hours	Professional Knowledge (Trade Theory)
Professional Skill 60 Hrs; Professional Knowledge 15 Hrs.	Set different machining parameters and cutters to prepare job by different milling machine operations. [Different machining parameters - feed, speed, depth of cut, different machining operation - facing, drilling, tapping, reaming, counter boring, counter sinking, spot facing, and boring slot cutting.] CSC/N9407	116.Practice of facing on milling Machine. (08 hrs.) 117.Drill on P.C.D on milling Machine with accuracy +/- 0.02 mm. (10 hrs.)	Geometrical tolerances, definition, symbol and their application. Depth Micrometer - Parts, reading, uses and safety. (05 hrs.)
		118.Perform Tapping and Reaming operation using milling Machine with an accuracy +/- 0.02 mm.(08hrs.) 119.Perform spot facing operation using milling machine with accuracy +/- 0.02 mm. (10 hrs.)	Different types of micrometers and their uses. Inside Micrometer - its parts, reading and uses. Bore Dial Gauge - its parts, reading (both in Metric and English system)and uses. Telescopic gauge. (05hrs.)
		120.Make slot on face of the job using milling Machine with an accuracy +/-0.02 mm. (10 hrs.) 121.Make Internal Grooving using milling Machine with an accuracy 0.02 mm. (14 hrs.)	Gauges - different types and their uses, difference between Gauges and Measuring Instruments. Gear introduction, use and type. Elements of a spur gear. Gear tooth of each forms types, merits and demerits of each. (5 hrs.)
Professional Skill 87Hrs.; Professional Knowledge 24Hrs.	Set the different machining parameters and cutters to prepare components by performing different milling operation and indexing. [Different machining parameters - feed, speed and depth of cut. Different components - Rack, Spur Gear, External Spline, Steel Rule, Clutch, Helical Gear] CSC/N9407	122.Make Straight Teeth Rack using Milling Machine with an accuracy 0.05 mm. (08 hrs.) 123.Make Helical Teeth Rack using Milling Machine with an accuracy 0.05 mm one straight rack. (08 hrs.) 124.Measurement of teeth by Vernier Gear Tooth Caliper.(03 hrs.)	Rack - types, uses and calculations. Selection of gear cutter type and form & various methods of checking gear and its parts. Vernier gear tooth caliper - its construction and application in checking gear tooth. (07hrs.)
		125.Make spur gear using Simple indexing with an accuracy 0.05 mm. (08 hrs.) 126.Make spur gear using differential indexing with an accuracy 0.05 mm. (08 hrs.)	Spur gear calculations, curves and their uses. Use of radius gauges and template. (04hrs.)
		127.Perform Boring operation on Vertical Milling Machine with an accuracy 0.05 mm. (16 hrs.)	Vertical Milling Machine- its parts. Method of boring in Vertical milling. Difference between Horizontal and Vertical Milling Machine. (04hrs.)
		128.Make helical gear on milling machine with an accuracy 0.05 mm. (18 hrs.)	Helix and Spiral introduction, types and elements. Difference between helix & spiral. Difference between R.H. and L.H. helix. Helical gear- elements, application. Calculations for cutting helical gear. (05hrs.)
		129.Make straight flute milling on Milling Machine with an accuracy 0.05 mm. (10 hrs.)	Reamer - types, elements and uses. Calculations for cutting Reamer.

Duration	Reference Learning Outcome	Professional Skills (Trade Practical) with Indicative hours	Professional Knowledge (Trade Theory)
		130.Make helical flute on Milling Machine with an accuracy 0.02 mm. (08 hrs.)	Twist drill-nomenclature, cutter selection. Calculations for cutting twist drill. (04hrs.)
Professional Skill 200Hrs.; Professional Knowledge 40Hrs.	Set (both job and tool) CNC turning centre and produce components as per drawing by preparing part programme. CSC/NO115	131.Know rules of personal and CNC machine safety, safe handling of tools, safety switches and material handling equipment using CNC didactic/ simulation software and equipment. (03 hrs.) 132.Identify CNC lathe machine elements and their functions, on the machine. (07 hrs.) 133.Understand the working of parts of CNC lathe, explained using CNC didactic/ simulation software. (09 hrs.) 134.Identify machine over travel limits and emergency stop, on the machine. (01 hr) 135.Decide tool path for turning, facing, grooving, threading, drilling. (04hrs.) 136.Identification of safety switches and interlocking of DIH modes. (01 hr)	Personal safety, safe material handling, and safe machine operation on CNC turning centers. CNC technology basics, Comparison between CNC and conventional lathes. Concepts of positioning accuracy, repeatability. CNC lathe machine elements and their functions - bed, chuck, tailstock, turret, ball screws, guide ways, LM guides, coolant system, hydraulic system, chip conveyor, steady rest, console, spindle motor and drive, axes motors, tail stock, encoders, control switches. Feedback, CNC interpolation, open and close loop control systems. Machining operations and the tool paths in them - stock removal in turning and facing, grooving, face grooving, threading, drilling. (05hrs.)
		137.Identify common tool holder and insert shapes by ISO nomenclature. (05hrs.) 138.Select cutting tool and insert for each operation. (03hrs.) 139.Fix inserts and tools in tool holders. (02hrs.) 140.Decide cutting tool material for various applications. (03hrs.) 141.Select cutting parameters from tool manufacturer's catalogue. (02hrs.) 142.Write CNC programs for simple tool motions and parts using linear and circular interpolation, check on program verification/ simulation software. (10hrs.) 143.Write CNC part programs using canned cycles for stock removal, grooving, threading operations, with drilling and finish turning. Use TNRC commands for finish turning. Check simulation on program verification/ simulation software. (18hrs.)	Concept of Co-ordinate geometry, concept of machine coordinate axis, axes convention on CNC lathes, work zero, machine zero. Converting part diameters and lengths into co-ordinate system points. Absolute and incremental programming. Programming - sequence, formats, different codes and words. ISO G codes and M codes for CNC turning. Describe CNC interpolation, open and close loop control systems. Co-ordinate systems and Points. Program execution in different modes like MDI, single block and auto. Canned cycles for stock removal (turning/facing), grooving, threading, for external and internal operations. Tool nose radius compensation (TNRC) and why it is necessary. Find the geometry page in CNC machine.

Duration	Reference Learning Outcome	Professional Skills (Trade Practical) with Indicative hours	Professional Knowledge (Trade Theory)
		144.Avoiding collisions caused by program errors. Knowing causes and effects of collisions due to program errors, by making deliberate program errors and simulation on program verification/ simulation software. (06 hrs.)	Cutting tool materials, application of various materials. Cutting tool geometry for internal and external turning, grooving, threading, face grooving, drilling. Insert holding methods for each. Insert cutting edge geometry. ISO nomenclature for turning tool holders, boring tool holders, Indexable inserts. Cutting parameters- cutting speed, feed rate, depth of cut, constant surface speed, limiting spindle speed. Tool wear, tool life, relative effect of each cutting parameter on tool life. Selection of cutting parameters from a tool manufacturer's catalogue for various operations. Writing part programs as per drawing & checking using CNC program verification/ simulation software. Process planning, work holding, tool and cutting parameters selection according to the part geometry and dimensions. Collisions due to program errors, effects of collisions. Costs associated with collisions - tool breakage, machine damage, injuries. (10hrs.)
		145.Conduct a preliminary check of the readiness of the CNC lathe - cleanliness of machine, functioning of lubrication, coolant level, correct working of sub-systems, on the machine. (05 hrs.) 146.Starting the machine, do homing on CNC simulator. (02 hrs.) 147.Entering the CNC program in EDIT mode for an exercise on Simple turning & Facing (step turning) without using canned cycles, on CNC simulator. (15 hrs.) 148.Mounting jaws to suit the part holding area on CNC machine (03hrs.) 149.Mounting tools on the turret according to part and process requirement, on CNC simulator &on CNC machine. (08hrs.)	Program execution in different modes like MDI, single block and auto. Process planning & sequencing, tool layout& selection and cutting parameters selection. Work and tool offsets. Inputs value to the offset/ geometry page into machine. Turning in multiple setups, hard and soft jaws, soft jaw boring, use of tailstock and steady rest. Length to diameter (L/D) ratio and deciding work holding based on it. Machine operation modes - Jog, MDI, MPG, Edit, Memory. Entering and editing programs on machine console, entering offsets data in offsets page.

Duration	Reference Learning Outcome	Professional Skills (Trade Practical) with Indicative hours	Professional Knowledge (Trade Theory)
		150.Perform Work and tool setting: Job zero/work coordinate system and tool setup and live tool setup. (08hrs.) 151.Determining work and tool offsets using JOG, MDI, MPG modes, on CNC simulator. (08hrs.) 152.Entering the tool offsets, tool nose radii and orientation for TNRC in offsets page, on CNC simulator. (05hrs.)	Use of Emergency stop, Reset, Feed rate override, spindle speed override, edits lock on/off buttons and keys. (10hrs.)
		153.Program checking in dry run, single block modes, on CNC simulator & CNC machine. (01hr) 154.Absolute and incremental programming assignments and simulation. (04 hrs.) 155.Checking finish size by over sizing through tool offsets, on CNC simulator. (02hrs.) 156.Prepare part program and cut the part in auto mode in CNC machine for the exercise on Simple turning & Facing (step turning) (08 hrs.) 157.Recovering from axes over travel, on CNC simulator (01 hr) 158.Part program writing, setup, checking and Automatic Mode Execution for exercise on Turning with Radius/ chamfer with TNRC on CNC machine (10hrs.) 159.Part program writing, setup, checking and Automatic Mode Execution for exercise on Turning with TNRC, grooving and threading, on CNC simulator & on CNC machine (12hrs.) 160.Checking finish size by over sizing through tool offsets, on the machine. (02 hrs.) 161.Machining parts on CNC lathe with combination step, taper, radius turning, grooving &threading, with external and internal operations, first and second operation, on the machine. (10 hrs.) 162.Machining long part on CNC lathe held in chuck and tailstock (between centers). (04 hrs.) 163.Starting from interruption due to power shutdown, tool breakage. (01hr)	First part checking: Program checking in single block and dry run modes - necessity and method. Tool offsets adjustment on first part for close tolerance dimensions, by over sizing (for outside dimensions) or under sizing (for inside dimensions) the dimension to prevent part rejection. Wear offset setting - necessity, relationship with tool wear, entering in offsets page. Process and tool selection related to grooving, drilling, boring and threading. Axes over travel, recovering from over travel. Collisions due to improper machine setup and operation - causes and effects. Recovering from collisions. Find out alarm codes and meaning of those codes. (15hrs.)

Duration	Reference Learning Outcome	Professional Skills (Trade Practical) with Indicative hours	Professional Knowledge (Trade Theory)
		164.Changing wear offsets to take into account tool wear. (02hrs.) 165.Part program preparation, Simulation & Automatic Mode Execution of CNC Machine for the exercise on Blue print programming contours with TNRC. (07 hrs.) 166.Carryout Drilling/Boring cycles in CNC Turning. (08 hrs.) (First 60% of the practice is on CNC machine simulator, followed by 40% on machine.)	
Professional Skill 313Hrs. Professional Knowledge 98Hrs.	Set CNC VMC (vertical machining center) and produce components as per drawing by preparing part program. CSC/ N9408	167. Identify CNC vertical machining center machine elements and their functions,on the machine. (10hrs.) 168.Understand working of parts of CNC VMC, explained using CNC didactic/ simulation software (20 hrs.) 169.Identify machine over travel limits and emergency stop, on the machine. (05hrs.) 170.Decide tool path for Face milling, Side milling, Pocket milling, Drilling, Counter sinking, tapping, Reaming, Rough boring, Finish boring, Spot facing. (03hrs.)	Safety aspects related to CNC VMC.CNC technology basics, Comparison between CNC VMC and conventional milling machines. Concepts of positioning accuracy, repeatability. CNC VMC machine elements and their functions - bed, chuck, Auto tool changer (ATC), ball screws, guide ways, LM guides, coolant system, hydraulic system, chip conveyor, rotary table, pallet changer, console, spindle motor and drive, axes motors, encoders, control switches. Feedback, CNC interpolation, open and close loop control systems. Machining operations and the tool paths in them - Face milling, Side milling, Pocket milling, Drilling, Countersinking, Rigid tapping, floating tapping Reaming, Rough boring, Finish boring, Spot facing. (15 hrs)
		171.Identify common tools, tool holders and inserts. (05 hrs.) 172.Select cutting tool, insert and holder for each operation. (05 hrs.) 173.Fix inserts and tools in tool holders. (03 hrs) 174.Decide cutting tool material for various applications. (04 hrs.) 175.Select cutting parameters from tool manufacturer's catalog. (02 hrs) 176.Write CNC programs for simple parts using linear and circular interpolation, absolute and incremental modes, check on program verification software. (15 hrs.)	Concept of Co-ordinate geometry& polar coordinate points, concept of machine axis, axes convention on CNC lathes, work zero, machine zero. Converting part dimensions into coordinate system points. Absolute and incremental programming. Programming - sequence, formats, different codes and words. ISO G and M codes for CNC milling. Canned cycles for drilling, peck drilling, reaming, tapping, finish boring. Subprograms.

Duration	Reference Learning Outcome	Professional Skills (Trade Practical) with Indicative hours	Professional Knowledge (Trade Theory)
		177.Write CNC part programs for parts with face milling, pocket milling with subprograms. Check on program verification software. (11hrs.) 178.Write CNC part programs for pocket milling, drilling with canned cycle, countersinking with canned cycle, tapping with canned cycle. Check on program verification software. (10hrs.) 179.Avoiding collisions caused by program errors. Knowing causes and effects of collisions due to program errors, by making deliberate program errors and simulation on program verification software. (06 hrs.)	Cutter radius compensation (CRC)and why it is necessary. Cutting tool materials, application of various materials. Cutting tool geometry for face mill, end mill, drill, countersink, tap, finish bore, reamer. Insert holding methods face mill, insert type end mill and insert type drill. Insert cutting edge geometry. Cutting parameters- cutting speed, feed rate, depth of cut. Tool wear, tool life, relative effect of each cutting parameter on tool life. Selection of cutting parameters from a tool manufacturer's catalog for various operations. Writing part programs as per drawing & check using CNC program verification software. Process planning, work holding, tool and cutting parameters selection according to the part geometry and dimensions. Collisions due to program errors, effects of collisions. Costs associated with collisions - tool breakage, machine damage, injuries. (20hrs.)
		180.Conduct a preliminary check of the readiness of the CNC VMC - cleanliness of machine, functioning of lubrication, coolant level, correct working of sub-systems. On the machine. (03 hrs.) 181.Starting the machine, do homing on CNC simulator. (03 hrs.) 182.Entering the CNC program in EDIT mode for an exercise on face milling and drilling without using canned cycles, on CNC simulator. (12 hrs.) 183.Mounting tools on the ATC according to part and process requirement, on CNC simulator & CNC machine. (08hrs.) 184.Determining work and tool offsets using JOG, MDI, MPG modes, on CNC simulator& CNC machine. (07hrs.) 185. Tool change in CNC milling and JOG, MDI, MPG mode operation. (06 hrs.)	Program execution in different modes like manual, single block and auto. Process planning & sequencing, tool layout & selection and cutting parameters selection. Work offset, tool length offset, tool radius offset. Work holding with temporary holding and fixtures. Truing of part and fixture. Machine operation modes - Jog, MDI, MPG, Edit, Memory. Entering and editing programs on machine console, entering offsets data in offsets page. Use of Emergency stop, Reset, Feed rate override, spindle speed override, edit lock on/off buttons and keys. (15hrs.)

Duration	Reference Learning Outcome	Professional Skills (Trade Practical) with Indicative hours	Professional Knowledge (Trade Theory)
		186.Program checking in dry run, single block modes, on CNC simulator. (04 hrs.) 187.Checking finish size by over or under sizing through tool offsets, on CNC simulator. (05 hrs.) 188.Prepare part programme, enter, edit and simulate. (04 hrs.) 189.Carryout tool path simulation. (04 hrs.) 190.Recovering from axes over travel, on virtual machine simulator (03 hrs.) 191.Part program writing, setup, checking and Automatic Mode Execution for exercise on side milling with CRC, on CNC simulator & CNC machine. (15 hrs.) 192.Part program writing, setup, checking and Automatic Mode Execution for exercise on face milling, drilling, countersinking, tapping using canned cycle, on CNC simulator & CNC machine (20 hrs.) 193.Automatic mode execution of CNC Machine Exercises with Block Search and restart. (12 hrs.) 194.Mounting clamps, locators, supports, truing part and fixture. (8 hrs.)	First part checking: Program checking in single block and dry run modes -necessity and method. Tool offsets adjustment on first part for close tolerance dimensions, by oversizing (for outside dimensions) or under sizing (for inside dimensions) the dimension to prevent part rejection. Axes over travel, recovering from over travel. Collisions due to improper machine setup and operation - causes and effects. Recovering from collisions. State the importance of Helical inter-polar and thread milling, advantage and limitation in CNC machine. (20hrs.)
		195. Machining part on CNC VMC with face milling, drilling. (05 hrs.) 196.Machining parts on CNC VMC with combination face milling, side milling with CRC, drilling, countersinking, tapping. Use canned cycles and subprograms wherever possible. (05 hrs.) 197.Machining of part with closely controlled slot dimension usingCRC. (05hrs.) 198.Machining of part with pockets. (02 hrs.) 199.End milling with polar co-ordinates. (04 hrs.) 200.Part programs & Simulation Automatic Mode Execution of CNC Machine for the exercise on End milling with polar co- ordinates and practical on Simple drilling-G 81. (06 hrs.) 201.Determining and entering wear offsets. (03 hrs.)	Tool wear and necessity for wear offsets change, entering wear offsets in offsets page. Effects of sudden machine stoppage due to power shutdown or use of emergency stop. Restarting machine from sudden stoppage. Means of program transfer through electronic media. Productivity concepts, cycle time, machine down time, causes of down time - breaks, machine breakdown, inspection, part loading and unloading, chip cleaning. Effect of down time on profitability, reducing down time. Machine hour rate, components of machine hour rate - principal repayment, interest, overheads (power, tooling, space, salaries, indirect expenses).

Duration	Reference Learning Outcome	Professional Skills (Trade Practical) with Indicative hours	Professional Knowledge (Trade Theory)
		202.Restarting machine from power shutdown or sudden stoppage. (01hr) 203.Program transfer to machine through electronic media - USB and flash drive. (01 hr) 204.Merging the work zero with program zero point, geometry and wear offset correction. (02 hrs.) 205.Practical on Chamfer and counter-sink drilling. (02 hrs.) 206.Carryout Deep hole drilling G 83. (03 hrs.) 207.Perform Threading and tapping G 84. (06 hrs.) 208.Carryout Boring cycles G 85 - G 89. (08 hrs.) 209.Preparations of part programs for thread cutting/thread milling for CNC machining centres.(06 hrs.) 210.Drilling milling patterns, Thread milling etc. (03 hrs.) 211. Circular and rectangular pockets machining. (03 hrs.) 212.Calculation of machine hour rates for typical CNC lathe and VMC.(05 hrs.) 213.Estimation of cycle time for parts with face milling, side milling, drilling, tapping operations. (05hrs.) (First 60% of the practice is on CNC machine simulator, followed by 40% on machine.)	Calculation of machining cost, cost of down time. (20hrs.)
		214.Prepare different types of documentation as per industrial need by different methods of recording information. (25 hrs.)	Machine productivity concepts - cycle time, down time, cycle time estimation. Costing - machine hour rate, machining cost, tool cost, cost of down time. Importance of Technical English terms used in industry. Technical forms, process sheet, activity log, job card, in industry-standard formats.(08hrs.)
Professional Skill 45 Hrs.; Professional Knowledge 12Hrs.	Plan and perform simple repair, overhauling of different machines and check for functionality. [Different Machines - Drilling Machine, milling machine and Lathe] CSC/N9403	215.Perform Periodic Lubrication system on Machines. (10 hrs.) 216.Perform simple repair work.(10hrs.)	Lubricating system-types and importance. (05hrs.)
		217.Perform the routine maintenance with check list. (05hrs.)	Maintenance: Definition, types and its necessity.

Duration	Reference Learning Outcome	Professional Skills (Trade Practical) with Indicative hours	Professional Knowledge (Trade Theory)
		218.Inspection of Machine tools such as alignment, leveling etc. (10 hrs.) 219.Accuracy testing of machine tools such as geometrical parameters.(10 hrs.)	System of symbol and colour coding. Possible causes for failure and remedies. (07hrs.)
Professional Skill 75Hrs; Professional Knowledge 28Hrs.	Set the different machining parameters and cutters to prepare components by performing different milling operation and indexing. [Different machining parameters - feed, speed and depth of cut. Different components - end mill, bevel gear, cam, worm & worm wheel] CSC/ N9407	220.Cutting teeth on helical slab/ cylindrical cutter and end mill cutter with an accuracy of +/- 0.05 mm. (15hrs.)	Calculations for cutting helical slab/ cylindrical cutter. Calculations for cutting End Mill cutter. (06hrs.)
		221. Cutting bevel gears on a milling machine with an accuracy of +/-0.05 mm. (15 hrs.)	Bevel gear-elements, types, application, calculation for cutting bevel gear. (06 hrs.)
		222. Cutting a plate cam with angular setting in milling machine with an accuracy of +/-0.05 mm. (15 hrs.)	Cam-types, elements & application, Plate cam- manufacturing & calculations. Drum cam- its calculation, advantages, types of follower & its purposes. (06hrs.)
		223. Cutting worm wheel on a milling machine with an accuracy of +/- 0.05 mm. (15 hrs.)	Worm wheel-application, elements & calculation, Worm-calculation.(05hrs.)
		224. Cutting worm thread on a milling machine with an accuracy of +/- 0.05 mm. (15 hrs.)	Types of Keys and their uses. Variation - types and causes. Testing of Gear and error. (05hrs.)

Tool and cutter grinder

Objectives: At the end of this lesson you shall be able to
- **describe the tool and cutter grinder**
- **explain the parts and construction of tool and cutter grinder**
- **specify the 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 continuous 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 (360º) it can be moved up and down by means of hand or power.

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 (180º).

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.

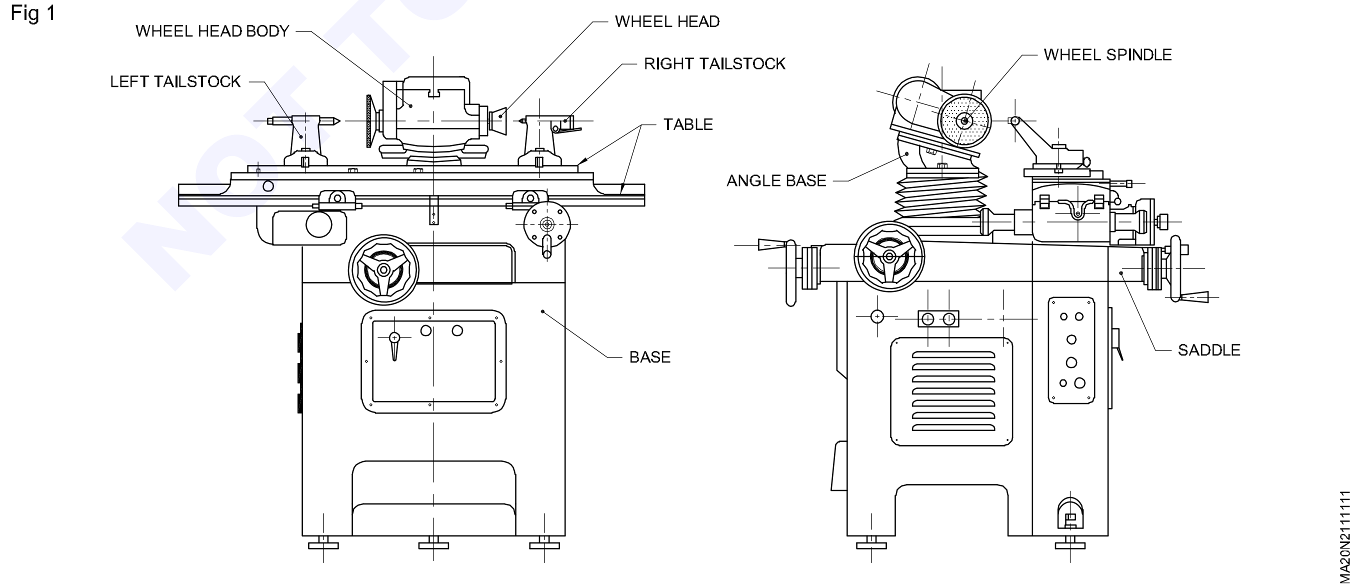
3 Maximum length of the job that can be ground.

4 Maximum breadth of the job that can be ground.

5 Type of drive
   - hydraulic
   - electrical.

6 Number of attachments.

Fig 1



Tooth rest

Objectives: At the end of this lesson you shall be able to
- **state the use of a tooth rest in cutter grinding operation**
- **explain the selection of right type of tooth rest holder and blade.**

Milling cutters have a number of teeth, each of which must be sharpened by a separate grinding operation. A tooth rest is used to support each individual tooth in the correct position for contact with the face of a grinding wheel.

Tooth rests vary in design, but consist essentially of a thin rest blade mounted in a holder attachment to the grinding machine. The blade may be flexible or rigid.

The rest must be mounted as close as possible to the wheel surface. The type of wheel used affects the location of the rest.

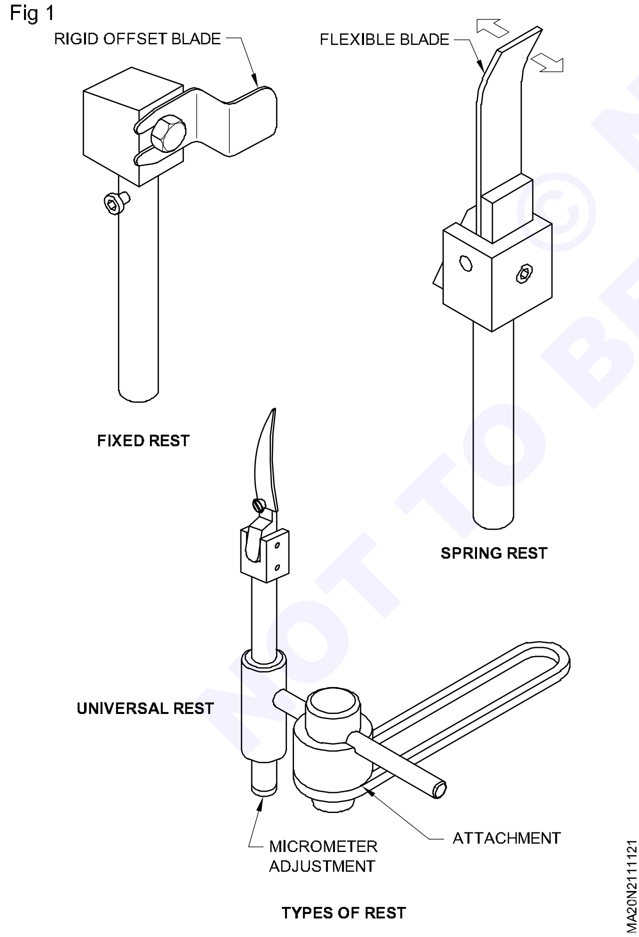
The tooth rest blade supports the tooth face and holds it in the position against the grinding wheel.

The blade prevents rotation of the cutter and enables the correct clearance angle to be set.

The tooth rest holder

The tooth rest holder is fitted into the tooth rest attachment and can be of one of three types. (Fig 1)

Fig 1



TYPES OF REST

i Fixed tooth rest holder

This one is usually used to the wheel head flange and used for the location of the teeth of straight face or fluted cutters. (e.g slab & helical mills)

ii pring tooth rest holder

Used for the location of the teeth of straight face or fluted cutters (e.g reamers) and the rest is fixed either to the work head or to the machine table.

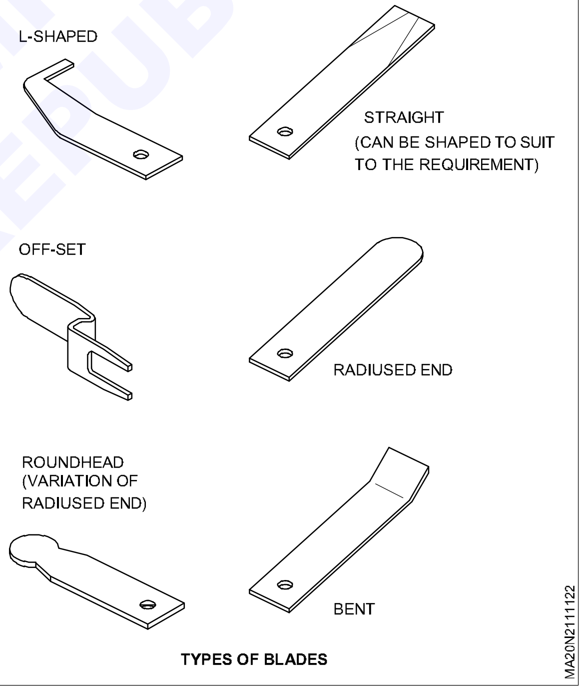
iii The universal tooth rest holder

This holder includes a micrometer screw adjustment to enable accurate setting of the tooth rest blade height and so sets the clearance angle for the tooth accurately before grinding starts.

The blade

There are many shapes of blades to suit a specific requirement. (Fig 2)

Fig 2



TYPES OF BLADES

i Straight

Used for grinding straight fluted cutters and fixed to a spring rest. It can be shaped to suit small work by grinding away each side, so as to reduce the width of the tip.

ii Radiused end

Fixed to a spring rest; used for grinding shell and small end mills.

Fixed to a fixed rest; used for grinding spiral fluted cutters.

iii L-shaped

Fixed to a spring rest; suitable for grinding straight fluted plain milling cutters of one tooth, woodruff cutters and slitting saws.

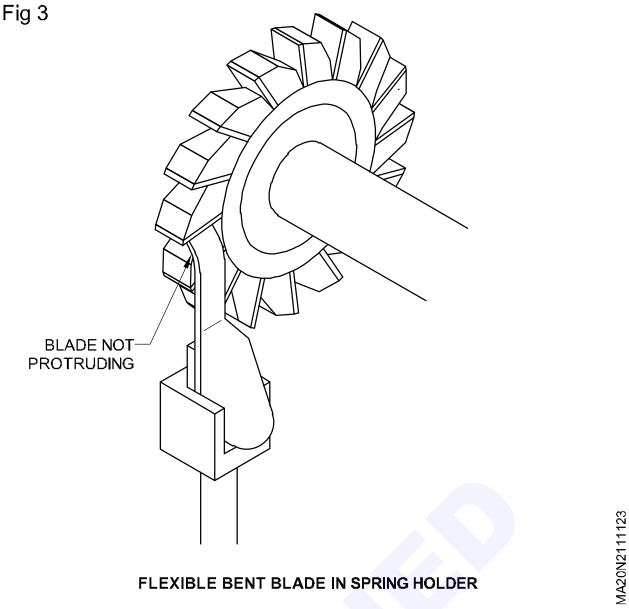
iv Off-set

Fixed to a fixed tooth rest; used for the grinding of large diameter spiral milling cutters with a coarse pitch and face mills with angular inserts.

v Bent

Fixed to a spring tooth rest used for grinding side and face cutters. The blade must not protrude beyond the cutting or side edges of the cutter. (Fig 3)

Fig 3



FLEXIBLE BENT BLADE IN SPRING HOLDER

The tooth rest may be mounted on the wheel head, the cutter head or the work table depending on the job.

The location of the tooth rest depends on

- the direction of rotation of the grinding wheel
- the wheel shape
- the types and dimensions of the cutter to be ground
- the desired clearance angle.

Methods of cutter grinding

Objectives : At the end of this lesson you shall be able to
- **explain the sharpening of slitting saw**
- **describe the importance of clearance angle of milling cutter**
- **state the methods of resharpening of the milling cutter.**

1 Sharpening a slitting saw

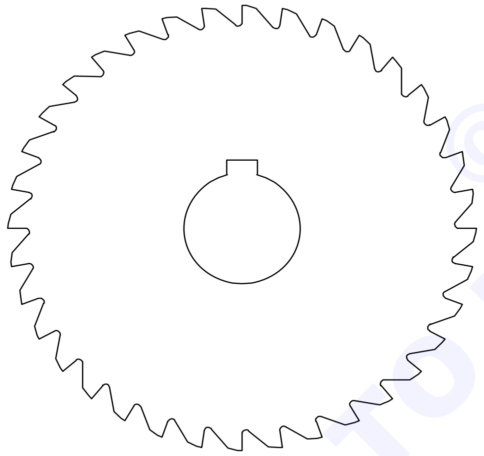
Slitting saw can be sharpened successfully on a universal tool and cutter grinder. Slitting saws are ground in the same way as milling cutters (Fig. 1)

- The saw may be held on mandrel or stub arbor which fits into the cutter head or it may be mounted on mandrel fixed between centres.
- A flaring cup wheel or taper cup wheel can be used to produce the required clearance angles.

To minimize the effect of wheel wear, the last cut should be very light a rule applying to all class of cutter grinding.

Clearance angles only ground for slitting saw.

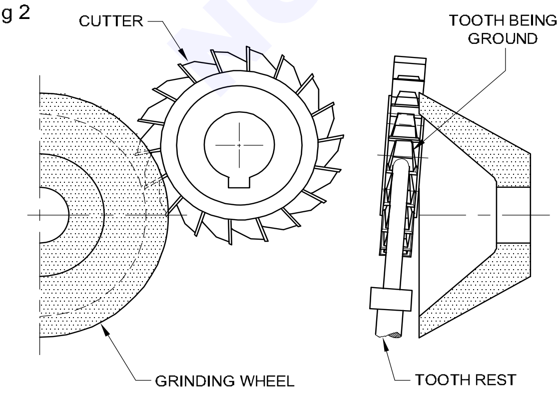
Fig 1



SLITTING SAW

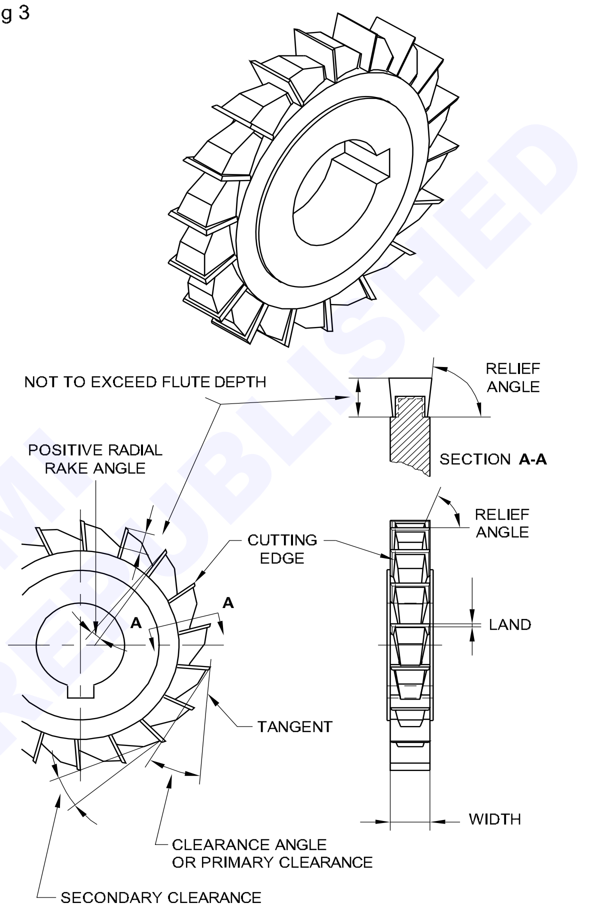
2 Sharpening of side and face cutter. (Figs 2&3)

Fig 2



GRINDING THE SIDE TEETH WITH A CUP WHEEL

Fig 3



SIDE AND FACE MILLING CUTTER

The peripheral teeth edges of a side and face milling cutter are ground as in the case of an end mill.

The edge on each side of each tooth must also be ground. These edges taper from the peripheral edge.

A cup wheel is used with the headstock tilted to provide the clearance angle. This also keeps the trailing face of the wheel away from the cutter.

Use a flexible tooth rest mounted on the headstock. When an edge has been ground, the side mill is simply rotated against the spring action of the tooth rest to position the next tooth for grinding.

The length of the primary clearance on the side teeth should not exceed the depth of the flute.

That part of the side teeth that extends below the flute depth is relieved by grinding back at a 3^o angle from the flute depth to the bottom of the side tooth.

Sharpening primary and secondary clearance on milling cutters

To increase the life of a milling cutter, it is to be re-sharpened as soon as it becomes blunt or dull.

Clearance angles

The most important point in cutter grinding is producing the correct relief or clearance behind the cutting edge.

This varies slightly according to

- The material of the job to be milled.
- The diameter of the cutter.
- The material of the cutter.

There are no hard and fast rules for determining the clearance but out of vast experience, the following angles may be taken as a guidance. Too little clearance causes bad surface finish, while too much clearance for chattering, causes chip off cutting edges.

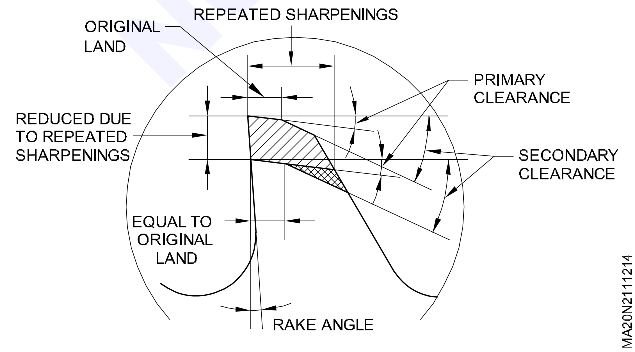
Materials to be cut	Clearance angle for H.S.S. cutters
Low carbon steels	5^{o} - 7^{o}
High carbon & alloy steels	4^{o} - 7^{o}
Cast iron	4^{o} - 7^{o}
Soft brass, bronze & copper	10^{o} - 12^{o}
Medium & hard bronze	4^{o} - 7^{o}
Aluminium, magnesium, Plastics etc.	10^{o} - 12^{o}

These angles are average for cutters between 50 to 150mm diameter. Minimum angles may be taken for larger dia. cutters and maximum angles for small cutter and end mills etc.

Secondary clearance

Sometimes where a heavy roughing cut is employed or where a cutter has been ground frequently until a wide land has resulted a secondary clearance, is ground to reduce the land to a nominal width. i.e. approximately 1mm to 1.5mm. (Fig 4) the recommended secondary clearance angles vary from 15^{o} to 25^{o}, depending upon the type, size and number of teeth of the milling cutter.

Fig 4



Centre setting

The basic principle of all set up is for the operator to first set the tooth to be ground at the same height as the centre of the cutter and then lower the tooth at an angle equal to the clearance required. This principle applied to all cutters for grinding both the peripheral and side teeth.

Fig 5 shows two types of centre setting gauge provided. The gauge 'A' which is common to the 10mm height of the centre and the gauge 'B; which is clamped to the top of the wheel head and carries a stop collar which is set of the wheel head and carries a stop collar which is set to correspond with the centre line of the wheel spindle.

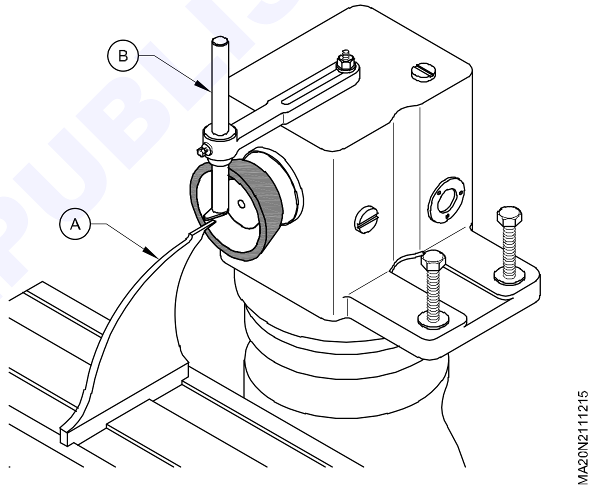
Setting for grinding clearance angles

The clearance angles can be obtained by means of three methods.

1 Linear setting

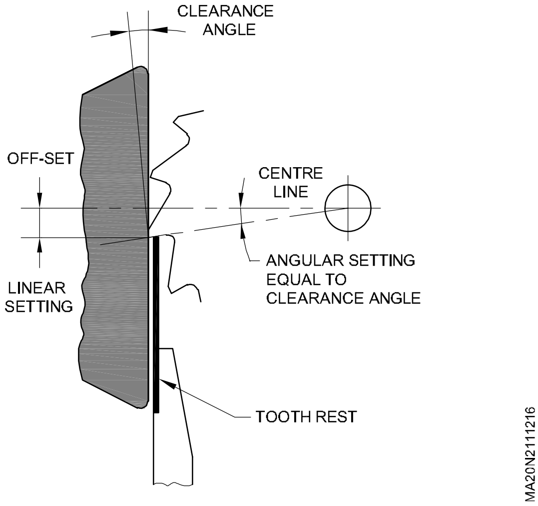
Adjust the height of wheel head until the tips of the height gauges in line with each other. (Fig 5)

Fig 5



Lower the wheel head by the amount calculated as below. (Fig 6)

Fig 6

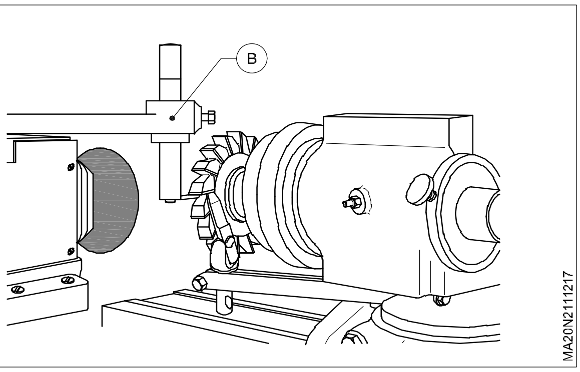


OFFSET=clearance angle x 0.0087 x dia. of cutter E.g. 7° x 0.0087 x 100mm (of cutter)

OFFSET = 6.09mm.

- Place the tooth on the center gauge 'B'. (Fig 7)

Fig 7

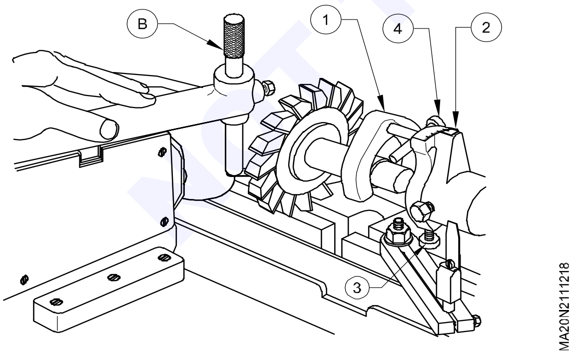


- Set the tooth rest under the tooth to be ground.
- Remove centre gauge 'B' and grind with the front face of the cup wheel, indexing from tooth to tooth until the whole cutter has been sharpened.
- Wheel head swivelling above 7° primary, secondary 25°

2 Clearance setting gauge

- The clearance angle setting gauge is a graduated carrier bracket, which fits to the left hand tailstock and can be adjusted radically through 15°. (Fig 8)
- Set the wheel center by the center setting gauge.
- Mount the cutter on the mandrel and fix the mandrel between the tailstock centers.
- Repeat the above steps for remaining teeth.
- Unlock the screw 'D' and index to next teeth. Lock the screw D and grind.
- Set the teeth to rest on the tip of the centre setting gauge.
- Set angle setting gauge (2) at (0°) and clamp the carrier (1) to the mandrel using the screw (4). (Fig 8)

Fig 8



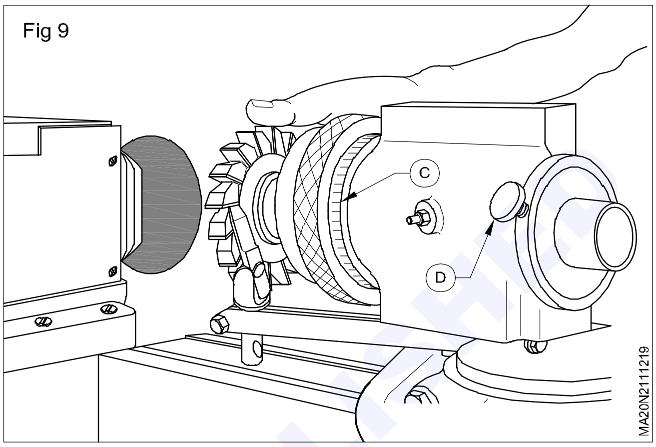
- Remove the center setting gauge and rotate the angle setting gauge (2) to the required clearance angle and lock firmly by the screw (3).
- Set the tooth rest under the tooth to be ground and remove the carrier (1).
- Start grinding by indexing from tooth to tooth.

3 Direct setting from cutter head

When the cutter is mounted on a stub arbor in the cutter head, the clearance angle can be obtained directly from the graduated dial (C) on the work head spindle. (Fig 9)

- Set one of the cutting edges at the center by center setting gauge.
- Note the reading on the graduated dial and rotate the spindle to the required number of degrees and lock the spindle in position by screw (D).

Fig 9



- Set the tooth rest under the tooth to be ground and commence grinding.

Clearance angle setting while using indexing drum for dividing. (Fig 10)

Fig 10

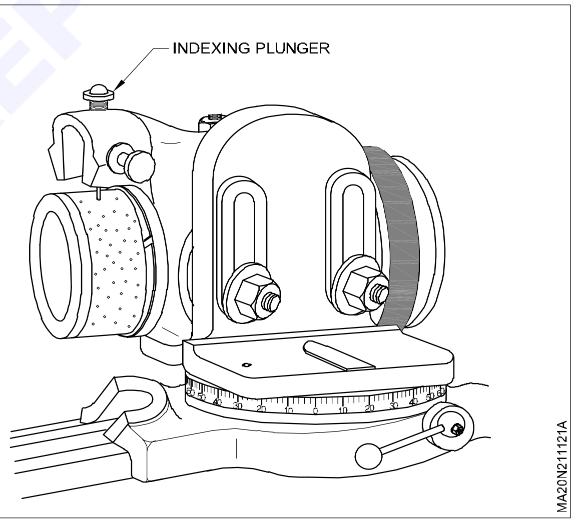


Fig 11 shows the details of the tooth setting to obtain clearance angle. The table 1 and table 2 provide the guide lines of setting at different angles corresponding to the wheel diameter.

"A" above centre for "FORWARD" grinding.

"A" below centre for "FRONT TO BACK" grinding.

This distance can also be calculated using the formula

"A"=Radius of wheel x sine of clearance angle.

'B' as for 'A' in table 1

'B'=Radius of cutter x sine of clearance angle.

Table 1

Milling cutter clearance tables

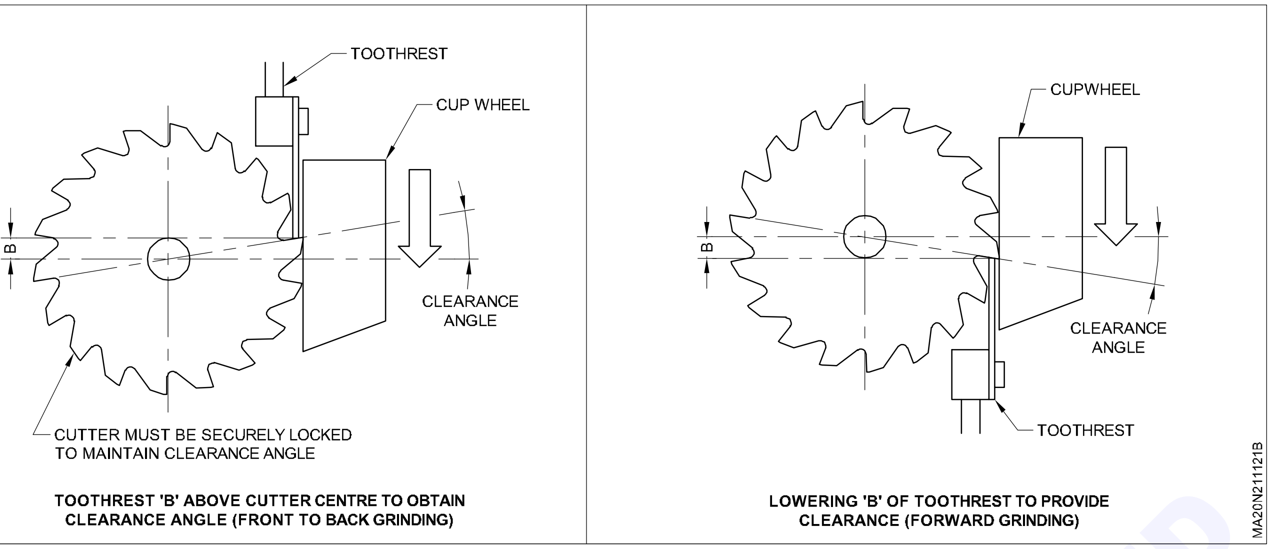
Straight Wheel Clearance								
"A" = Distance in mm to set wheel spindle center above or below center of work								
Diameter of grinding Wheel (mm)	"A" for clearance angle of							
	3°	4°	5°	6°	7°	8°	10°	12°
76	2	2.67	3.33	3.99	4.65	5.31	6.60	7.92
83	2.18	2.87	3.61	4.32	5.03	5.74	7.16	8.59
89	2.34	3.09	3.89	4.65	5.41	6.20	7.72	9.25
95	2.49	3.33	4.14	4.99	5.82	6.63	8.28	9.91
102	2.67	3.56	4.42	5.31	6.20	7.06	8.81	10.57
114	3.00	3.99	4.99	5.64	6.56	7.52	9.37	11.23
120	3.18	4.22	5.26	6.30	7.34	8.41	10.46	12.55
127	3.35	4.42	5.54	6.63	7.75	8.84	11.02	13.21
133	3.51	4.65	5.82	6.96	8.13	9.27	11.58	13.87
140	3.66	4.88	6.10	7.29	8.51	9.73	12.14	14.53
146	3.86	5.11	6.38	7.65	8.89	10.16	12.67	15.19
152	4.01	5.31	6.63	7.98	9.30	10.62	13.23	15.85

Table 2

Cup Wheel Clearance

"B" = Distance in mm to set end of tooth below center of cutter								
Diameter of Cutter (mm)	"A" for clearance angle of							
	3°	4°	5°	6°	7°	8°	10°	12°
13	0.33	0.43	0.56	0.66	0.76	0.89	1.09	1.32
19	0.48	0.66	0.84	0.99	1.17	1.32	1.65	1.98
25	0.66	0.89	1.12	1.32	1.55	1.78	2.21	2.64
32	0.81	1.12	1.37	1.65	1.93	2.21	2.77	3.30
38	0.97	1.32	1.65	1.95	2.31	2.64	3.30	3.96
50	1.32	1.78	2.21	2.67	3.09	3.53	4.42	5.28
63	1.65	2.21	2.77	3.33	3.86	4.42	5.51	6.60
76	1.98	2.67	3.33	3.89	4.65	5.31	6.60	7.92
89	2.31	3.09	3.89	4.65	5.41	6.20	7.72	9.25
102	2.64	3.56	4.42	5.31	6.20	7.06	8.81	10.57
114	2.97	3.99	4.99	5.97	6.96	7.95	9.93	11.89
127	3.30	4.42	5.54	6.63	7.75	8.84	11.02	13.21
152	3.36	5.31	6.63	7.98	9.30	10.62	13.21	15.85

Fig 11



TOOTHREST 'B' ABOVE CUTTER CENTRE TO OBTAIN CLEARANCE ANGLE (FRONT TO BACK GRINDING)

LOWERING 'B' OF TOOTHREST TO PROVIDE CLEARANCE (FORWARD GRINDING)

End-mills and shell end-mills sharpening

Majority of end-mills posses spiral cutting edges; the method for sharpening is same to that of sharpening a helical milling cutter but one additional set-up is required for grinding to end teeth.

The secret of obtaining long and useful life from end-mills lies simply in keeping them sharp on their ends. It does not matter much, how efficiently end-mills are sharpened on side cutting edges, best results cannot be achieved if end-teeth are not sharpened precisely.

End-mills with taper shanks are directly supported in the universal cutter-head; if of the shell types, they are first mounted on a suitable arbor or mandrel and then secured as a unit in the work-head or cutter-head. In operation, the cutter-head is swivelled about half-degree from zero setting in horizontal plane so as to get hollow effect on the teeth; whilst sharpening relief angle to end teeth, the cutter is swivelled vertically to the desired relief angle. The cup wheels are generally used in conjunction with plain disc wheels, which are mainly required for notching or secondary backing off.

Example

To sharpen a 2 - flutes end-mill at end teeth.

After sharpening side cutting teeth, the end teeth may be ground as under

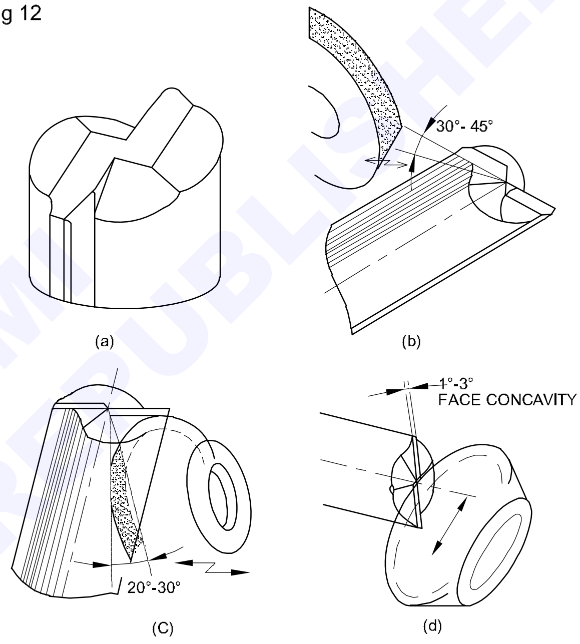
1 Clean the machine table and the bottom of the cutter-head and clamp it on the table. Insert the cutter in the cutter-head.

2 Fix the cup wheel to the wheel-head spindle and face end square of the end-mill to remove apparent corner wear as shown in Fig 12 (a). (It may also be done free-hand on a bench grinder).

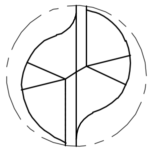
3 Clamp the tooth rest and position the end-mill for notch grinding.

4 Replace plain grinding wheel to the wheel spindle and tip the cutter-head 35°to 45° approximately upward; grind notch slightly past center so that end teeth are gashed radial, as shown at Fig 12 (b), and at zero degree Front rake.

Fig 12



(a) TO (d) GRINDING END TEETH TO 2-FLUTES END-MILL



(e) PLAN OF PROPERLY GROUND 2-FLUTES END MILL

TOOL AND CUTTER GRINDING

5 Swivel the cutter-head around 90° to bring the end teeth of the end-mill to the grinding wheel.

6 With the aid of center gauge, level one tooth in a horizontal plane with the table and lock in position with thumb screw.

7 Swivel the cutter upward to the desired secondary clearance and clamp in position.

8 Again mount flaring cup wheel to the wheel-head spindle and swivel 1° to 2° so as to prevent grinding from both sides of grinding wheel: grind secondary clearance at the heel of the tooth. The secondary

clearance may vary depending on the type of cutter and is usually 20° to 30°, as shown in Fig 12(c).

9 Swivel the cutter-head to 88° to 98° for developing end concavity (or hollow grind)

10 Tilt the work-head upward to get proper primary clearance angle, say 4° to 15°, depending upon the ability of material.

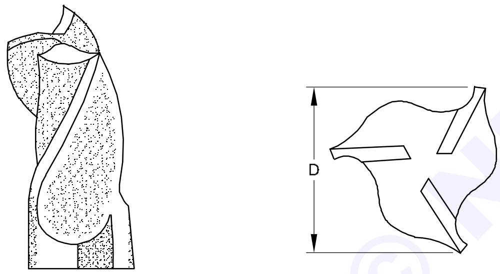
11 Grind the primary end clearance so that the two cutting edges are joined at the centre as shown in Fig 12 (d). The end teeth will look as shown at Fig 12 (e) when properly ground.

Example

To grind end teeth to the 3 - flutes end-mill.

The sharpening procedure to end teeth of 3-flutes end-mill is similar to that of sharpening end teeth a 2-flutes end-mill. Fig 13 shows a properly ground 3-flutes end-mill. This type of grind is centre cutting and tooth will not work out when plunge cut. Care should be taken that the teeth are exactly spaced around 120°apart. All the three teeth should have equal rake and clearance and are identically ground.

Fig 13



PROPERLY GROUND 3-FLUTES END-MILL

PLAN OF PROPERLY GROUND 3-FLUTES CENTRE CUTTING END-MILL

TOOL AND CUTTER GRINDING

MA20N211121D

Example: To grind end teeth to a four (or more) flutes end-mill.

The sequence order to be followed for grinding end teeth to four flutes end mill is as under

1 Grid off the face end square by off-hand method on a pedestal or bench grinder.

2 Clean the machine table and the bottom of the cutter-head and clamp it to the table. Insert the cutter in the cutter-head.

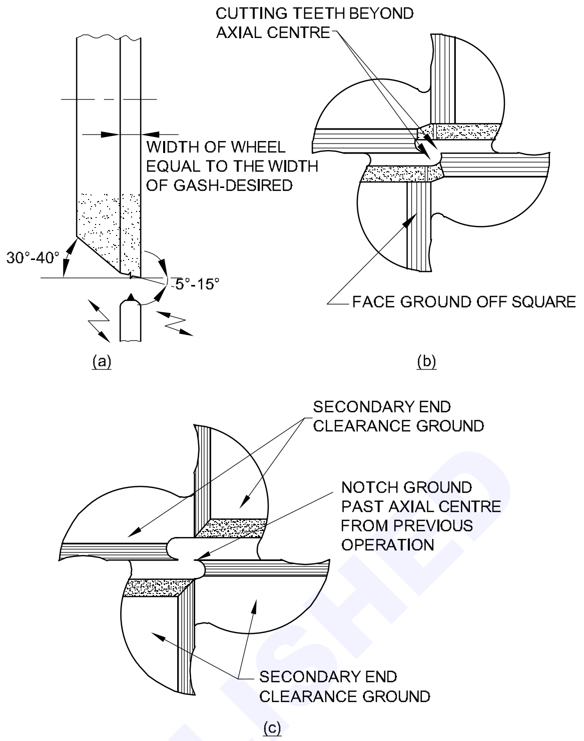
3 Mount the plain thin wheel or preferably saucer wheel to the wheel-head spindle. Dress it to the shape shown in Fig 14(a).

4 Clamp the tooth rest and position the end-mill for notch grinding.

5 Raise the cutter through 30° to 45° from vertical plane plane. Grind notch to the centre cutting teeth to 0° – degree. Tooth rake beyond the centre as shown in Fig 14(b)

6 Replace for straight wheel of wide width and grind secondary clearance to all four teeth to get same and identical land width as shown in Fig 14(c)

Fig 14



(a) TO (c) METHOD OF SHARPENING END TEETH TO 4-FLUTES END-MILL

TOOL AND CUTTER GRINDING

MA20N211121E

7 Change for flaring cup wheel to the wheel-head spindle to grind primary clearance.

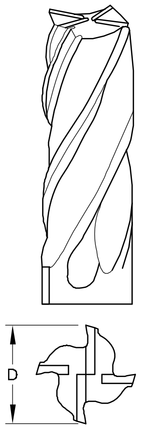
8 Swivel cutter-head to 89° for developing end concavity (or hollow grind).

9 With the aid of centering gauge, level one tooth in a horizontal plane with the table and lock in position with thumb screw.

10 Tilt the work-head upward to get proper clearance angle, say 7°.

11 Grind the primary clearance so that two center cutting-edges are joined together at the centre 'D'. The end-teeth will look like as shown in Fig 15

Fig 15



A PROPERLY GROUND 4-FLUTES 'E' END-MILL AND PLAN OF END TEETH

MA20N211122C

Example

To grind end-teeth to a shell end-mill.

The sequence of grinding operations is as under :

(a) Grinding end-teeth

1 Clean the machine table and place the cutter-head in position on it. Mount the cutter in the nose of cutter-head.

2 Fasten a flaring cup wheel Swivel the wheel-head to 90° setting so that it is faced towards the face of cutter. Turn backward 1° to 2° so as to prevent from grinding both sides of wheel when cutter is passed in front of it.

3 Swivel the cutter downward to the desired clearance and clamp in position, rotate at bottom 1° to get hollow grind.

4 Set the table parallel to its movement.

5 Level one tooth in a horizontal plane with the table using center height gauge.

6 Lock the cutter-head spindle in position with thumb screw.

7 Clamp tooth rest assembly to the bottom of cutter-head resting the tooth rest blade underneath the tooth to be ground.

8 Lower the wheel-head so that the tooth next to the one being ground will not foul the rim of the wheel.

9 Loosen the thumb screw of the cutter-head and proceed grinding by indexing tooth to tooth.

(b) Grinding on the corner of teeth

As stated earlier that wear tends to localize at the corner junction of the side and end cutting teeth: so it is good plan to grind chamfer on all shell end-mills; otherwise the cutter edges will become dull rapidly and poor finish on the work piece will be produced. The chamfer grinding on the corner is not simple as compared to the grinding of end teeth and side cutting edges, therefore, to get good performance from the end-mills and shell end-mills, corner sharpening should not be neglected. The procedure for grinding chamfer or corner is as under

10 Swivel the work-head to a convenient position so that the centring height gauge can be used from table top to centre a point approximately in the middle of the cutter chamfer in the same horizontal plane as the centre line of the work-head.

11 Lock the spindle of the work-head with thumb screw.

12 Swivel the work-head horizontally to the desired bevel angle, say 30-degrees, as shown on the base graduation in the figure appeared at the end of the book.

13 Adjust the wheel-head centre by using centring gauge from the table.

14 Set the clearance setting dial to work-head to zero and lock it in position.

15 Tilt and roll the work-head to the desired angle. In this case, say for achievement of a clearance of 5° at 30° chamfer angle, the axial tilt and radial roll comes to 2° 30' and 4° 3' respectively.

16 Lock work-head spindle by thumb screw again.

17 Mount the tooth rest assembly on the work-head with the tooth rest blade under the tooth to be ground.

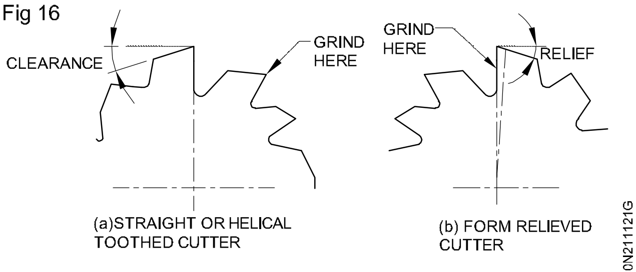
18 Loosen the thumb screw of the spindle-head and set the table dogs to length. Grind by indexing tooth to tooth.

Straight and spiral toothed cutters:

The first classification includes cutters sharpened on the periphery or the outside diameter and sides by grinding cutting and clearance angles behind the cutting edges. The selection includes cutters with straight and spiral teeth. such as key-way cutters, also mills slitting saws, reamers and angular cutter. (Fig 16a)

Form relieved cutters: The second class includes those cutters which have a definite contour and can only be sharpened by grinding the teeth radically or the front face of the clearance is provided during the manufacture of such cutters and need not be considered in resharpening operation. The examples are convex and concave cutters, gear cutters, taps and hobs and special form relieved cutters. (Fig 16b).

Fig 16



(a)STRAIGHT OR HELICAL TOOTHED CUTTER

(b) FORM RELIEVED CUTTER

THE TYPE OF CUTTERS AND METHODS OF SHARPENING

Sharpening of helical milling cutter

Objectives: At the end of this lesson you shall be able to
- **list the two stages of reconditioning of cutting face**
- **describe the procedure of gashing**
- **explain the procedure of grinding the cutting face.**

On repeated re-sharpening the cutting face of helical milling cutter becomes short of chip space. To increase the life and cutting efficiency of the cutter, it has to be reconditioned by deepening the helical flute, at the same time maintaining the original positive rake angle on the cutting face.

This reconditioning is done in two stages

1 Gashing

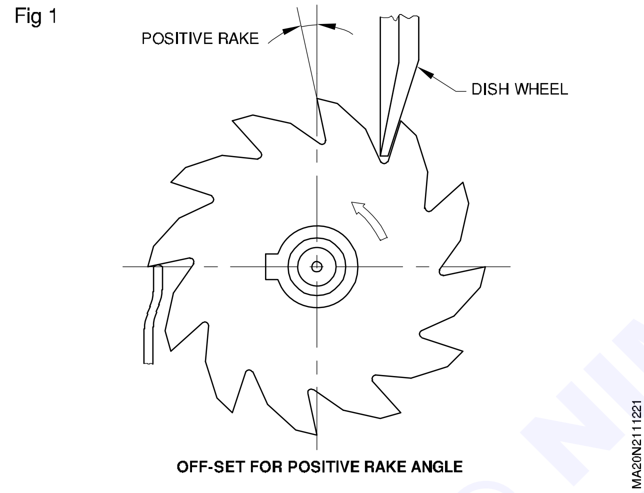
2 Grinding the cutting face

Gashing

Gashing or roughing is nothing but deepening the depth of flute. For this operation dish wheel is mounted and true on the extension spindle. Cutter is mounted on the mandrel and set in between centres.

The wheel head is to be swivelled and aligned with the hand of the helix and helix angle of the flute. By this wheel touches the bottom of the flute, not on the cutting face of the flute. Tooth rest is fixed to the wheel head and its blade is set touching the cutting face of the flute (Fig 1) set the table stops.

Fig 1



OFF-SET FOR POSITIVE RAKE ANGLE

Start the wheel head motor and commence grinding by traversing the table and at the same time keeping in constant touch with tool rest blade manually applying a turning force to the mandrel.

For successive depth of cut lower the wheel head until the cut reaches the required depth of flute.

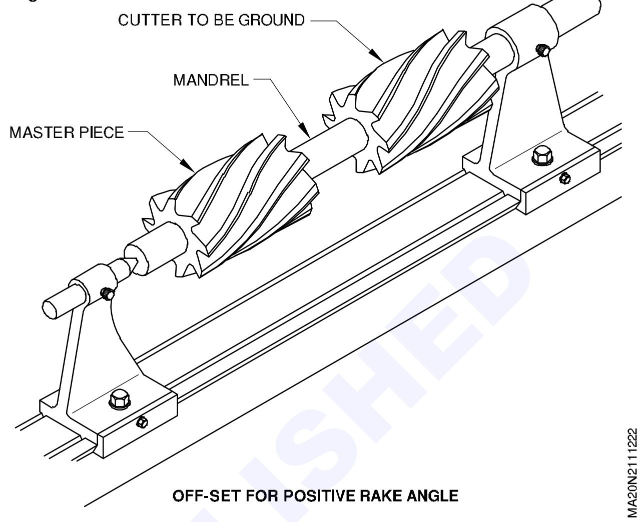
Grinding the cutting face

Dish wheel is to be mounted on the extension spindle is inverted position.

Select a master piece new helical fluted cutter of same number of teeth, helix angle, hand of helix and diameter (approximately) of the cutter to be sharpened.

Mount master piece and the cutter to be sharpened on a long mandrel (Fig 2) in perfect alignment and in between centres.

Fig 2



OFF-SET FOR POSITIVE RAKE ANGLE

The wheel is to be swivelled and aligned with the hand of helix and helix angle of the flute, so that the tapered face of the wheel perfectly matches with the cutting face of the tooth of the cutter to be ground.

Tooth rest with micrometer adjustment is to be fixed to the wheel head and set its blade touching the cutting face of the tooth of the master piece cutter.

Set table stop carefully not allowing over travel on the blade of the tooth rest, in order to control the helical path of the cutter.

Start the wheel head motor and commence the grinding by traversing the table and at the same time keeping in constant touch with the tooth rest blade by manually applying a turning force to the mandrel.

Same way grind the cutting face of all the teeth of the cutter.

Cutter grinding attachments and their uses

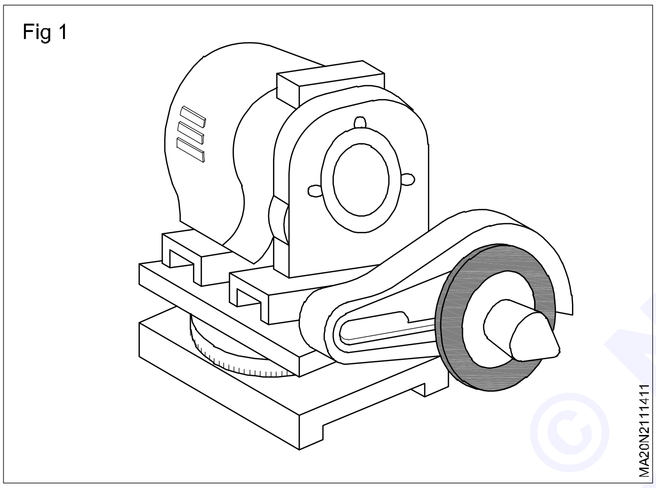
Objectives : At the end of this lesson you shall be able to
- **list out the attachments of tool and cutter grinder**
- **state the uses of the various attachments.**

Cutter grinding attachments and uses

1 Motor drive arrangement (Fig 1)

A motorized cutter head and also an internal grinding attachment can be supplied for cylindrical grinding, but it is a semi-precision accessory only. A cylindrical and internal grinder's scope in this case is limited. This attachment has a single speed only, with a motor, pulleys guard, starter with standard voltage and No.1 Morse taper centre.

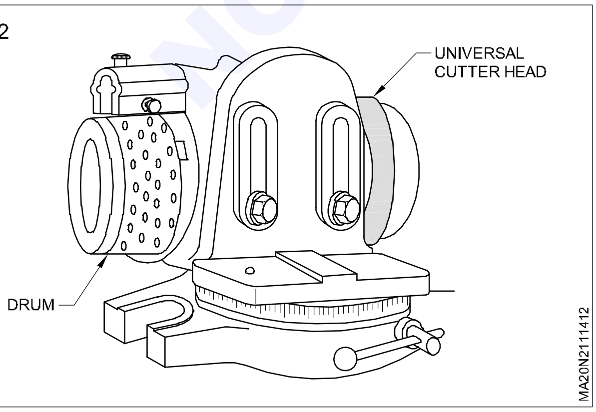
Fig 1



2 Positive indexing attachment (Fig 2)

This attachment is to be fixed on to the universal cutter head and is complete with a drum, having seven rows of holes 20, 30, 28, 24, 22 32 and 18 per row. To fit a positive indexing attachment to the universal cutter head, remove the catch and screw from the rear dust cap and fix the index drum on the taper nose of spindle bracket by means of a tee bolt and adjust the position to suit the number of divisions required. The final positioning of cutter teeth in relation to the drum, is obtained by adjusting the screws which rotates the plunger over the drum.

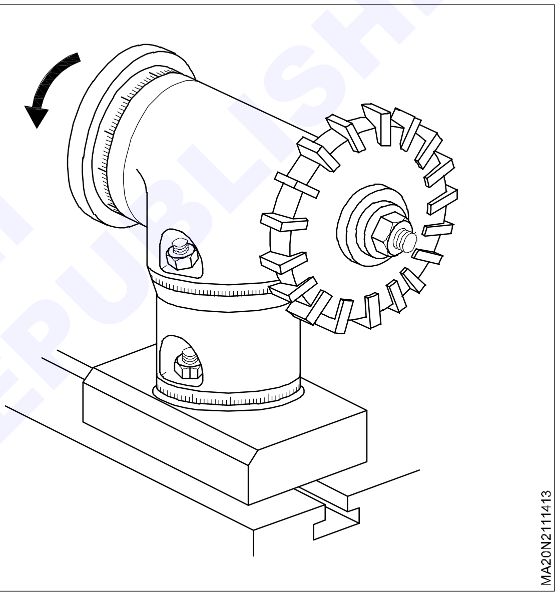
Fig 2



3 Face mill grinding attachment (Fig 3)

This attachment is for grinding face mills up to 400 mm dia. By means of an inclined face and dual swivels, the cutter can be positioned to any combined clearance angle desired, both from the cutting edge and also inwards to the centre of the cutter. It can also be swivelled, to grind a bevelled edge or the periphery of the cutter. The spindle is bored to ISO 40 taper. An adapter is provided for the smaller dia. mills.

Fig 3



4 Tap relieving and sharpening attachment

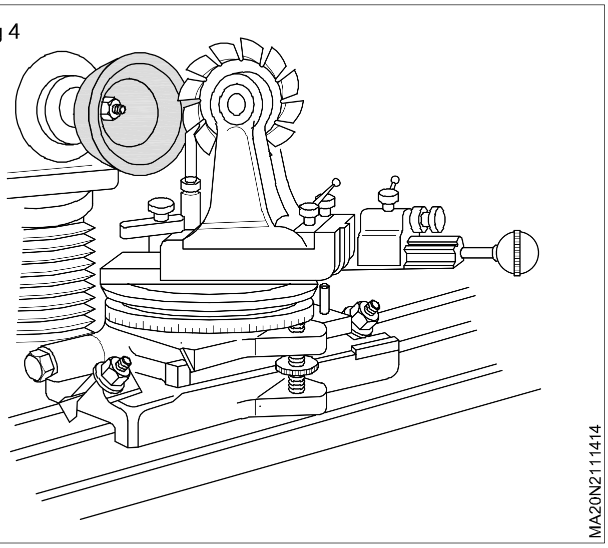
It is specially designed for grinding the relief of the lands of taps having 2,3,4,5,6 or 8 flutes, either with right or left hand threads and to grind edges on the flute faces of taps, reamers, milling cutters etc having divisions covered by the indexing gear. The cam shaft is housed in the fixed portion of the head, with the cams provided, which transmit an oscillating motion to the spindle and tailstock on the rocking frame. The work spindle is bored No.2 MT.

5 Radius grinding attachment (Fig 4)

For grinding radius on all forms of milling cutters, including side and face up to 180° convex, small face mills, end mills and concave cutters up to 90°, the setting is effected by inserting gauge blocks to the radius required.

The cutter heads are provided with interchangeable buttons for varying size bores and will also accommodate a diamond stick for forming a radius on the grinding wheel. A tilting head is also provided with a No.8 Brown and Sharpe taper adapter for holding end mills etc. The equipment includes tooth rests (for both forms for grinding) and gauge plugs.

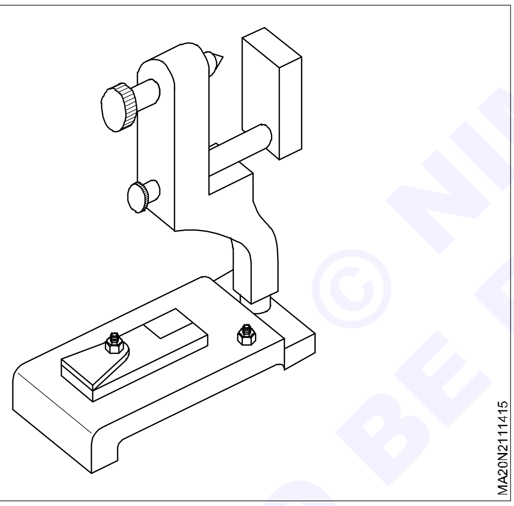
Fig 4



5 Radius wheel truing attachment (Fig 5)

A simple type radius grinding attachment for forming radius up to 25 mm on grinding wheels. The attachment is supplied with a setting gauge, which, in conjunction with slip blocks, enables the diamond to be set to dress the wheel to the required radius.

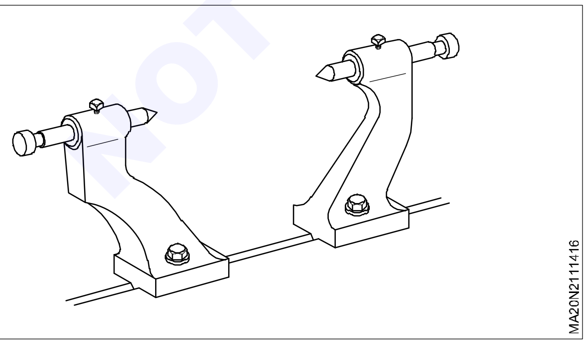
Fig 5



6 R.H and L.H extension tailstocks (Fig 6)

Specially designed to enable work up to 750 mm long, to be admitted between centres.

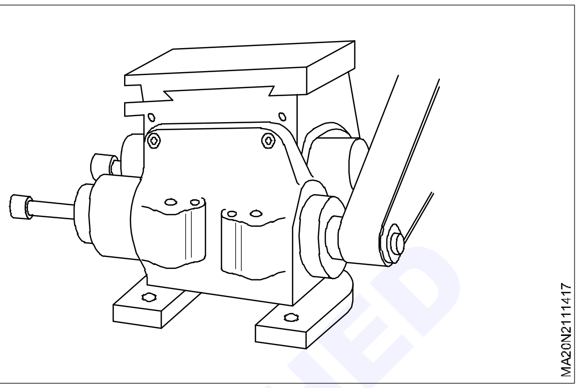
Fig 6



7 Internal grinding attachment (Fig 7)

This attachment bolts on top of the wheel-head and carries the internal grinding spindle. The drive to the internal grinding spindle is by an endless flat belt from a pulley which fits on to the wheel head spindle in place of the grinding wheel. Guards are removed for illustration purposes.

Fig 7

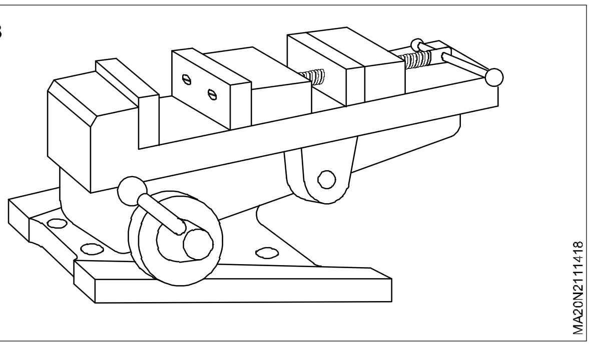


8 Angular sine vice (Fig 8)

This vice has been specially designed for precision grinding of angular parts. Sine bar setting enables the vice to be tilted and set accurately to any angle up to 55°. A hardened, ground and tapped steel plate is fitted on the upper side of the base on which standard gauge blocks are placed. The vice is then lowered until the roller rests on the gauge blocks.

The vice is arranged to lock at the swivel point, ensuring perfect rigidity. The jaws of the vice have hardened and ground steel facings.

Fig 8



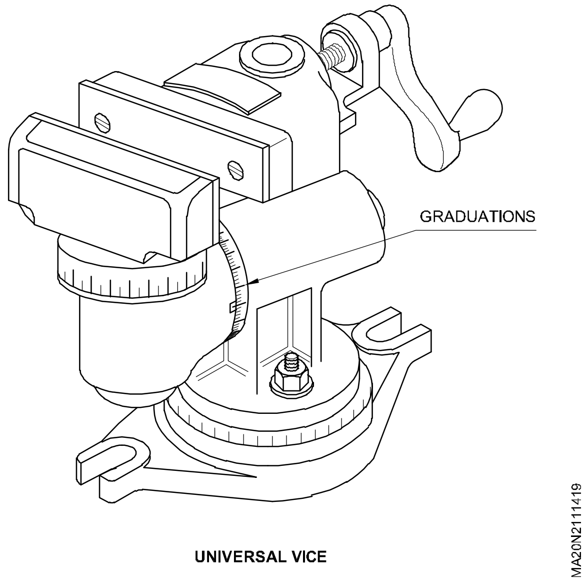
9 Universal vice (Fig 9)

This is designed specially for holding work to any desired compound angle. Three separate swivelling movements are provided, which are fully graduated.

10 Mandrel (Fig 10)

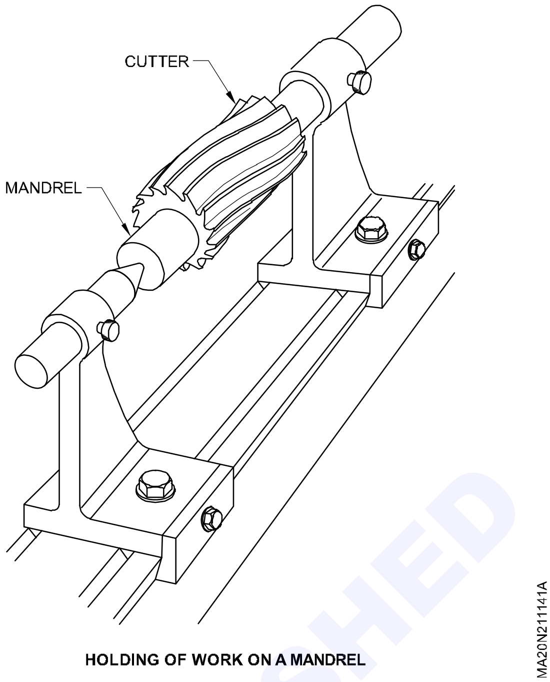
The mandrel may be mounted between R.H. and L.H. extension tailstocks on dead centres. R.H. and L.H. extension tailstocks are specially designed to enable work up to 750mm long to be admitted between centres.

Fig 9



UNIVERSAL VICE

Fig 10



HOLDING OF WORK ON A MANDREL

Geometrical tolerance

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 tolerance under three graphs of form, attitude and location.**

Definition of geometrical tolerance

Geometrical tolerance is the maximum permissible overall variation of form or position or measurements of a feature.

Reason for using geometrical tolerance

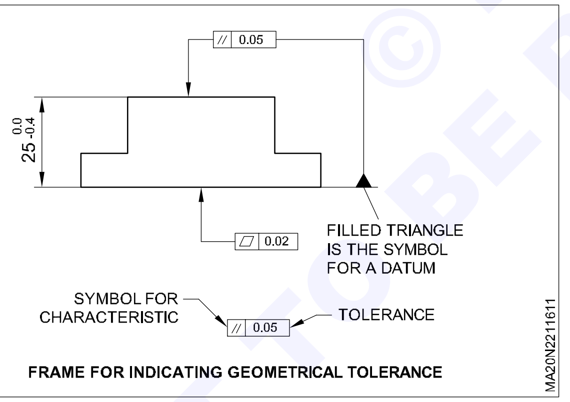
This will help the operator to produce the components, particularly those parts which must fit together precisely.

The idea is 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



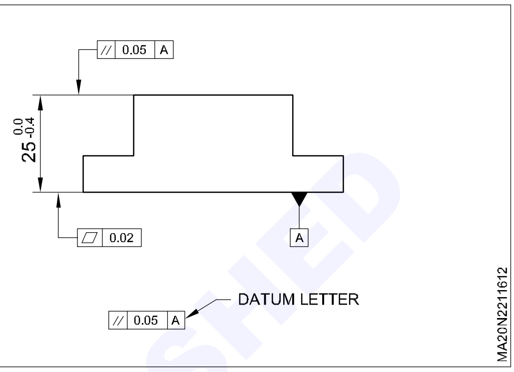
FRAME FOR INDICATING GEOMETRICAL TOLERANCE

You will notice that from each of the frames a leader is drawn so that it is normal, i.e. at 90° to the relevant face and ending with an arrowhead against the face.

Notice also that from the 'parallelism' frame, another leader is drawn terminating in a blacked-in equilateral triangle on a projection drawn out from the base line. The blacked-in triangle (about 4.5 mm high from base to apex) is the symbol used to represent a datum face or line.

An alternate method of arranging the frames and symbols is shown in Fig 2 where the datum is given a letter and a frame of its own with 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, attitude and location.

Form

Straightness, flatness, roundness, cylindricity and profile of a line and a surface.

Attitude

Parallelism, squareness and angularity.

Location

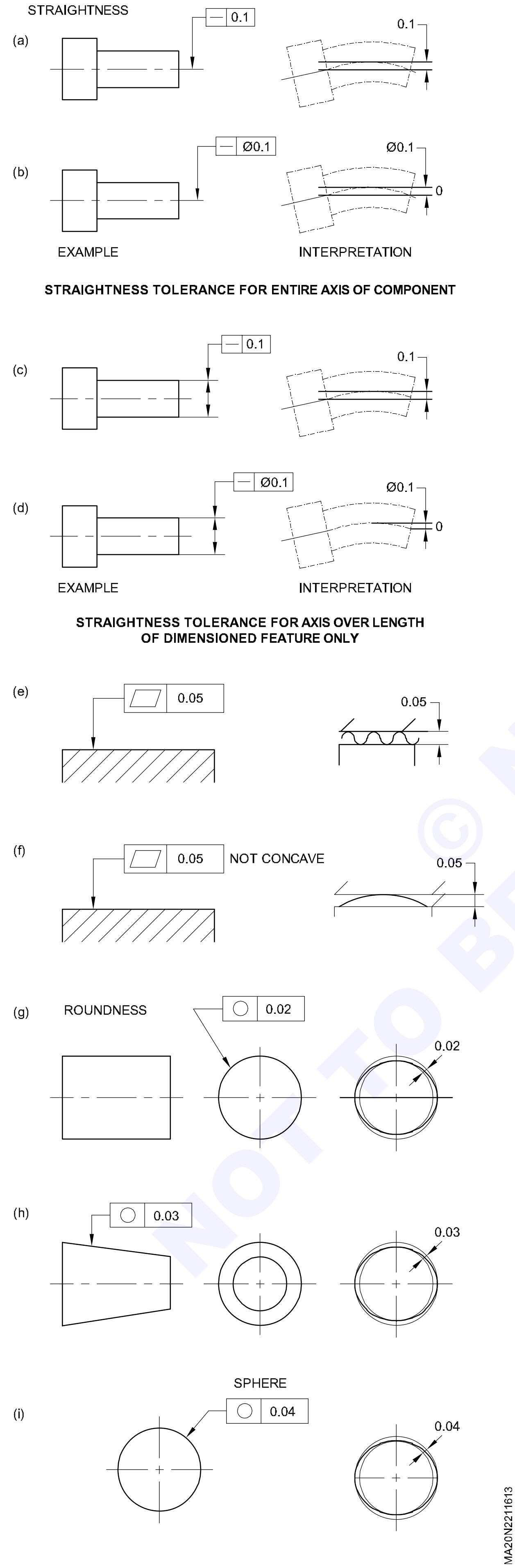
Position, concentricity and symmetry.

Tolerances of FORM are identified by the use of symbols for the following characteristics.

Characteristics	Symbol
Straightness	—
Flatness	▱
Roundness	○
Cylindricity	⌭
Profile of a line	⌒
Profile of a surface	⌓

The application of symbols is indicated in Fig 3, where (3a), (3b), (3c) and (3d) show the use of geometrical tolerances controlling the straightness of a circular section part. In (3a) and (3b) the leader lines from the tolerance frame end in an arrowhead 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

Fig 3



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 proceed the tolerance. This means that the entire main axis must lie within a cylindrical tolerance zone 0.1 mm diameter.

Fig (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(3e) and Fig(3f) deal with the geometrical tolerance for flatness of a surface, where the symbol for flatness is followed by the tolerance Fig of 0.05. This fig indicates that the actual surface (as previously shown in Fig 1) must be between two parallel planes 0.05 apart. If a particular form of direction is prohibited, then this is stated in a note form against the tolerance frame. Eg. 'Not concave'.

The geometrical tolerance controlling the roundness of a part is shown in Fig (3g), (3h) and (3i). The interpretation for (3g) and (3h) is that the true form of the periphery of the part at any cross-section perpendicular to the axis must lie between two concentric circles whose radial distance apart is 0.02 for (3g) and 0.03 for (3h).

For the sphere shown in (3i) the geometrical tolerance applies to concentric circles with the radial distance 0.04 apart at the periphery at any section of maximum diameter.

The sphere controlling cylindricity is shown in Fig 4. Here the interpretation shows, that for acceptance, the surface of the part must be within two coaxial cylinders, whose radial distance apart is 0.05.

Fig 4

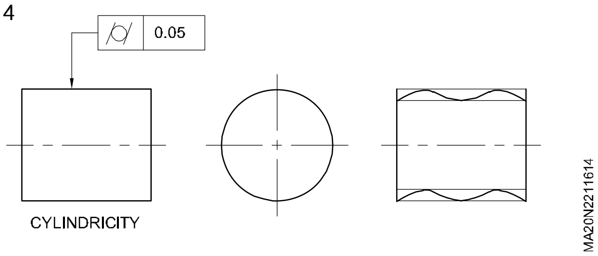


Fig 5 shows the method of applying a geometrical tolerance to a curved surface. The symbol is followed by the tolerance 0.05 which means that the actual surface must lie between two surfaces enveloping a succession of sphere 0.05 diameter whose centre lies on the theoretical surface.

In Fig 6 the geometrical tolerance is applied to linear dimensions controlling the profile. The rectangular 'boxes' around the 250 centre dimension and the 50 radius is the method used to indicate theoretical dimensions i.e. the dimensions relevant to perfect form.

The interpretation of the geometrical tolerance is that the actual profile must be between two lines which touch a succession of circles 0.2 dia. whose centre lies on the theoretical profile.

Fig 5

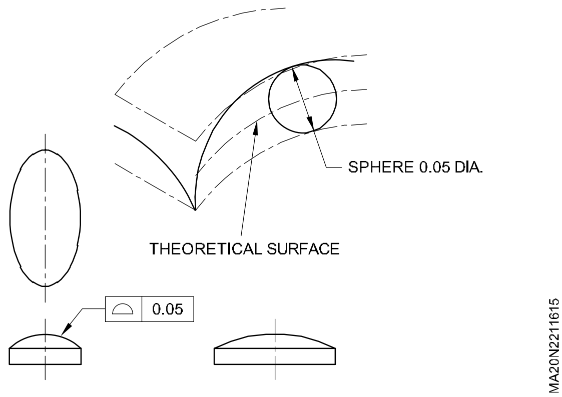
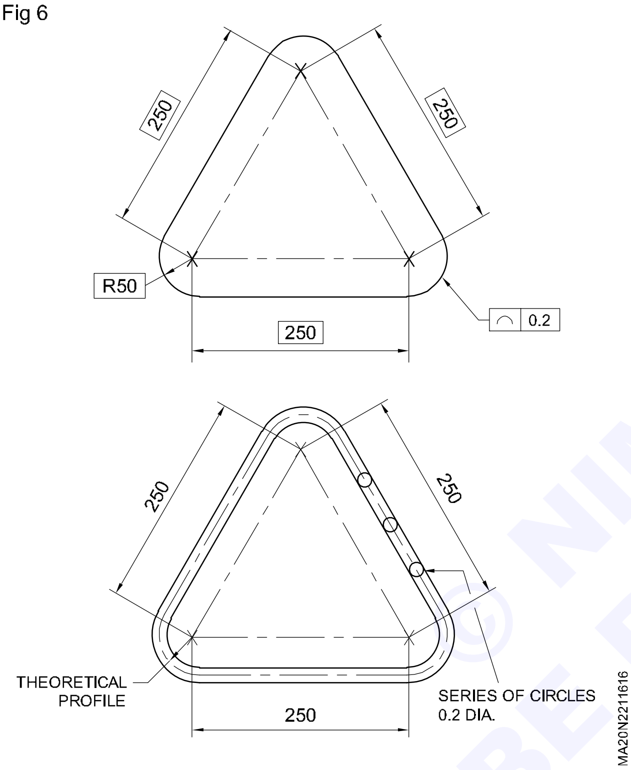


Fig 6

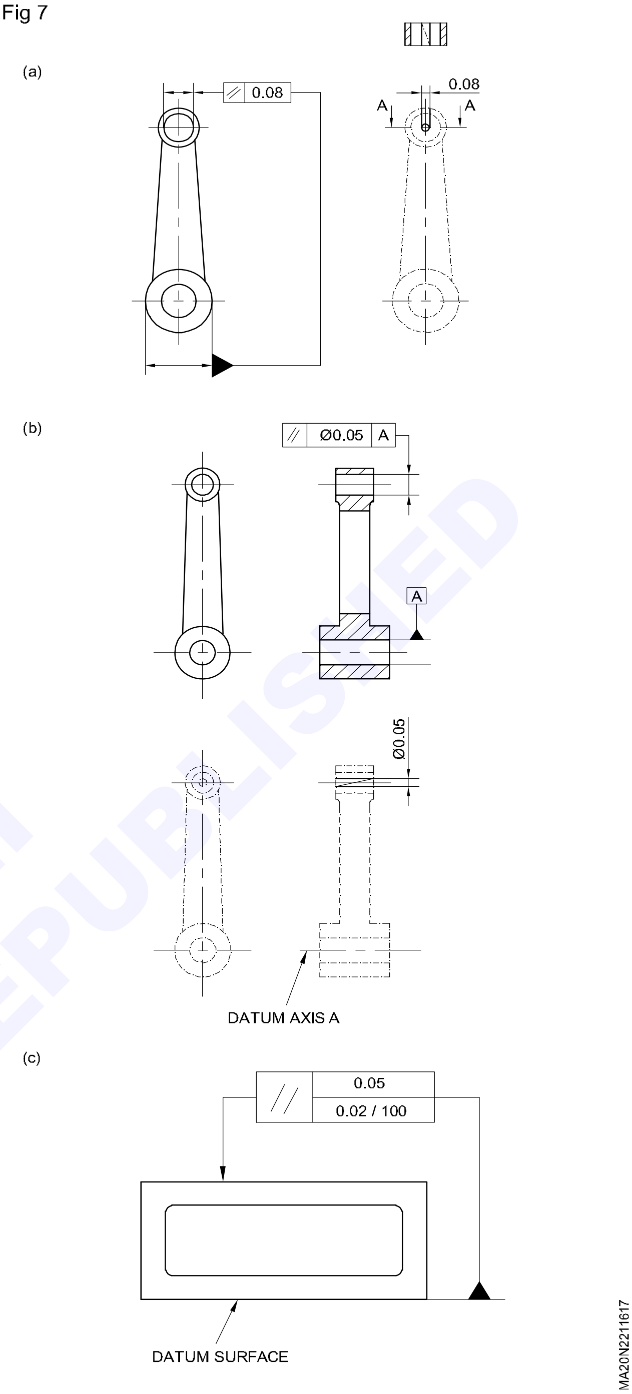


Attitude

Tolerances of attitude are indentified and indicated by the use of symbols for the following.

Characteristic	Symbol
Parallelism	//
Squareness	⊥
Angularity	∠

Fig 7



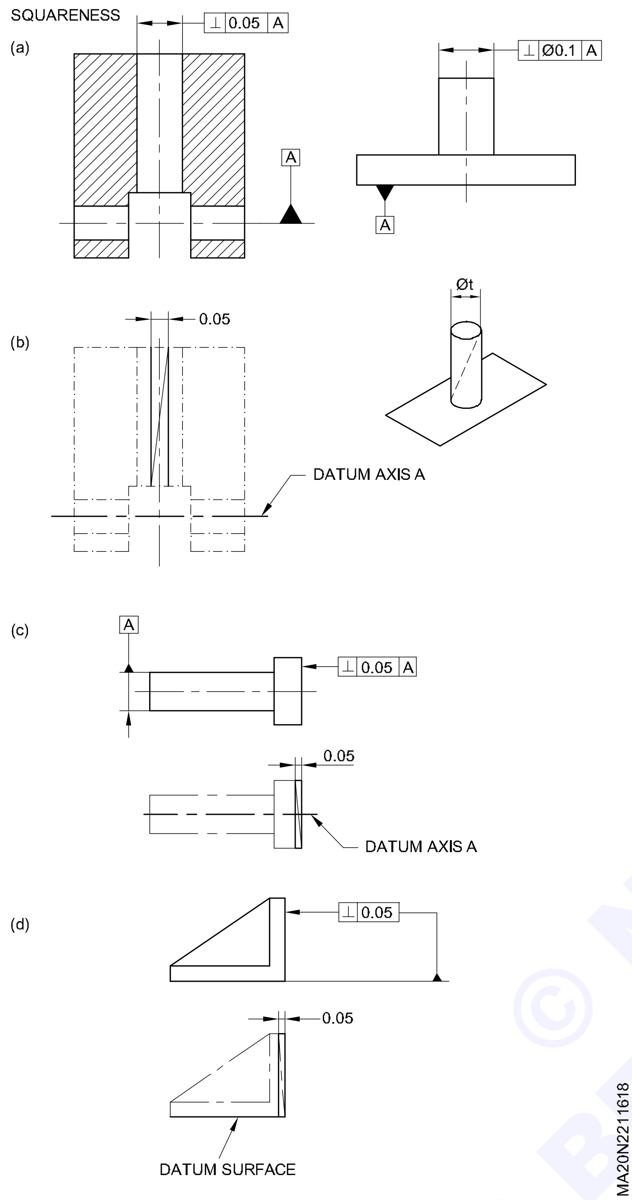
A typical application of tolerances for these three characteristics is shown in figures 7, 8 and 9. Fig (7a), (7b) and (7c) show the application of tolerancing to control 'parallelism'. (7a) shows that the axis of the upper hole must lie between the two lines 0.08 apart the lower hole, as indicated by the leader ending in the blacked-in triangle. In (7b) the method uses a separate datum letter 'A' which is added to the frame after the tolerance of 0.05 diameter. (Note the symbol is m.) 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 (7c) shows a component whose upper surface must be between two parallel planes 0.05 apart, parallel with the bottom datum surface. While the overall tolerance zone is 0.05 as shown in the upper section of the frame, the fig 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 (8a), (8b), (8c) and (8d) with (9a), (9b) and (9c) using the separate box method for indicating the datum.

The interpretation is as follows.

The axis of the vertical hole must be between two parallel lines, 0.05 apart, which are perpendicular to the common datum axis 'A' of the two horizontal holes. (8a)

Fig 8



The axis of the upper cylindrical portion must lie within a cylindrical tolerance zone of 0.1 diameter, the axis of which is perpendicular to the datum axis 'A'. (8b)

This shows that the right hand end face must lie between two parallel planes 0.05 apart, which are perpendicular to the datum axis, (8c).

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. (8d)

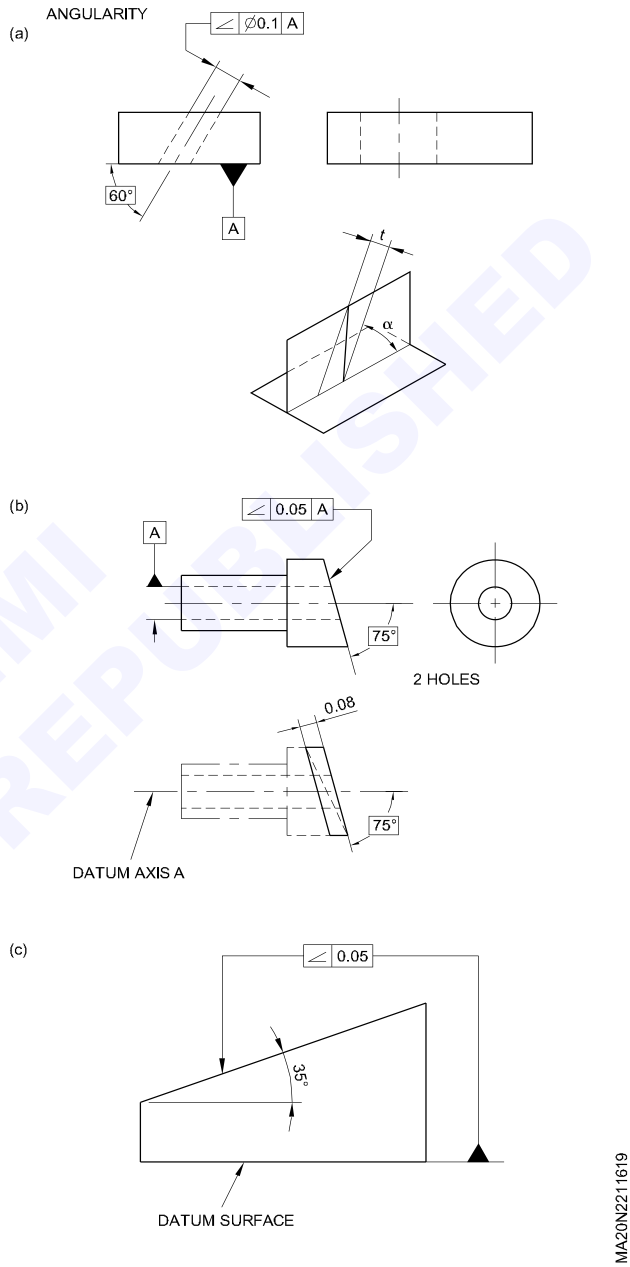
Geometrical tolerances for the control of ANGULARITY are shown in Fig (9a), (9b) and (9c).

The Fig (9a) shows that the requirement is the axis of the hole must lie within the cylindrical tolerance zone 0.1 diameter, the axis of which must be included at the theoretical angle of 60°to the datum surface A.

In (9b) the requirement is that the right hand end face must lie within the two parallel planes 0.08 apart which are inclined at the theoretical angle of 75°to the datum axis A of the through hole.

Fig (9c) shows a component whose upper angle face, must lie between 0.05 apart which are inclined at the theoretical of 35° to the base, the datum surface. Notice that the theoretical angle in each example is boxed.

Fig 9



Location

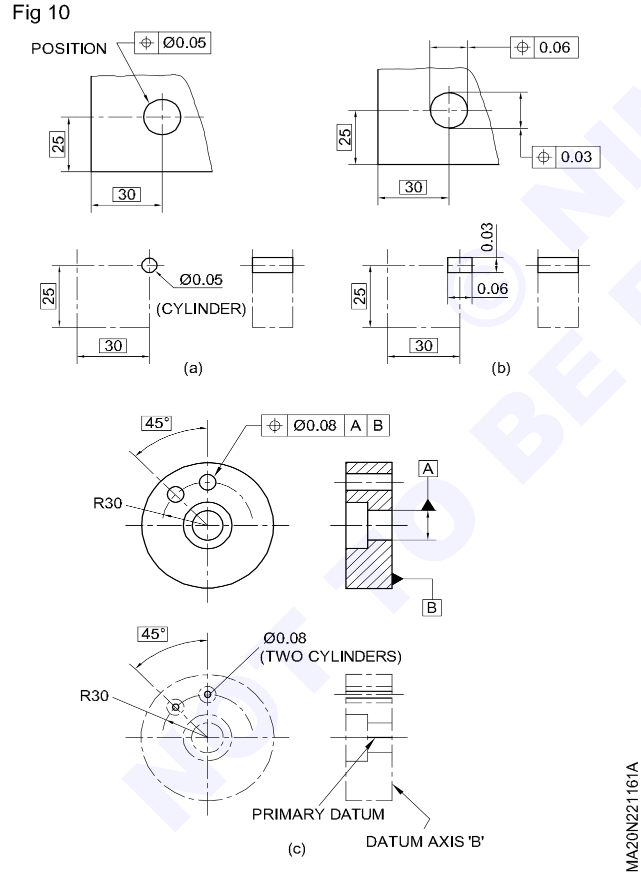
Tolerances of location are identified and indicated by the use of symbols for the following characteristics.

Characteristics	Symbol
Position	⌖
Concentricity	◎
Symmetry	⌯

Fig (10), (11), (12) show typical examples of these characteristics and symbols. In Fig (10a) the hole centre dimensions of 25 and 30 are boxed to show that these are the theoretical dimensions. The geometrical tolerance requires that the hole centre must lie within a cylindrical zone 0.05 diameter. The use of theoretical positions, also known as 'true positions', implies that the axis of the cylinder is square with the plane of the drawing. Fig (10b) 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.

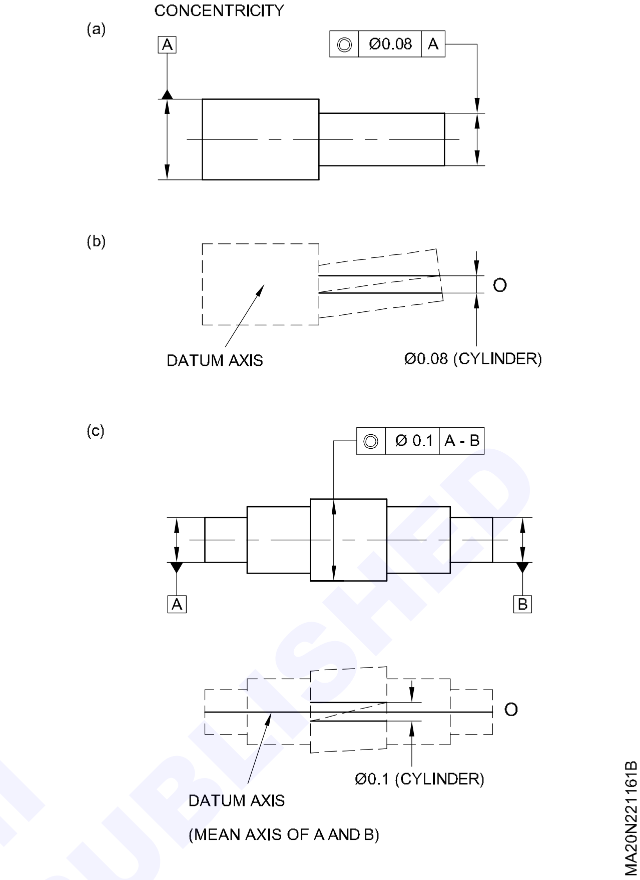
In Fig (10c) the two holes are shown with their true position spaced at 45° on a 30 mm pitch circle radius. The geometrical tolerance shows that each actual hole centre must lie within a cylindrical zone 0.08 diameter whose axis lies at the theoretically exact true centre position. The tolerance cylinders 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 10



Examples of geometrical tolerance for 'CONCENTRICITY' are given in Fig (11a), (11b) and (11c). The interpretations are as follows.

Fig 11



In Fig (11a) 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 (11b) the axis of two end portions must lie within a cylindrical tolerance zone 0.08 dia.

Fig (11c) shows that the axis of the large central portion must lie within a cylindrical zone 0.1 diameter which is co-axial with the mean axis of the datum diameters 'A' and 'B'. (Notice that to indicate the requirement of the mean axis the datum letters are separated by a hyphen and enclosed in the same compartment of the tolerance frame.)

The geometrical tolerance of 'SYMMETRY' is indicated in Fig (12a), (12b) and (12c) where the interpretations are:

- In Fig(12a) 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'
- In Fig (12b) the mean plane of the slot must be between two parallel planes 0.05 apart symmetrically disposed about the mean plane of the datum width 'W'.
- In Fig(12c) the median planes of the two end slots must be between two parallel planes, 0.06 apart.

Fig 13 gives details of geometrical tolerances.

Fig 12

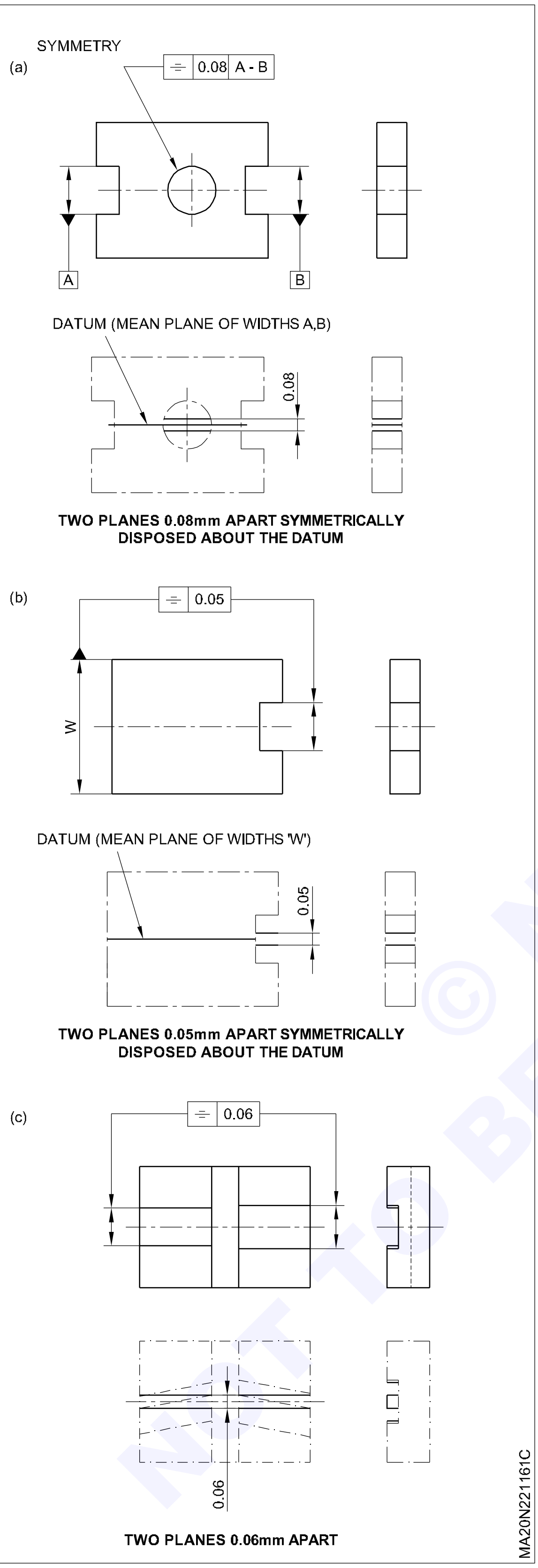
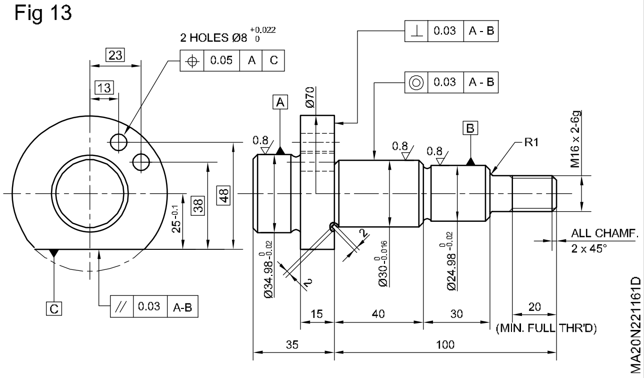

SYMMETRY
(a)
0.08 A - B
A
B
DATUM (MEAN PLANE OF WIDTHS A,B)
0.08
TWO PLANES 0.08mm APART SYMMETRICALLY DISPOSED ABOUT THE DATUM
(b)
0.05
W
DATUM (MEAN PLANE OF WIDTHS 'W')
0.05
TWO PLANES 0.05mm APART SYMMETRICALLY DISPOSED ABOUT THE DATUM
(c)
0.06
0.06
TWO PLANES 0.06mm APART
MA20N221161C

Fig 13


2 HOLES Ø8 +0.022 0
0.05 A C
23
13
0.03 A - B
0.03 A - B
A
Ø70
0.8
0.8
0.8
B
R1
M16 x 2-6g
48
38
25-0.1
C
0.03 A-B
Ø34.98 0 -0.02
2
2
Ø30 0 -0.016
Ø24.98 0 -0.02
ALL CHAMF.
2 x 45°
15
40
30
20
(MIN. FULL THR'D)
35
100
MA20N221161D

Depth micrometer

Objectives : At the end of this lesson you shall be able to
- **state the constructional features of a depth micrometer**
- **explain about graduation and least count**
- **state the uses of depth micrometer**
- **list the safety precautions of depth micrometer.**

Constructional features

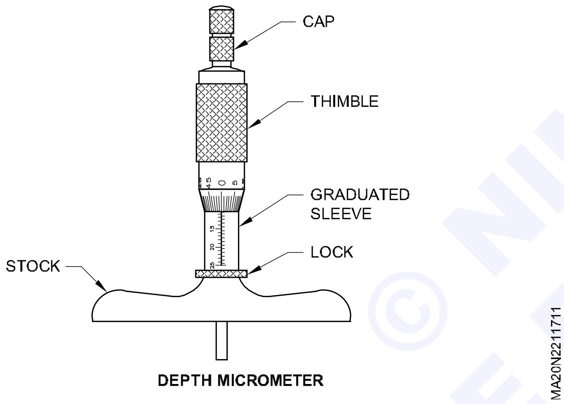
The depth micrometer consists of a stock on which a graduated sleeve is fitted.

The other end of the sleeve is threaded with a 0.5mm pitch 'V' thread.

A thimble which is internally threaded to the same pitch and form, mates with the threaded sleeve and skills.

The other end of the thimble has an external step machined and threaded to accommodate a thimble cap. (Fig 1)

Fig 1

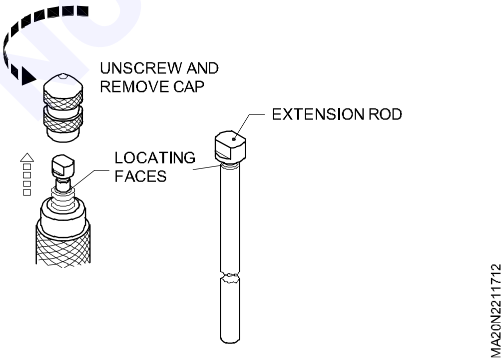


A set of extension rods is generally supplied. On each of them the range of sizes that can be measured with that rod is engraved as 0-25, 25-50, 50-75, 75-100, 100-125 and 125-150.

The extension rods can be inserted inside the thimble and the sleeve.

The extension rods have a collar head which helps the rod to be held firmly (Fig 2)

Fig 2



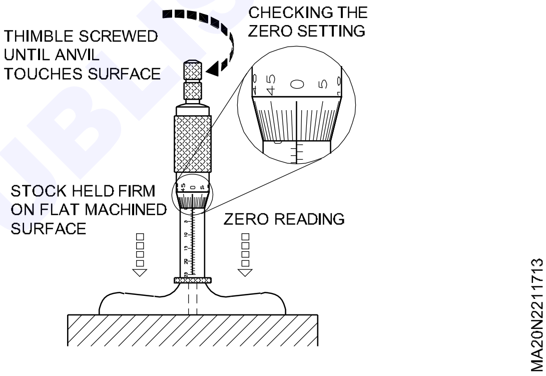
The measuring faces of the stock and the rods are hardened, tempered and ground. The measuring face of the stock is machined perfectly flat.

The extension rods may be removed and replaced according to the size of depth to be measured.

Graduation and least count

On the sleeve a datum line is marked for a length of 25 mm. This is divided into 25 equal parts and graduated, each line representing one millimetre. Each fifth line is drawn a little longer and numbered. Each line representing 1mm is further sub divided into two equal parts. Hence each sub division represents 0.5 mm. (Fig 3)

Fig 3

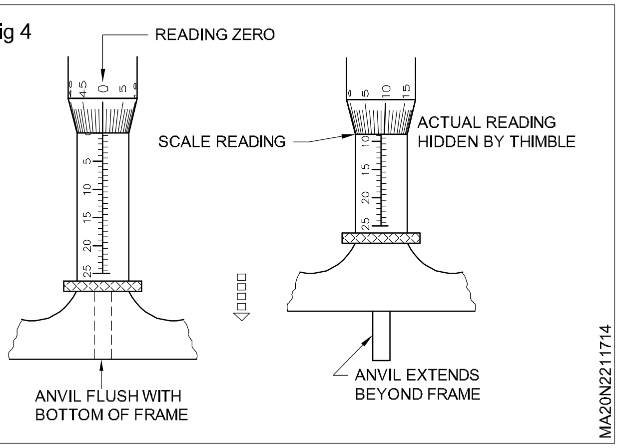


The graduations are numbered in the reverse direction, to that marked on an outside micrometer.

The zero graduation of the sleeve is on the top and the 25 mm graduation near the stock.

The bevel edge of the thimble is also graduated. The circumference is equally divided into 50 equal parts and every 5th division line is drawn a little longer and numbered. The numbering is in the reverse direction and increases from 0, 5, 10, 15, 25,30,40, 45 and 50 (Fig 4)

Fig 4



The advancement of the extension rod for one full turn of the thimble is one pitch which is 0.5 mm.

Therefore, the advancement of the extension rod for division movement of the thimble will be equal to 0.5/50 = 0.01mm.

This will be the smallest measurement that can be taken with this instrument,and the accuracy of the instrument is 0.01mm

Uses of depth micrometer

Depth micrometer are special micrometers used measure:

- The depth of holes
- The depth of grooves and recesses
- The heights of shoulders or projections.

Safety precautions

- Make sure micrometer and items to be measured are clean and free of dirt or debris.
- Make sure micrometer is zeroed.
- Never overtighten it.
- While measuring, make sure that the base is properly resting on the reference surface.
- Make sure that the interchangeable extension rods are disassembled from head of a depth micrometer before storing.
- Always take multiple measurements.

Different types of micrometer and its uses

Objectives : At the end of this lesson you shall be able to
- **name the different types of micrometers other than regular micrometers**
- **state the specific use of each micrometer.**

In addition to regular micrometers, there are several other types of micrometers, with the same fundamental principle, but specifically designed to meet the various special applications, such as external, internal, depth measurement etc.

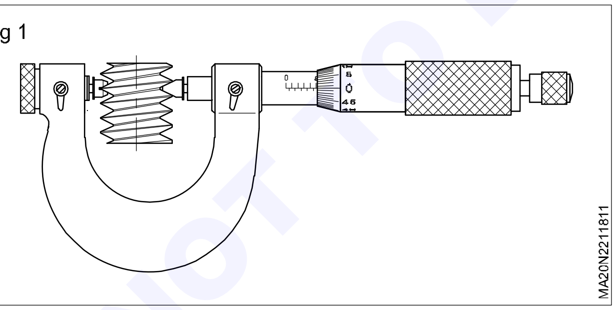
Types of micrometers other than regular

- Screw thread micrometer
- Tube micrometer
- Digital micrometer
- Depth micrometer
- Flange micrometer
- Ball micrometer
- Stick micrometer
- External micrometer with interchangeable anvils
- Keyway depth micrometer

Screw thread micrometer (Fig 1)

A screw thread micrometer is similar to an outside micrometer except that the spindle is pointed to fit between 60° V threads, and the anvil is shaped to fit over 60° V thread. It is used to measure the pitch diameter of the thread. Screw thread micrometers are available in many sizes depending on the pitch of the thread to be measured.

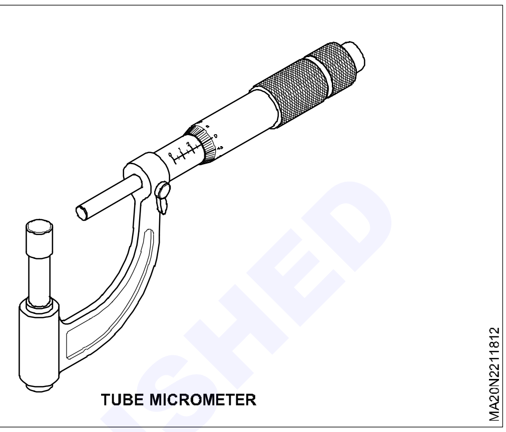
Fig 1



Tube micrometer (Fig 2)

A tube micrometer is specially designed to measure the thickness of the material of piping, tubing and and other parts of similar shapes.

Fig 2

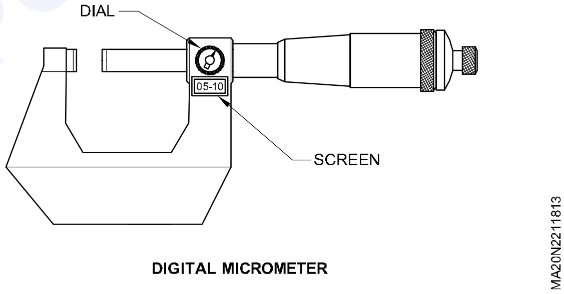


TUBE MICROMETER

Digital micrometer

This type of micrometer has got a dial on the frame of the micrometer and an illuminated screen below it. The dial pointer has an internal connection with the micrometer screw for measuring. The graduations on the sleeve and thimble are the same as on a regular micrometer. This micrometer is used to measure the dimensions similar to those measured by the outside micrometers, and the reading can be noted. (Fig 3)

Fig 3



DIGITAL MICROMETER

The advantage of this micrometer is, the readings are seen on the screen or the dial directly, without any difficulty. We need not look on the sleeve or the thimble scale coincidence. This avoids errors in reading and saves time. A layman can also read the measurement directly.

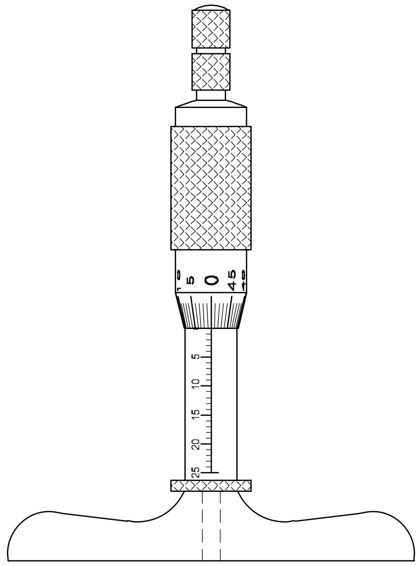
Depth micrometer (Fig 4)

A depth micrometer is designed to measure accurately the depth of grooves, bores, counter bores, recesses and holes. The graduations are read in the same manner as is done in the case of regular micrometers. Larger ranges of depth can be measured by inserting an extension rod through the top of the micrometer. The graduations are in the reversed direction to those of an out side micrometer.

Flange micrometer (Fig 5)

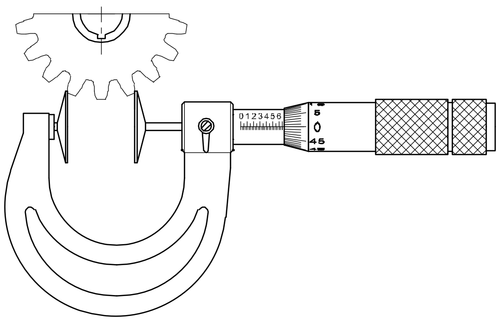
A flange micrometer is similar to a regular micrometer and is equipped with two flanges in the place of the anvil and spindle. This is used to measure chordal thickness of the gear teeth and the thickness of the fins of an engine and the collar thickness of the job.

Fig 4



DEPTH MICROMETER

Fig 5



FLANGE MICROMETER

Ball micrometer

In this form of micrometer, hemispherical balls are fitted at the anvil and spindle. Measurement is similar to that in a regular micrometer. It is used to measure a sphere where the point of contact comes in between.

Stick micrometer

Stick micrometers are designed for the measurement of longer internal lengths.

This comprises of:

- a 150 mm or 300 mm micrometer unit, fitted with a micrometer of 25 mm range and having rounded terminal faces
- a series of extension rods, which together with the micrometer unit, permits a continuous range of measurement up to the maximum length required.

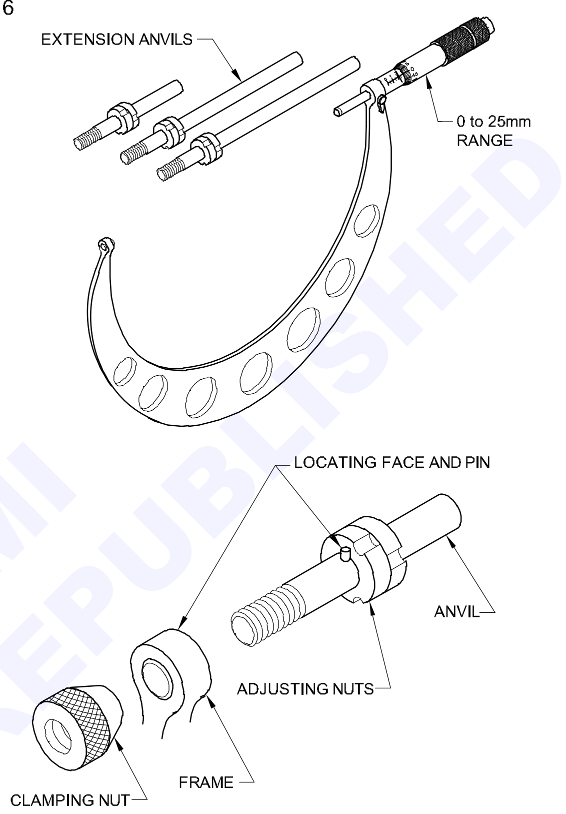
Secured joints are used for joining the end piece, extension rod and the measuring unit. The screw unit generally has threads of 0.5 mm pitch. The extension rod is generally hollow and has a minimum external diameter of 14 mm.

In this type of micrometer, there should be sufficient play between the external and internal threads of the joint to permit the abutment forces of the various parts of the micrometer to butt together solidly.

External micrometer with interchangeable anvils (Fig 6)

It is nothing but an external micrometer. The advantage in this micrometer is the range of the micrometer can be increased by merely changing the different anvils.

Fig 6



A set of replaceable anvils is supplied in a box and the size of the anvil is marked on each anvil. Depending upon the size of the job, the anvil size can be changed, and reading can be taken. Thus it is an economy micrometer, i.e. in one micrometer itself, long ranges can be accommodated. To fix the anvils to the frame, a guide is provided and locked by a nut.

Keyway depth micrometer

It is similar to a depth micrometer except that the frame has 120° inclined butting surfaces to rest on the circumference of a cylindrical job. It is used for measuring depth of keyways on a cylindrical shaft. While measuring the depth of the keyway, first take the measurement on a cylindrical job opposite to the keyway; then take the measurement of the keyway depth, subtract the initial measurement from the final measurement to know the exact depth of the keyway.

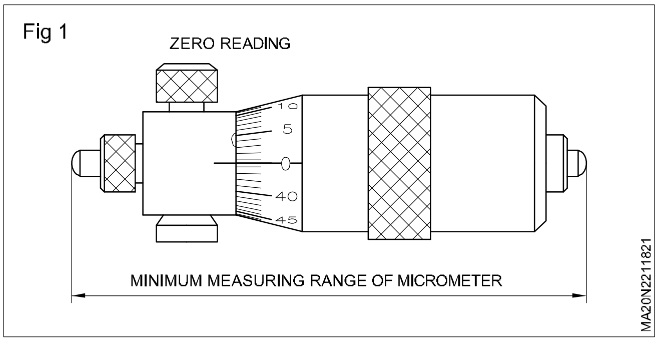
Inside micrometer - metric

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

- **name the parts of an inside micrometer**
- **state how to measure the size of the bore or hole using inside micrometer**
- **state how to measure the size with a spacing collar and extension rods**
- **explain three point internal micrometer.**

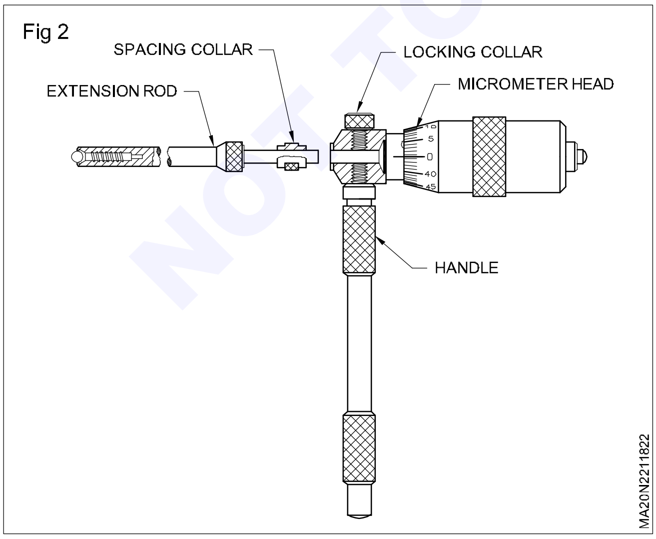
The inside micrometer is similar to an ordinary outside micrometer but without the 'U' frame. (Fig 1)

Fig 1



The measurement is taken over the contact points. As the thimble opens or closes, the contact points get opened or closed. The inside micrometer consists of a sleeve, thimble, anvils, a spacing collar and extension rods. It is also equipped with a handle to measure deep bores. The least count of the instrument is also 0.01 mm. The inside micrometer is equipped with a 12 mm spacing collar and 4 extension rods for measuring holes of ranges 50-75 mm, 75-100 mm, 100-125 mm and 125-150 mm. The sleeve is marked with the main scale and the thimble with the thimble scale. The barrel has a limited adjustment of 13 mm. When the inside micrometer is closed (when the zero of the thimble coincides with the zero of the barrel), it is capable of reading the minimum dimension of 25 mm. In addition to this, it is possible to read up to 38 mm with the thimble opening to the extreme right. In order to read further higher ranges, a standard spacing collar of 12 mm width is to be added. This facilitates the micrometer to read a maximum range of 50 mm. (Fig 2)

Fig 2

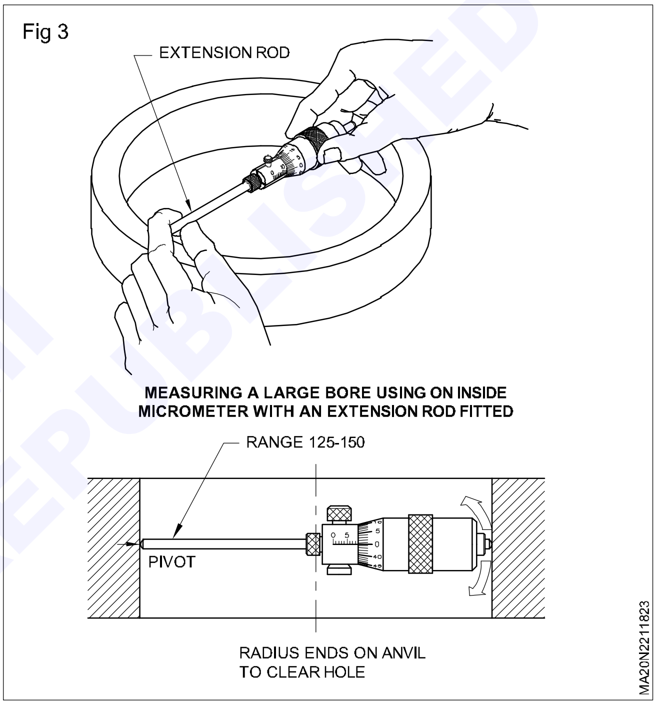


Similarly, each extension rod has to be used without the collar for measuring a minimum range up to 13 mm variation and with the collar for a maximum range of measurements. A clamping screw is also provided to clamp the extension rod firmly.

Measuring the size of a bore or hole

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

Fig 3



Measuring the taper of a bore

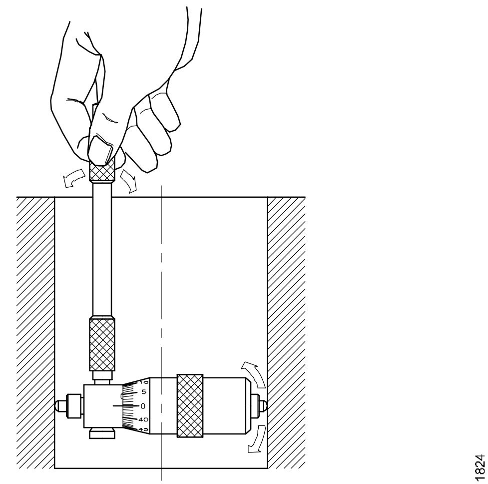
While checking taper of a deep bore, a handle must be used along with the inside micrometer. The Fig 2 shows the inside micrometer with a handle. In order to ascertain the taper, a minimum of two readings has to be taken, i.e. one at the top of the deep bore and the other at the bottom of the bore. If there is no difference in the two readings, we may take it for granted that the bore has no taper. Any variation in the reading shows the bore has a taper. (Fig 4)

Three-point internal micrometer (Fig 5)

A three-point internal micrometer is used for direct measurement of an internal diameter accurately and efficiently. It is also used to measure the diameter of a deep hole, the end of a blind hole, internal recess etc.

The instrument is checked for its zero error with a master ring gauge. (Fig 6)

Fig 4



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

Fig 6

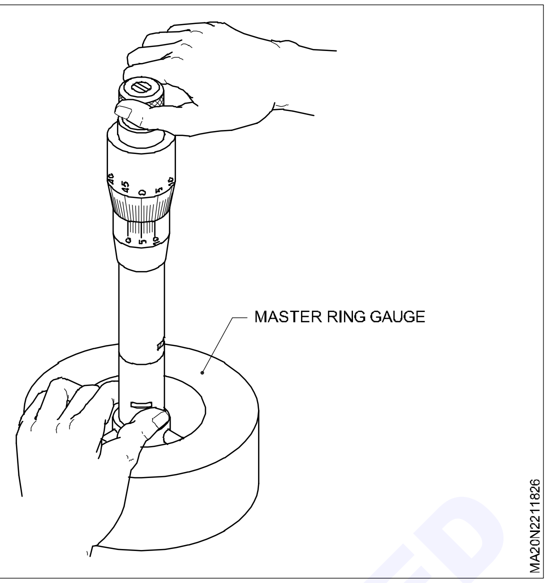
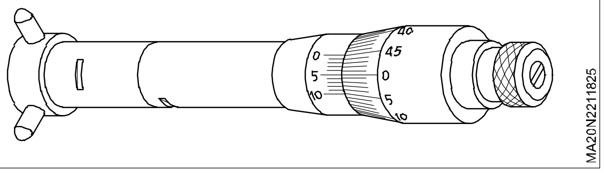


Fig 5



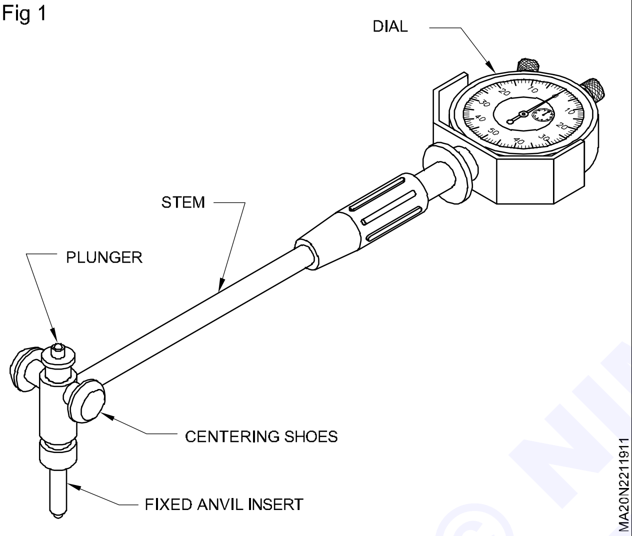
Bore dial gauge and telescopic gauge

Objectives : At the end of this lesson you shall be able to
- **name the parts of bore dial gauge**
- **brief the functions of each parts of a bore dial gauge.**

The dial bore gauge is a precision measuring instrument used for measuring the internal dimensions. The dial bore gauge normally available is of a two-point, self-Centering type.

Parts (Fig 1)

Fig 1



Stem

This holds all the components together and contains the mechanism for transmitting the plunger motion to the dial.

Fixed anvils/inserts

These anvils are interchangeable. The selection of the anvil is made depending upon the diameter of the bore to be measured. For certain types of bore dial gauges extension rings/washers are provided for extending the range of measurement.

Sliding plunger

This actuates the movement of the dial for reading the measurement.

Centering shoes / spherical supports

Certain types of bore dial gauges are provided with two pairs of ground disks, called centering shoes. (Fig 1)

This maintains the alignment of the measuring faces in the center of the bore. (Fig 2) For some types spherical supports which are spring-loaded are provided.

Dial indicator (Fig 3)

This has graduations marked on the dial. The graduations are marked in clockwise and anticlockwise directions.

Fig 2

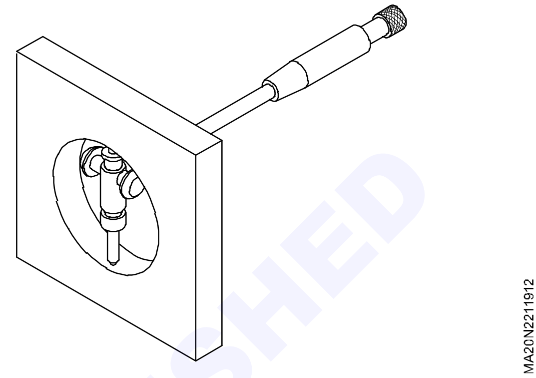
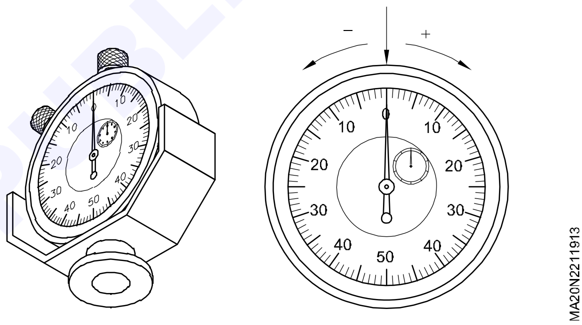


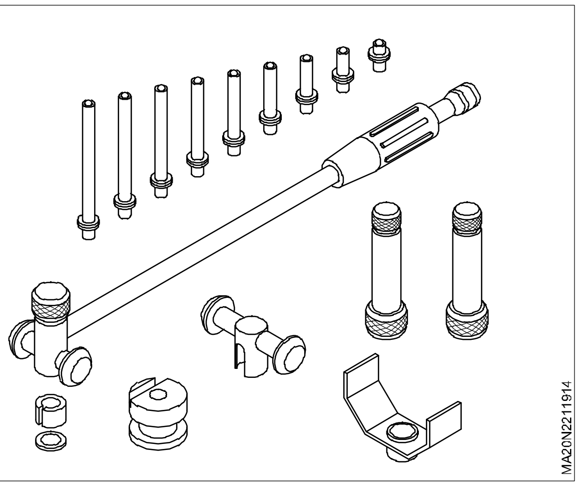
Fig 3



Bore dial gauges are available in various sizes with different measuring ranges.

Interchangeable measuring rods (external rods or combination washers) for measuring different sizes are available. (Fig 4)

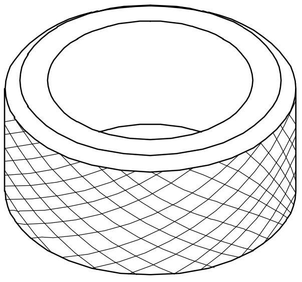
Fig 4



The accuracy of the instrument depends upon the type of the graduations on the dial. The most frequently used instruments have accuracies of 0.001 mm or 0.01 mm.

The dial of the bore dial gauge should be set to zero before taking the measurement. Setting rings are available for zero setting. (Fig 5)

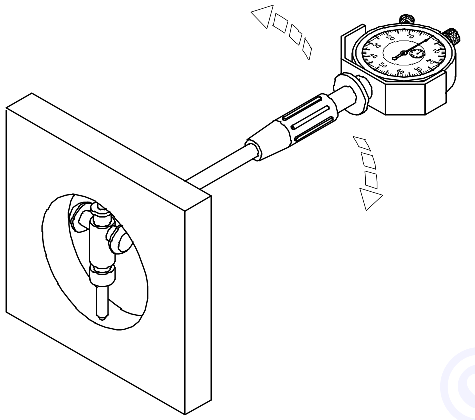
Fig 5



While taking measurements press the spring-loaded end (plunger) as it enters into the setting device or in the bore being measured.

Slightly rock and steady the device for keeping the measuring faces in position. (Fig 6)

Fig 6



Slip gauges fixed in a setting fixture can also be used for zero setting. (Fig 7)

Reading the dial indicator (Fig 8)

When taking the reading first check the measuring range and the subdivisions of the scale. The indicator in the figure has a range of 0.8 mm and is graduated 0-40 on both directions. Thus the value of each division is 0.01 mm.

Fig 7

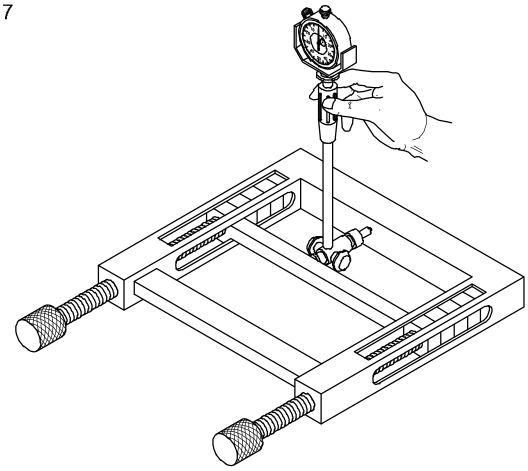
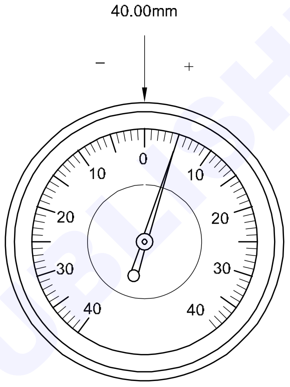


Fig 8



The indicator shows under size in the clockwise direction and over size in the anticlockwise direction.

Example

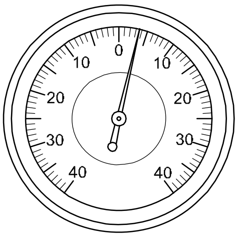
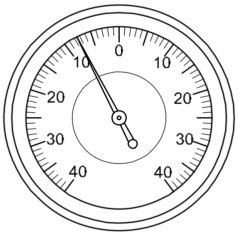
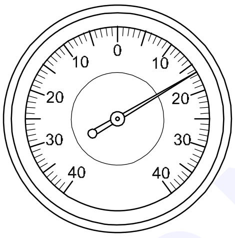
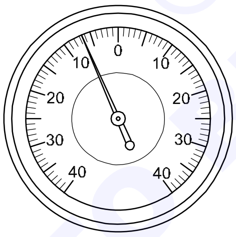
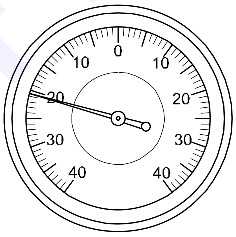
The basic measurement is 40.00 mm. The indicator is between 5 and 6 divisions on the clock wise direction.

The actual value will be between 39.94 to 39.95.

For example in a range of 18 to 35 mm bore dial gauge, the available interchangeable anvils are18,20, 22,24,26,28,30,32,34 with 2 spacer rings of 1mm and 0.5mm.

The different ranges of sizes are 6-10,10-18.5,18-35,35-60 50-100,50-150,100-160,160-250,250-400 in metric system for each range of size a separate box is provided with extension rods and spacers.

Examples to be solved in the classroom

Basic measurement		Value measured	Correct measurement
30.0 mm		29.97 – 29.98 30.02 – 30.03 29.96 – 29.97 30.04 – 30.05	________________
23.0 mm		22.92 – 22.93 23.07 – 23.08 22.94 – 22.95 22.96 – 22.97	________________
47.8 mm		47.86 – 47.87 47.63 – 47.64 47.92 – 47.93 47.96 – 47.97	________________
53.0 mm		52.92 – 52.93 53.06 – 53.07 53.96 – 53.97 53.97 – 53.98	________________
65.0 mm		64.75 – 64.76 64.79 – 64.80 65.20 – 65.21 64.87 – 64.88	________________

Telescopic gauge

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

- **name the parts of a telescopic gauge**
- **state the constructional features of telescopic gauges**
- **state the ranges of telescopic gauges.**

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

Uses

Used for measuring the sizes of hole, slots and recesses.

Construction (Figs 1 & 2)

Telescopic gauges are 'T' shaped. They consist of a pair of telescopic legs or plungers connected to a handle.

Fig 1

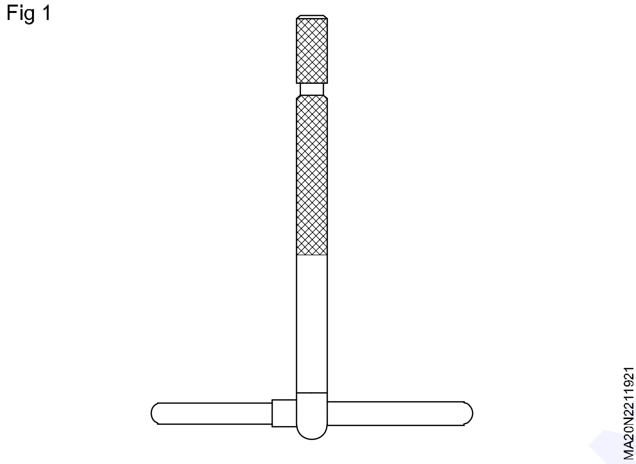
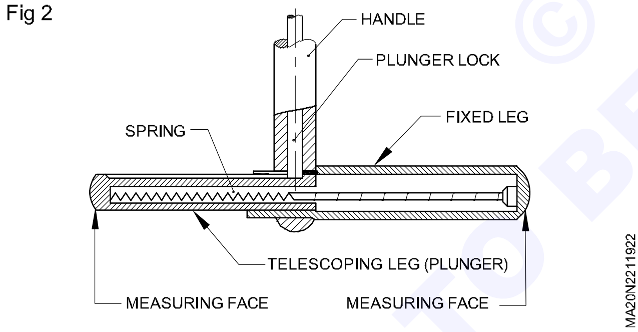
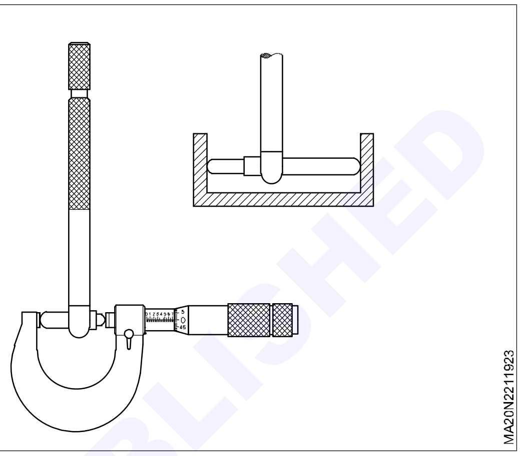


Fig 2



The plungers are spring-loaded to force them apart. After inserting the gauge in a hole or slot, it can be locked in position by turning the knurled handle. It may then be withdrawn from the hole and measured with a micrometer. (Fig 3)

Fig 3



Telescopic gauges are available in a set of 6 Nos, to measure holes from 8 mm to 150 mm. (as per MITUTOYO - Series 155)

No.1	8.0 mm to 12.7 mm
No.2	12.7 mm to 19.0 mm
No.3	19.0 mm to 32.0 mm
No.4	32.0 mm to 54.0 mm
No.5	54.0 mm to 90.0 mm
No.6	90.0 mm to 150.0 mm

Gauges different types and uses

Objectives : At the end of this lesson you shall be able to
- **list the quality of material used in gauges**
- **explain the care and maintenance of gauges.**

Materials for gauges

The material used for manufacturing the gauge must fulfil the following quality, either by virtue of its own properties, or by a heat treatment process.

- Hardness to resist wear.
- Stability to ensure that its size and shape will not change over a period of time.
- Corrosion resistance
- Machinability to enable it to be machined easily into the required shape and to the required degree of accuracy.
- Low coefficient of linear expansion to avoid effect of temperature.
- The parts of the gauge which are to be held in the hand should have low thermal conductivity.

A good quality high carbon steel is usually used for gauge manufacture. Suitable heat treatment can produce a high degree of hardness coupled with stability. High carbon steel is relatively inexpensive, it can be easily machined and brought to a high degree of accuracy and surface finish.

Gauges can also be made from steels, special wear resisting materials, like hard chrome plated surfaces and tungsten carbide, Invar etc. Glass gauges were used during World War. Chromium plating makes the gauge corrosion and wear resistant. Also, the size of worn gauging surface can be increased by this method. The gauge surfaces can also be plated to provide hardness, toughness and stainless properties.

Care of Gauges

While using the gauges proper care should be taken to prolong their maximum useful life. Some suggestions for their use and care are:

- Master, inspection and working gauge should be employed only to the uses for which they are intended, as follows:

Type of gauge	intended use
Master gauge	To check inspection and working gauges
Inspection gauge	To check the finished product
Working gauge	To check the product as it is being manufactured.

Care and maintenance of gauges

- A plain cylindrical gauge should be cleaned properly, and a thin film of light oil should be applied to the gauging surfaces before it is used.
- The work should also be cleaned before checking with the gauges. Then the gauge should be aligned with the hole to be measured and given a forward motion combined with a slight rotation. Go plug gauge will enter the hole if the hole is of correct size otherwise, it will not enter it. The same procedure may be applied while using plane cylindrical gauges.
- Force should not be applied in gauging operation as it tends to harm the gauge, the work or both. Therefore, the snap gauge should not be forced over the work because it will cause the gauge to pass on oversized parts and it may also spoil the frame of the gauge.
- A gauge should be properly cleaned after use and prepared for storage. It should be coated with a rust preventive oil, if it is to be stored for a short time only. However, if it is to be stored for a longer period, it should be dipped in a molten plastic material designed as a protective coating for tools, and gauges.

Angle gauges

Objectives : At the end of this lesson you shall be to
- **state the purpose of angle gauges**
- **state the advantages of using angle gauges**
- **state the method of selecting angle gauges to a required angle**
- **state the precautions to be observed while handling angle gauges.**

Angle gauges are wedge-shaped steel blocks made from hardened non-creep alloy steel to prevent wear and expansional errors.(Fig 1) Its faces are lapped in the same manner as slip gauges, enabling them to be wrung together in combinations.

Applications

– To measure the angle of a workpiece. (Fig 2a)

– To set up a component upon a machine table. (Fig 2b)

Fig 1

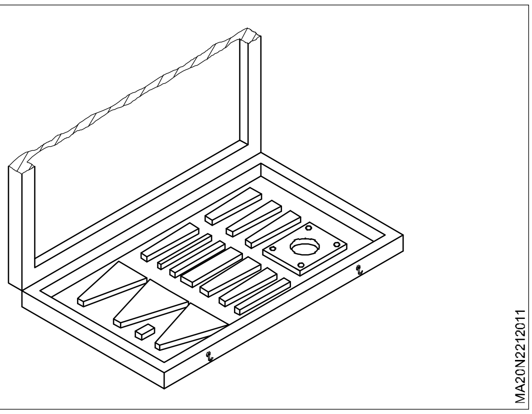
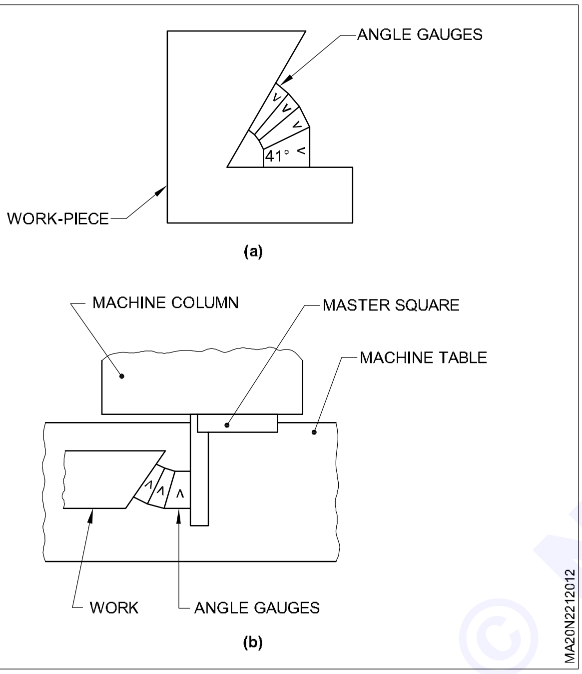


Fig 2



(a)

(b)

– To calibrate adjustable angular measuring instruments.

Advantages

- They can be used either horizontally or vertically.
- They do not require a datum surface or reference plane.
- They can be used to check jobs on which it is not easy to use other precision instruments.
- They are not restricted to angles smaller than 45^0.

As is the case with slip gauges, angle gauges are also available in different sets.

A typical set comprises of twelve pieces divided into three series - degrees, minutes and seconds as follows.

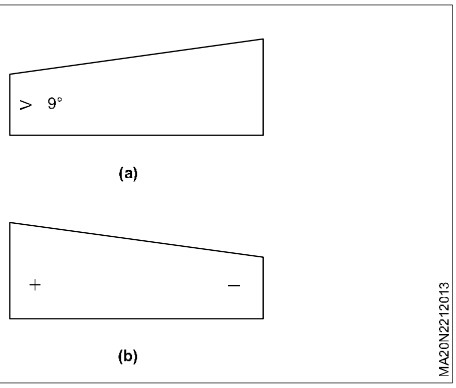
Pieces

Degrees	-	1,3,9,27 and 41	5
Minutes	-	1,3,9 and 27	4
Seconds	-	6,18 and 30	3
			12

There are two sets of angle gauges available in the above typical set, designated as 'A' and 'B'. The standard 'B' contains all the 12 gauges, while standard 'A' contains one additional gauge of 3 seconds which enables combinations to be built up to 3 seconds.

The angle of the gauge and the arrowheads are etched on each gauge to indicate the angle direction and size. (Fig 3a)

Fig 3



(a)

(b)

Some types of angle gauges are etched + and – instead of arrow head to indicate the direction of the angle. (Fig 3b)

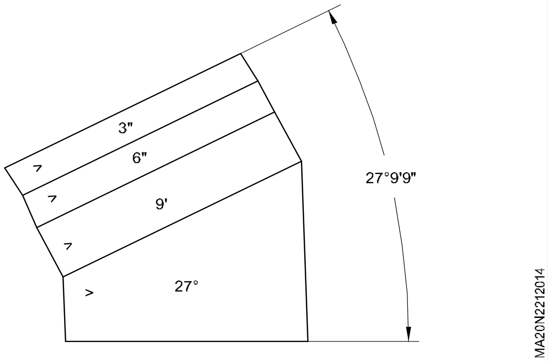
Building up a combination

To build up a size of 27^0 9' 9" (Fig 4)

Gauges required

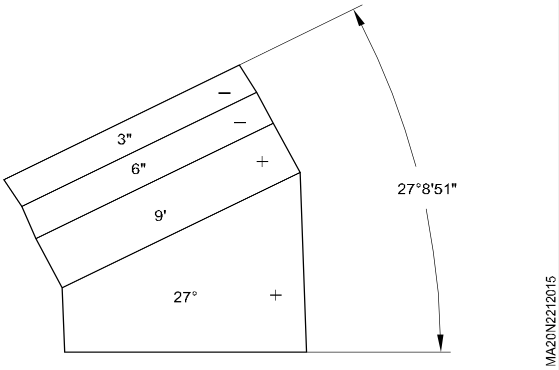
1st series	-	27^0 0' 0"
2nd series	-	0^0 9' 0"
3rd series	-	0^0 0' 6"
Additional block	-	0^0 0' 3"
		27^0 9' 9"

Fig 4



To obtain an angle of 27^0-8'-51" the same gauges may be used, but they must be wrung together as shown in Fig 5.

Fig 5



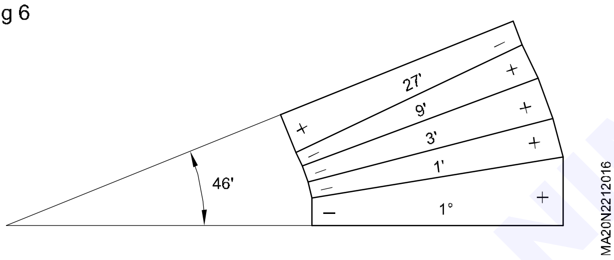
Work out the gauges required for	27^0	8'	51"	
1st series	27^0	0'	0"	
2nd series	0^0	9'	0"	To be added
3rd series	0^0	0'	6"	To be subtracted
4th series	0^0	0'	3"	from the sum of the 1st and 2nd series as these two are positioned in the opposite way.

If the angle includes minutes, and is greater than 40 minutes, increase the angle by 1° and subtract the number of minutes necessary to obtain the required minute.

This is because the total minutes available with the pieces in the series is 40' only.

To obtain 46' the build up will be as shown in Fig 6.

Fig 6



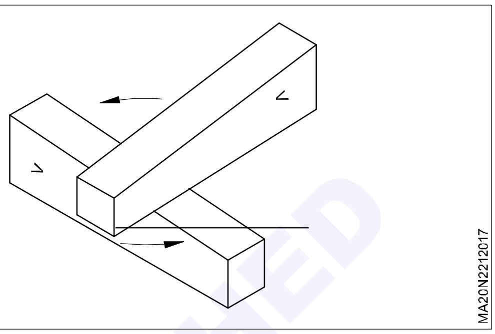
Handling and wringing

Gloves of cotton fabric or chamoise leather must be worn to prevent corrosion.

Before the gauges are wrung together the faces should be wiped clean using soft muslin cloth or chamois leather.

The wringing of the angle gauges should be carried as shown in Fig 7.

Fig 7



After use, clean the gauges thoroughly with a soft cloth and white spirit.

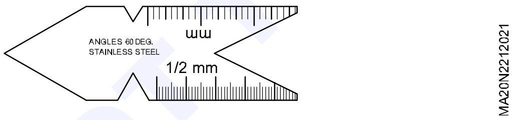
Apply Vaseline lightly and store the gauges in a box.

Centre gauges

Objectives: At the end of this lesson you shall be able to
- **check the angle of single point cutting tool while grinding profile**
- **list the uses of centre gauge.**

Centre gauges are used in lathe work for checking the angles when grinding the profile of single point screw cutting tool bits and centers. (Fig 1)

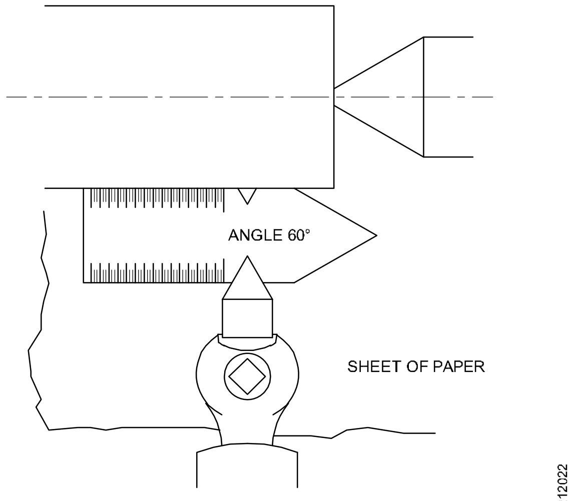
Fig 1



These gauges are most commonly used when hand grinding threading tool bits on a bench grinder, although they may be used with tool of cutter grinders. When the tool bit has been ground to the correct angle, then may be used to set the tool perpendicular to the work piece. (Fig 2)

They can incorporate a range of sites and types on the one gauge. The two most common being metric at 60º and BSW 55º gauges also exist for the acme thread form.

Fig 2



SETTING 60° CUTTING TOOL SQUARE TO THE WORK USING A CENTRE GAUGE

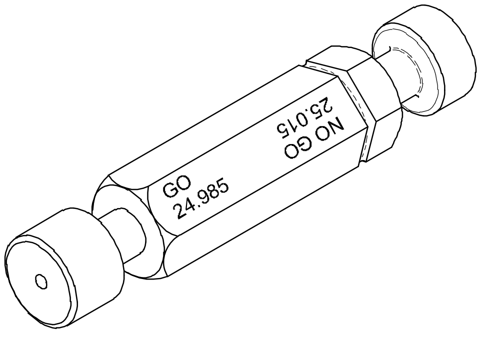
Limit gauges

Objectives: At the end of this lesson you shall be able to
- **state the features of the 'Go' and 'No-Go' limit gauges**
- **Identify the common types of limit gauges and state their uses.**

When a number of components have to be checked, it is not necessary to measure the size exactly but only to check whether the components' size lies within the limits. The most economical method of checking a component is with a limit gauge. These gauges are used in inspection because they provide a quick means of checking a specific dimension.

'Go' and 'No-Go' limit gauges (Fig 1)

Fig 1



The dimensions of the 'Go' and 'No-Go' ends of gauges are determined from the limits stated on the dimension to be gauged.

In the 'Go' and 'No-Go' method of gauging, the 'Go' end of the gauge must go into the feature being checked and the "No-Go" end must not go into the same feature.

The dimension of the 'Go' end is equal to the minimum permissible dimension and that of the 'No - Go' end is equal to the maximum permissible dimension of the component being checked.

Essential features

These gauges must be easy to handle and accurately finished. They are generally finished to one-tenth the tolerance it is designed to control.

For example, if the tolerance is to be maintained to 0.02 mm, then the gauge must be finished to one-tenth the tolerance it is designed to control.

For example, if the tolerance is to be maintained to 0.02 mm, then the gauge must be finished to within 0.002 mm of the required size.

They must be resistant to wear, corrosion and expansion due to temperature.

Their production cost must be low.

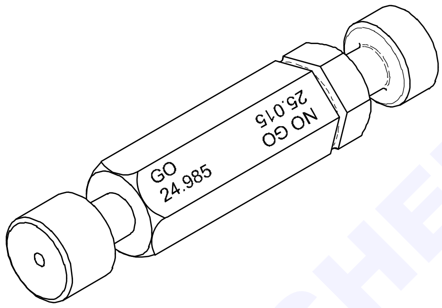
The 'Go' end is made longer than the 'No-Go' end for easy identification. Sometimes a groove is cut on the handle near the 'No - Go' end to distinguish it from the 'Go' end.

Types

Cylindrical plug gauges

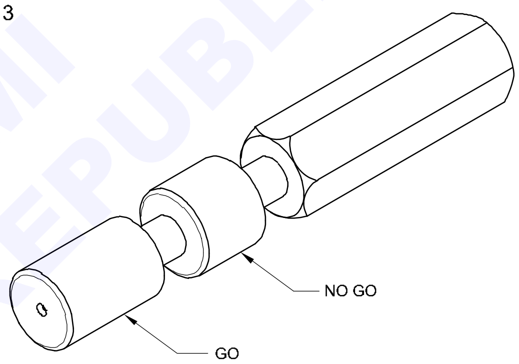
Double ended plug gauge (Fig 2)

Fig 2



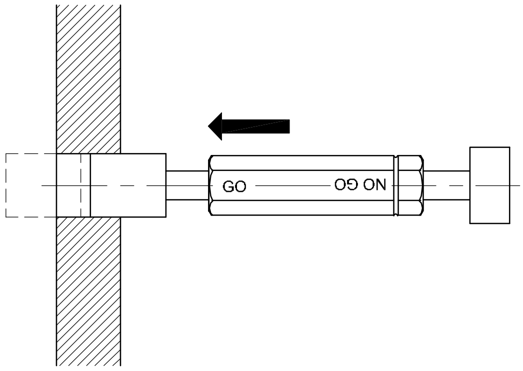
Progressive plug gauge (Fig 3)

Fig 3



Plain cylindrical plug gauges are used for checking the inside diameter of the straight hole. the 'Go' gauge checks the lower limit of the hole and the 'No-Go" gauge checks the upper limit. The plugs are ground and lapped. (Fig 4)

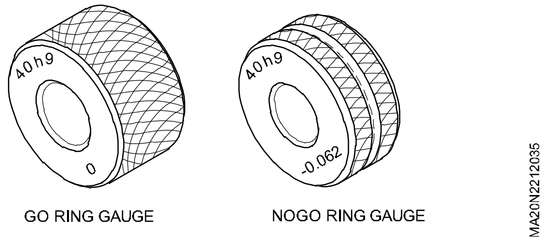
Fig 4



Plain ring gauges (Fig 5)

They are used to check the outside diameter of pieces. Separate gauges are used for checking 'Go' and 'No-Go' sizes. The 'No-Go' gauge is identified by an annular groove on the knurled surface.

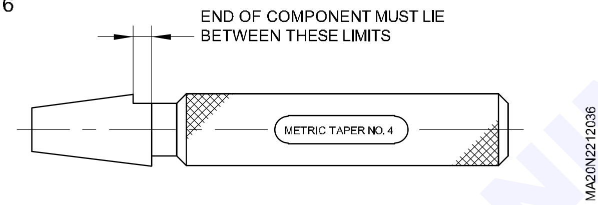
Fig 5



Taper plug gauges (Fig 6)

These are made with standard or special tapers and are used to check the size of the hole and the accuracy of the taper. The gauge must slide into the hole for a prescribed depth and fit perfectly. An incorrect taper is evidenced by a wobble between the plug gauge and the hole.

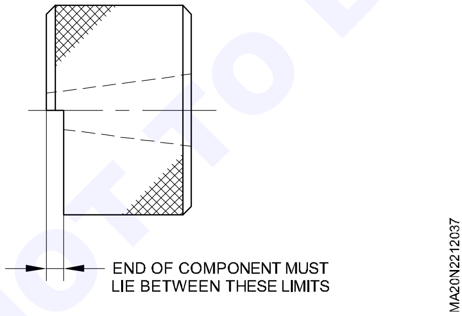
Fig 6



Taper ring gauges (Fig 7)

They are used to check both the accuracy and the outside diameter of an external taper. Taper ring gauges often have scribed lines or a step ground on the small end to indicate the 'Go' and 'No-Go' dimensions.

Fig 7



Thread plug gauges (Figs 8 & 9)

Internal threads are checked with thread plug gauges of 'Go' and 'No-Go' variety and employ the same principle as cylindrical plug gauges.

Fig 8

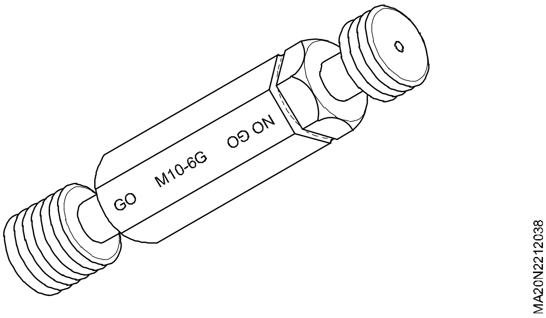
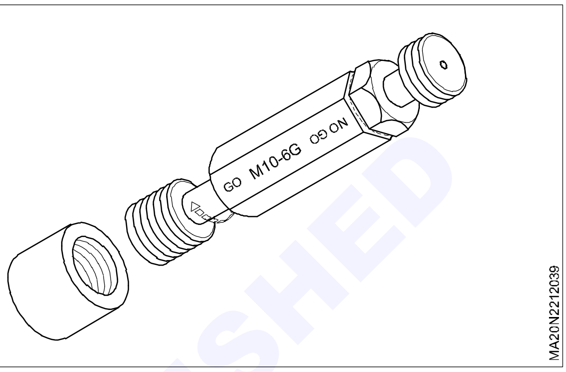


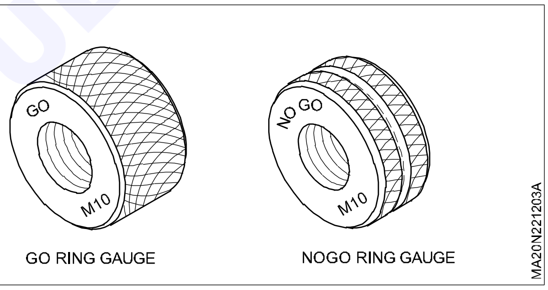
Fig 9



Thread ring gauges (Fig 10)

These gauges are used to check the accuracy of external threads and have a threaded hole in the centre.

Fig 10



Snap gauges (Figs 11 & 12)

Snap gauges are a quick means of checking diameters and thickness within certain limits by comparing the part size, to the dimension of the snap gauge.

Snap gauges are generally C-shaped and are adjustable to the maximum and minimum limits of the part being checked. When in use, the work should slide into the 'Go' gauge but not into the 'No-Go' gauge.

Fig 11

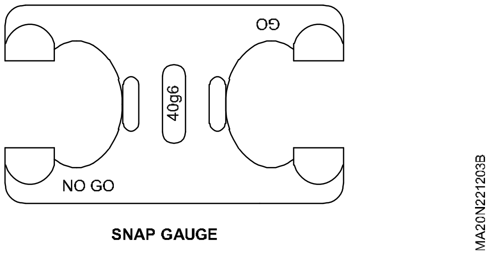
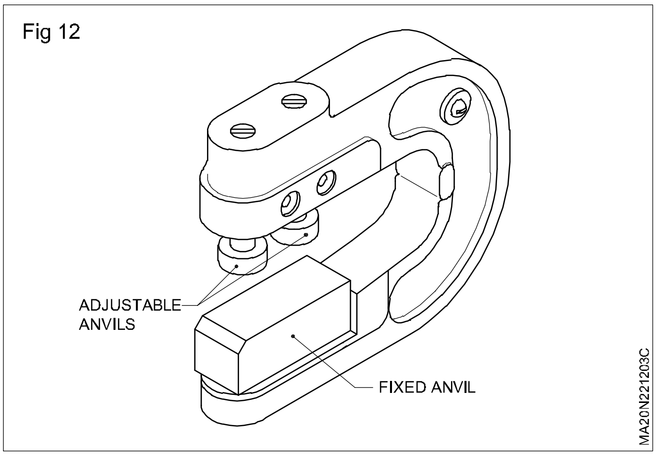


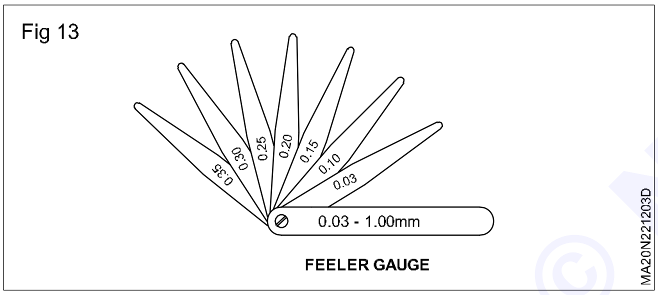
Fig 12



Feeler gauge (Fig 13)

Feeler gauge is used to measure/check the clearance between the two mating parts. For example, it can be used in gauging of the clearance between the piston and cylinder and also for adjusting the spark gap between the distributor points of an automobile. The feeler gauge set consists of narrow strips of sheet steel of different thickness assembled (hinged) together in a holder. (Fig 13)

Fig 13



Their working entirely depends upon the sense of feel. In using the strips (blades), it is essential that the strips should neither be forced between the surfaces nor slide freely. The Correct strip or a combination of strips will give a characteristic ‘gauge fit’ type of feel.

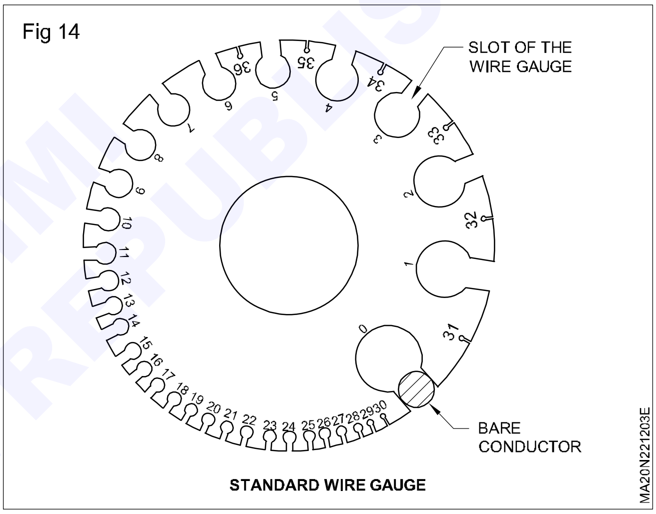
A set of feeler gauge generally consists of series of blades of thickness varying from 0.03 to 1mm. The blades are made of heat treated bright polished tool steel. The width of the blade in 12mm at the heal and tapered for outer part of their length so that the width at the tip is approximately 6mm. The holder protects the blade when not is used. The nominal thickness of the blade is marked on it legibly.

IS: 3179 recommends seven sets of feeler gauges with thickness from 0.03. to 1.00mm. Each set is devised so as to permit maximum utility with minimum number of blades. Table below gives the thickness and number of blades in each set as recommend by Indian standard IS: 3179.

Wire Gauge (Fig 14)

Wire gauge (Fig 14) are used for finding diameters of wires by inserting the wire in the notches provided and finding out which it fits. The diameter and the number marked on the disc are read off from the gauge. The wire gauge has the range from 0.1mm to 10mm.

Fig 14



Comparator gauges

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

- **state the purpose of comparator gauges**
- **state the principle of working of comparator gauges**
- **state the essential features of a good comparator gauge**
- **differentiate between gauges and measuring instruments.**

Purpose of a comparator gauge

The purpose of all comparator gauges is to indicate the difference in the size between the standard (slip gauge or ring gauge) and the work being measured by means of some form of pointer on a scale at a magnification which is sufficient to read to the accuracy required. Almost every possible principle known to the Science of Physics for providing magnification has been used for the construction of these comparator gauges.

Essential features of a good comparator gauge

– Should be compact.

– Maximum rigidity.

– Maximum compensation for temperature effects.

– No backlash in the movement of the plunger and recording mechanism.

– Straight line characteristics of the scale readings.

- Most suitable measuring pressure which remains uniform throughout the scale.
- Indicator should be consistent in its return to zero.
- Method of indication should be clear and the pointer 'dead beat' (ie. free from oscillations).
- Should be able to withstand reasonable wrong usage.
- Should have a wide range of operations.

Principles of working

The following principles are employed in the commonly used comparator gauges.

- Mechanical
- Electronics
- Pneumatic
- Optical

Difference between measuring instrument and gauges

Measuring instrument	Gauges
It is used to measure actual size of component.	It is used to check the dimensions within the specified limits.
These have an adjustable facility.	In most cases these have no adjustable facility.
Variable dimension instrument.	Fixed dimension instrument.
They have a graduations.	They have no graduation.
These are used to measure processing and finishing of the component.	Only for finished component.
Skill required.	Un skilled person may also used.
These are used in Tool room work.	Mainly used in mass production.

Gears - Types and uses

Objectives : At the end of this lesson you shall be able to
- **name the different types of gears**
- **state the broad area of application of the different types of gears.**

A gear is a machine element. It is used to transmit power and motion between rotating parts.

Positive transmission of power is accomplished by providing projections called teeth on the circumference of the gear.

There will be no slippage in gear drives as it is found in friction and belt drives.

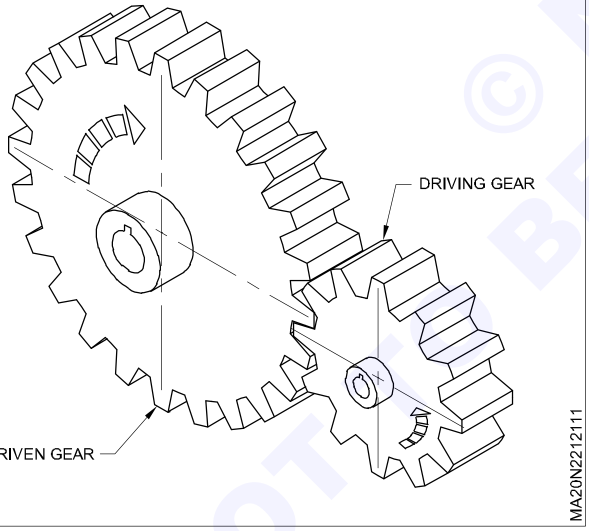
General classification of gears

Depending upon the axes, the shape of the solid on which the teeth are developed, the curvature of the tooth-trace and any other special features, gears are categorized into the following types.

Spur gear (Fig 1)

Such gears as are having their teeth element parallel to the rotating shafts are known as spur gears. (Fig 1) These gears are most commonly used to transmit power and motion through parallel shafts.

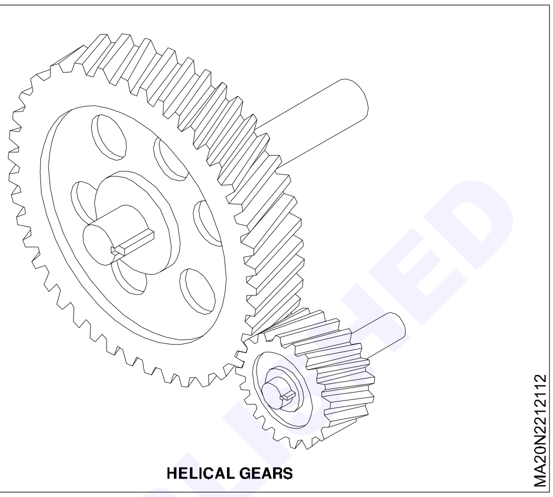
Fig 1



Helical gear (Fig 2)

If the elements of the teeth are twisted or helical as shown in Fig 2, they are known as helical gears. These gears may be used for connecting shafts that are at an angle in the same plane or in different planes. Helical gears are smooth acting because there will always be more than one tooth in contact. The only disadvantage of using this gear is that there will be an axial thrust tending the shaft to move axially.

Fig 2

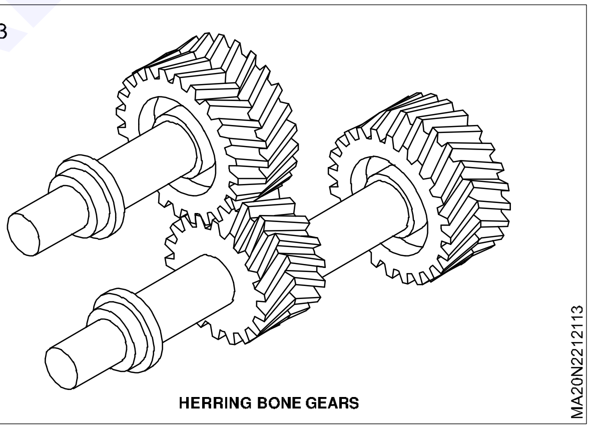


HELICAL GEARS

Herringbone gear (Fig 3)

A herringbone gear is equivalent to two helical gears, one having a right hand and the other a left hand helix as shown in Fig 3. This type of gears does not produce axial thrust, and being strong are used in heavy duty machines like steel roll mills.

Fig 3



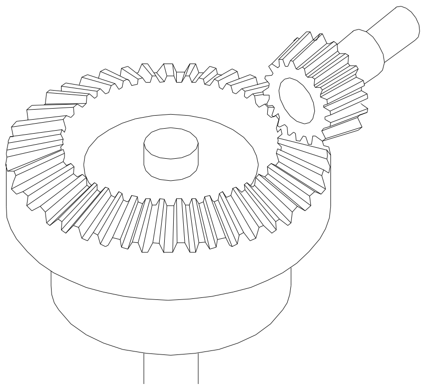
HERRING BONE GEARS

Bevel gear (Fig 4)

Such a gear is similar in appearance to the frustum of a cone having all the elements of the teeth intersecting at a point. (Fig 4)

When two shafts are in the same plane but at an angle with one another, bevel gears are used.

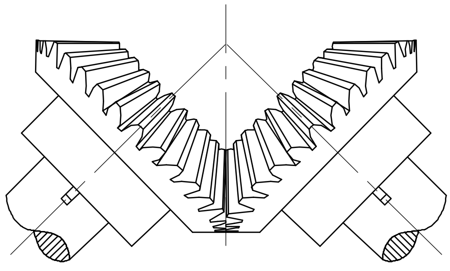
Fig 4



BEVEL GEAR

When the shafts are at right angles and two bevel gears are of the same size, they are known as ;Mitre gears' (Fig 5)

Fig 5

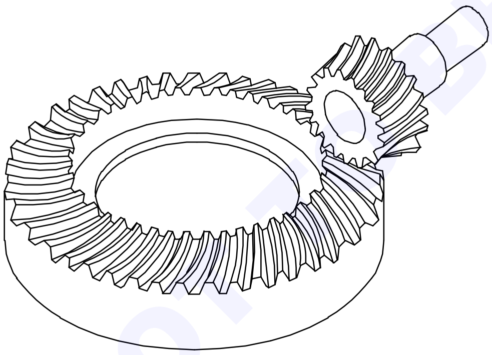


MITRE GEARING

Hypoid gears (Fig 6)

It is the modification of a bevel gear where the shafts are at right angle but they do not intersect as do the shafts for bevel gears. The other type of bevel gears is the spiral bevel gears is the spiral bevel gear which has helical teeth used in automobile transmissions.

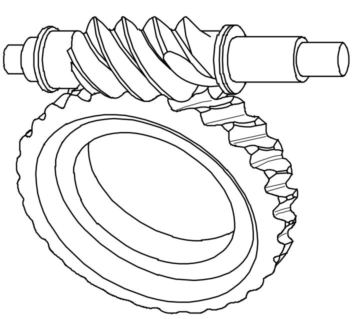
Fig 6



Worm gear (Fig 7)

Worm gears are used where a large speed reduction is desired. The small driving gear is called a worm and the driven gear is called a wheel as shown in Fig 7.Shafts for such gears are at right angles but not in the same plane. In worm and worm wheel gearing, the worm will always be the driver. On machne tools these gears are used in the feed mechanism.

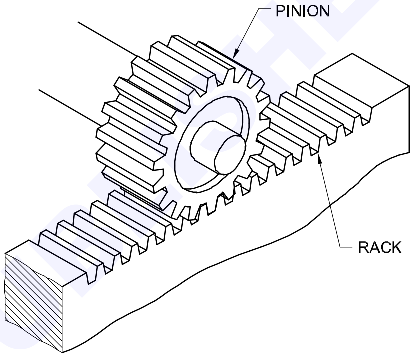
Fig 7



Rack gears (Fig 8)

Rack gears are straight and have no curvature. They represent a gear of infinite radius and are used in feeding mechanisms and for reciprocating drives. They may have straight or helical teeth.

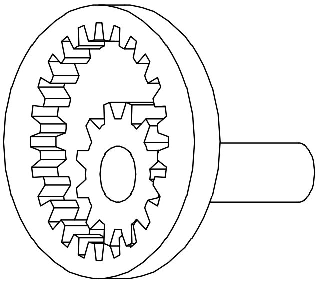
Fig 8



Annular gear (Fig 9)

A gear with internal teeth is known as annular gear. Annular gears are used in automobiles.

Fig 9



Spur gear calculations, curves and their uses

Objective : At the end of this lesson you shall be able to
- **state the basic elements of a spur gear.**

Elements of Spur Gear

Spur gear elements

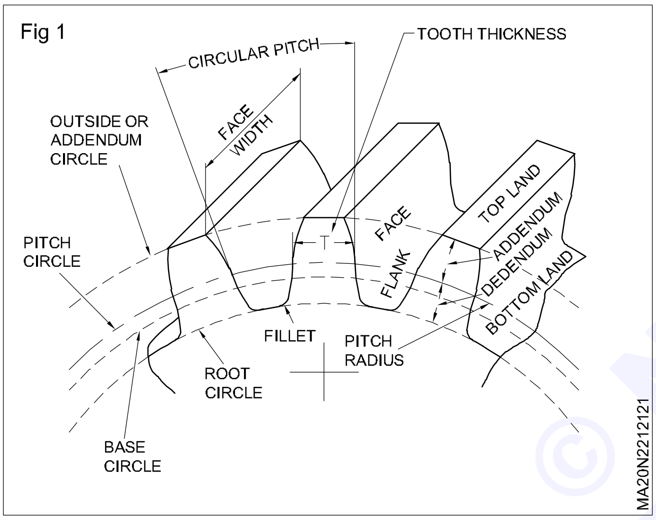
A spur gear is the simplest form of gears. The tooth proportions of the spur gears are expressed in terms of modules.

Module

It is defined as the ratio of the pitch diameter to the number of teeth of a gear. The module is denoted by the letter 'm' and is expressed in millimeters. The module is one of the major determining parameters of a gear.

Basic Elements (Fig 1)

Fig 1



Pitch circle

It is the imaginary circle on which two mating gears seems to be rolling.

The gear calculations are based on this circle.

Circular pitch: 'CP or 'P'

It is the distance from the point of one tooth to the corresponding point of the adjacent tooth measured on pitch circle.

Pitch circle diameter (PCD)

The diameter is called pitch circle diameter (PCD) or simply pitch diameter.

It is denoted by the letter 'd' with proper subscripts eg. d1 for pinion and d2 for the matting gear.

Addendum circle

Addendum circle or outside circle bounds the outer edges of the teeth of a gear and its diameter is denoted by 'da'.

Root circle

The root circle or addendum circle bounds the bottom of the teeth and its diameter is denoted by 'df'.

Base circle ('db')

This is the circle from which the in volute tooth profile is developed. Its diameter is denoted by db.

Addendum (ha) (Fig 2)

It is the radial distance between the pitch circle and the addendum circle and is denoted by ha.

Dedendum (hf) (Fig 2)

It is the radial distance between the pitch circle and the root circle, and is denoted by hf.

Land (Fig 2)

The land and the bottom land are surfaces at the top of the tooth and the bottom of the tooth space respectively.

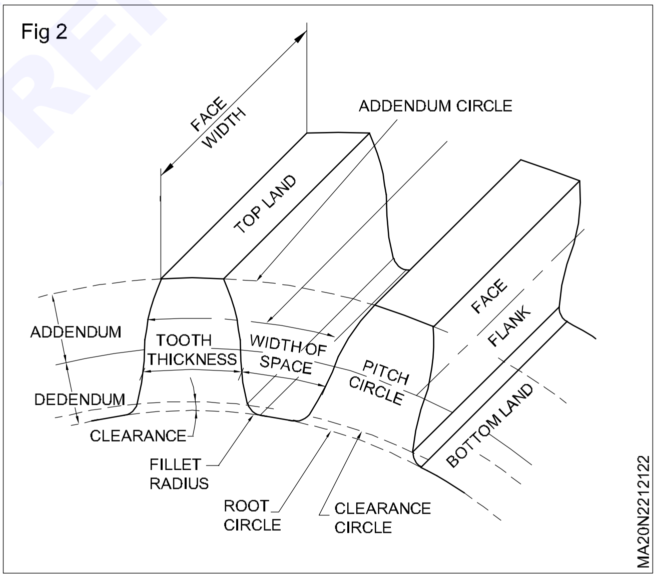
Working depth (Fig 2)

This is the distance of engagement of two mating teeth and is equal to the sum of addendums of the mating teeth of the two gears in the case of standard systems and is expressed as '2ha'.

Whole depth (Fig 2)

This is the height of a tooth and is equal to the addendum plus addendum and is expressed as (ha+hf).

Fig 2



Clearance (Fig 3)

This is the radial distance between the top land of a tooth and the bottom land of the mating tooth.

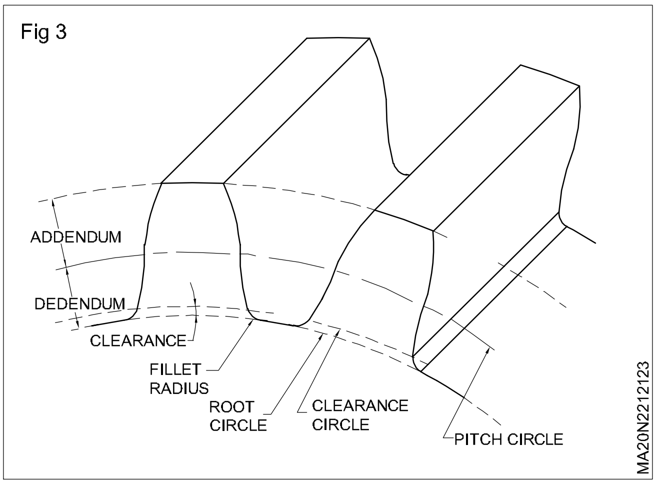
Face width (Fig 2)

This the width of the gear and it is the distance from one end of a tooth to the other end.

Face of a tooth (Fig 2)

This is the surface of the tooth between the pitch circle and the outside circle.

Fig 3



Flank of tooth (Fig 2)

This is the surface of the tooth between the pitch circle and the root circle.

Chordal tooth thickness (Fig 4)

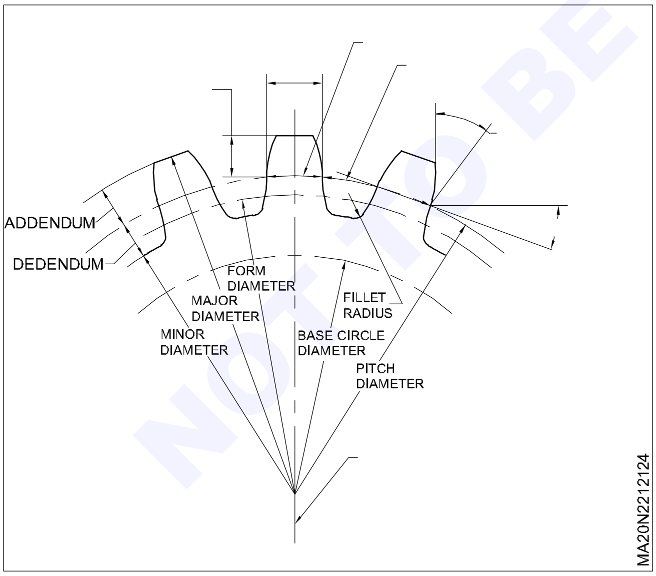
This is the chord referred to above. chordal addendum (or) chordal height, as it is sometimes called) as well as the chordal tooth thickness are important in the checking of gears.

Chordal addendum (Fig 4)

This is the height bound by the top of the tooth and the chordal corresponding to the arc of the pitch circle representing the circular tooth thickness.

Diametric pitch (DP) (Fig 4)

This is a term used in gear technology in the F.P.S. system. It is defined as the ratio of the number of teeth to the pitch diameter in inches.



It is usually denoted as 'DP'. It is equal to the number of gear teeth per inch of the pitch diameter. The unit of DP is the reverse of inch.

The following relation exists between DP and module.

Module (mm)

25.4

Line of action

This is the line along which the point of contact of the two mating teeth profiles moves.

Pitch line

It is the line of contact of two pitch surfaces.

Pressure angle

The angle which the common normal to the two teeth at the point of contact makes with the common tangent to the two pitch circles at the pitch point is known as the pressure angle.

Spur gear tooth proportions

Symbols

Pd = Pitch diameter
da = Outside diameter
df = Root diameter
P = Circular pitch
Z = Number of teeth
m = Module
h = Height of tooth (ha + hf)
ha = Tooth addendum
hf = Tooth addendum

Tooth proportions for 14 1/2° pressure angle

Pitch diameter (Pd) = Zm

Outside diameter (da) = m(Z + 2)

Root diameter (df) $= m\left(Z-\frac{7}{3}\right)$

Circular pitch (p) $= \frac{\pi}{Z} Xpd$

Module (m) $= \frac{pd}{Z}$

Height of tooth (h) = ha + hf = m(1 + 7/6)

Tooth addendum (ha)= m

Tooth addendum (hf) = $\frac{7}{6}$ m

Spur gear calculations

Objective : At the end of this lesson you shall be able to
- **calculate the spur gear tooth proportions with the given data.**

Example 1

Determine P, ha, hf, h, Pd, da and df for a gear wheel with a module of 3 mm 20 teeth.

Given m = 3 mm

Z = 20.

Pd (Pitch diameter) = Zm

= 20 x 3 = 60mm.

da (Outside diameter) = m (Z+2)

= 3 (20 + 2)

= 3 x 22 = 66mm.

(or)

da = Pd + 2 addendum

da = 60 + 2 x ha

da = 60 + 2 x 3 (because ha = m)

60 + 6 = 66mm.

$$\text{df (Root diameter)} = m\left(Z - \frac{7}{3}\right)$$

$$df = 3\left(20 - \frac{7}{3}\right) = 3\frac{60-7}{3} = 3\frac{53}{3}$$

$$= 3 x \frac{53}{3} = 53\,mm$$

ha (Tooth addendum) ha = m, ha = 3mm.

$$\text{hf (Tooth dedendum)} = \frac{7}{6}m$$

$$\frac{7}{6} x 3 = \frac{7}{2} = 3.5\,mm$$

h = (height of tooth) = (ha + hf) = 3 + 3.5 = 6.5mm.

$$P = (\text{Circular pitch}) = \frac{\pi\, x pd}{Z} = \frac{22}{7} x \frac{66}{7}$$

= 9.43mm.

Ans. P = 9.43mm. ha = 3mm. hf = 3.5mm.

h = 6.5mm. Pd = 60mm. da = 66mm.

df = 53mm.

Example 2

Standard spur gear Fig 1 shows the meshing of standard spur gears. The meshing of standard spur gears means the reference circles of two gears contact and roll with each other. The calculation formulas are in Table 1.

The meshing of standard spur gears. (Fig 1)

α = 20°, z1 = 12, z2 = 24)

Fig 1

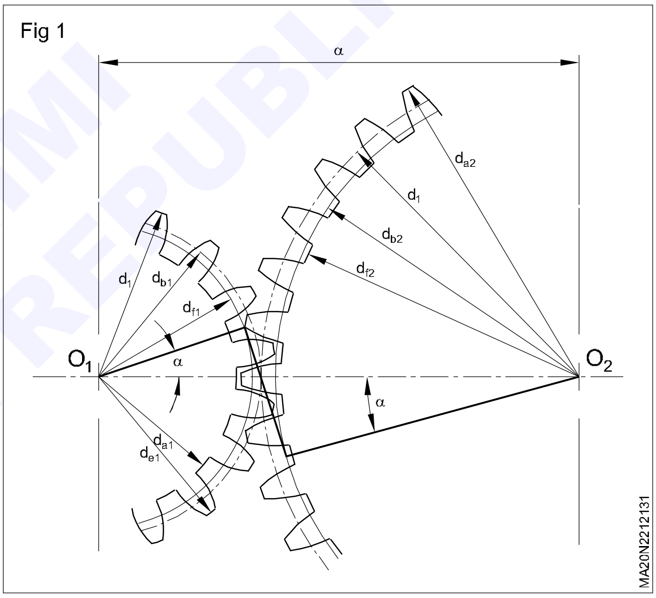


Table 1 - Calculations for standard spur gears

Sl.No.	Item	Symbol	Formula	Example	
				Pinion (1)	Gear (2)
1	Module	m	Set value	3	
2	Reference Pressure Angle	α		20 deg	
3	Number of Teeth	z		12	24
4	Center Distance	a	(z1+z2) m/2	54.000	
5	Reference Diameter	d	zm	36.000	72.000
6	Base Diameter	db	d cos α	33.829	67.658
7	Addendum	ha	1.00m	3.000	3.000
8	Tooth Depth	h	2.25m	6.750	6.750
9	Tip Diameter	da	d + 2m	42.000	78.000
10	Root Diameter	df	d - 2.5m	28.500	64.500

Gears tooth profile

Objectives : At the end of this lesson you shall be able to
- **state the involute gear tooth profile**
- **brief the cycloidal gear tooth profile.**

Gear tooth profile

A gear tooth profiles is the shape of the teeth on a gear. The profile of a gear tooth determines how the gear will mesh with other gears, and it is an important factor in the performance and defficiency of a gear system. There are several different types of gear tooth profiles, including spur gears, bevel gears, and more. The choice of a particular tooth profile will depend on the specific requirements of the application and the desired properties of the gear system.

Forms of gear profiles

Any two curves of any shape that fulfil the law of gearing can be used as the profiles for gear. In other words, an arbitrary shape of any of the mating teeth can be taken and the shape of the other can be determined using the law of gearing. Such teeth are called conjugate teeth.

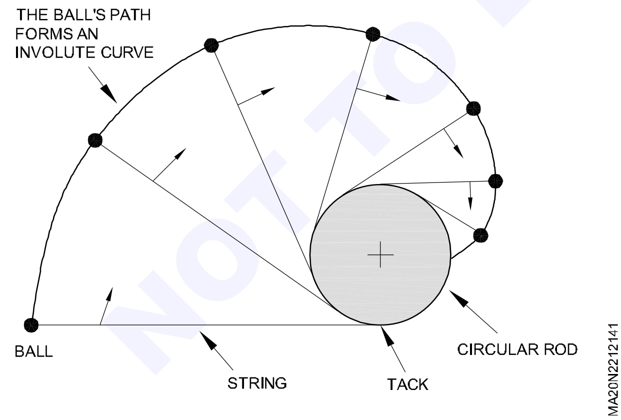
The conjugate teeth are not in common use because of the difficulty in manufacturing, and the cost of production. Therefore, in actual practice, the following are the two types of gear profiles commonly used

- Involute gear profile, and
- Cycloidal gear profile

Involute gear profile

An involute of a circle is a plane curve generated by a point on a tangent, which rolls on the circle without slipping or by a point on a taut string which is unwrapped from a reel as shown in Fig 1.

Fig 1

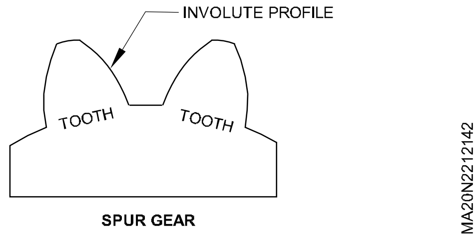


Involute gar tooth profile is shown in Fig 2.

Cycloidal gear profile

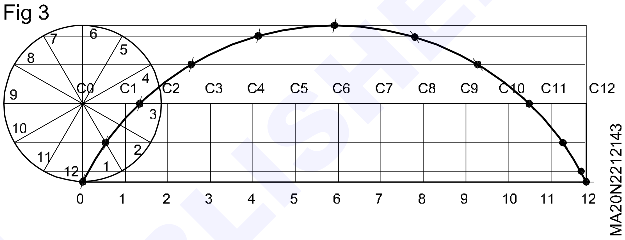
It is important to learn the mathematical point of view of the term "Cycloid" to understand the design and construction of a cycloidal profile of a gear.

Fig 2



Now, mathematically, "A cycloid is the curve traced by a point on the circumfernce of a circle which rolls without slipping on a fixed straight line. (Fig 3)

Fig 3



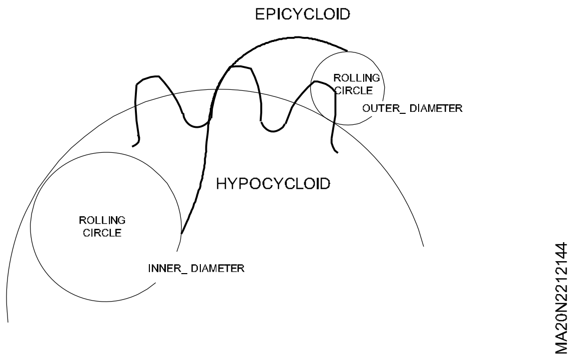
When a circle rolls without slipping on the inside of a fixed circle, then the curve traced by a point on the circumference of a circle is called a hypocycloid. In this case, straight foot flanks that extend radially outwards are obtained.

These are also commonly known as "clock toothing", since it was often found in clockworks in the past (nowadays, however, circular arc toothing is mostly used).

When a circle rolls without slipping on the outside of a fixed circle, the curve traced by a point on the circumference of a circle is known as epicycloid.

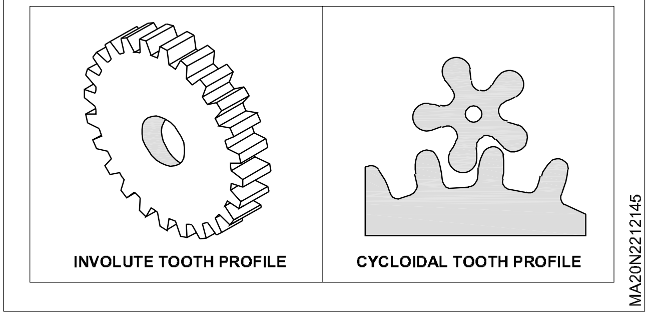
Cycloidal gear tooth profile is shown in Fig 4

Fig 4



Comparison between involute gear tooth and cycloidal gear tooth profiles provides a realistic assessment of the positive and negative points of both involute and cycloidal tooth profiles. (Fig 5)

Fig 5



Comparison between involute gear tooth and cycloidal gear tooth profiles

Table 1

Sl.No	Involute gear	Cycloidal gear
1	Pressure angle remains constant throughout the operation this leads to smooth-running operation of the gears	Pressure angle keeps on changing varies from a maximum at the beginning, reduced to zero at the pitch point and again increases to maximum this result leads to less smooth-running operation of the gears.
2	It involves a single curve for the teeth resulting in simplicity of manufacturing	It involves a double curve for the teeth resulting in the complication in manufacturing
3	Teeth have radial flanks thus are weaker	Teeth have spreading flanks thus are stronger.
4	It is simple to manufacture due to the convex surface and thus are cheaper.	It is difficult to manufacture due to the requirement of hypocycloid and epicycloid and thus are costlier.
5	The velocity ratio is not affected by a little variation in the centre distance	To transmit a constant velocity ratio, an exact centre distance is needed.
6	Interference takes place	There is no interference
7	Due to two convex surfaces are in contact, more wear and tear takes place	Due to concave surfaces are in contact, less wear and tear takes place.
8	Line of action is straight	Line of action is curve
9	Suitable for motion as well as power transmission	Suitable for motion transmission only.

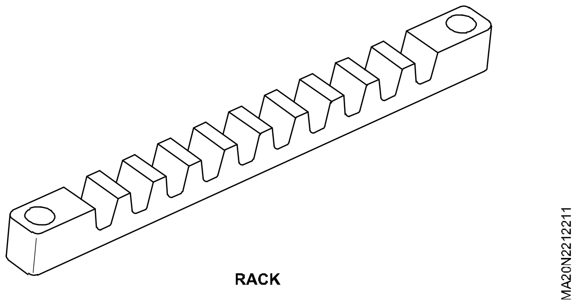
Rack - Types, uses and calculations

Objectives : At the end of this lesson you shall be able to
- **define rack**
- **explain the purpose of the rack**
- **list the types of rack**
- **describe the parts and functions of rack cutting attachments.**

Rack

A rack or a gear rack is having teeth cut on a flat surface. (Fig 1). A rectangular stock is commonly used. At times square or round stock may also be used for cutting a rack.

Fig 1

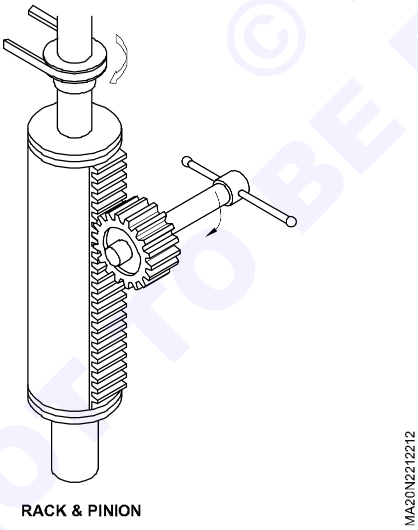


RACK

Purpose

A gear rack when meshed with a gear, is used for converting rotary motion into reciprocating motion. Eg: drilling machine spindle - quill. (Fig 2)

Fig 2



RACK & PINION

The definition of the terms like, module, addendum, dedendum, whole depth, clearance and pressure angle are same as spur gears.

Types

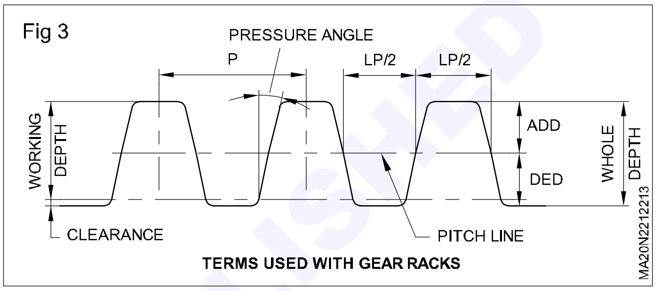
Racks may be of two types.

- Straight toothed rack
- Helical toothed rack

Terms used in connection with racks (Fig 3)

Pitch line, addendum, dedendum, whole depth clearance and pressure angle.

Fig 3



TERMS USED WITH GEAR RACKS

A rack, in conjunction with a gear (pinion), is used to convert rotary motion into longitudinal motion.

Racks are found on lathes, drill presses and many other machines in a workshop.

A rack may be considered as a spur gear which has been straightened out so that the teeth are all in one plane. The circumference of the pitch circle of this gear would now become a straight line which would just touch the pitch circle of a gear, meshing with the rack.

The pitch of a rack is measured in linear pitch, which is obtained by the formula.

Pitch = p m where 'm' is the module.

The method used to cut a rack will depend generally on the length of the rack.

For short lengths the work piece may be clamped in a vice such that the tooth will be parallel to the side of the cutter and the teeth may be cut accurately by moving the cross -slide of the machine.

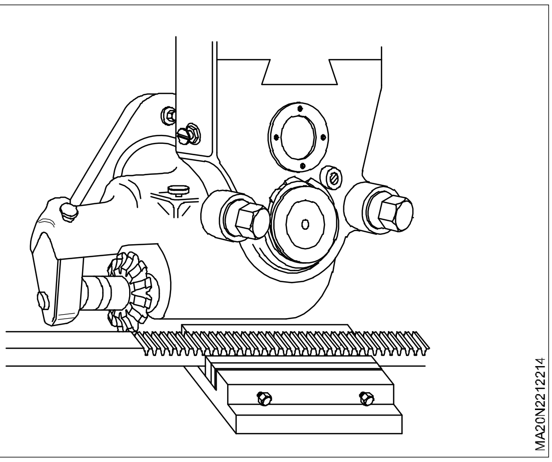
If the rack is longer than the maximum cross travel of the milling machine table, it must be held longitudinally on the table and should be generally held in a special fixture. The milling cutter is held in a rack milling attachment. (Fig 4)

Rack milling attachment

The rack milling attachment is designed to mount between the face of the column and the arbor support on a plain or universal milling machine. (Fig 4)

It consists of a fixed housing fitted with an angle drive and a spindle. The spindle axis is fixed parallel to the table.

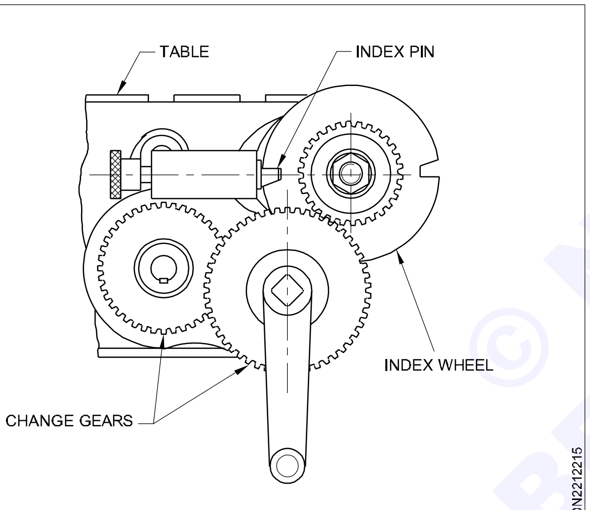
Fig 4



In rack milling, the cross-slide is used to feed the job against the cutter and the longitudinal table is used to index the job to produce the rack teeth.

In order that the work may be moved longitudinally, and to ensure the exact amount needed for accurate spacing of the teeth, a special rack indexing attachment Fig 5 is used.

Fig 5



RACK INDEXING ATTACHMENT

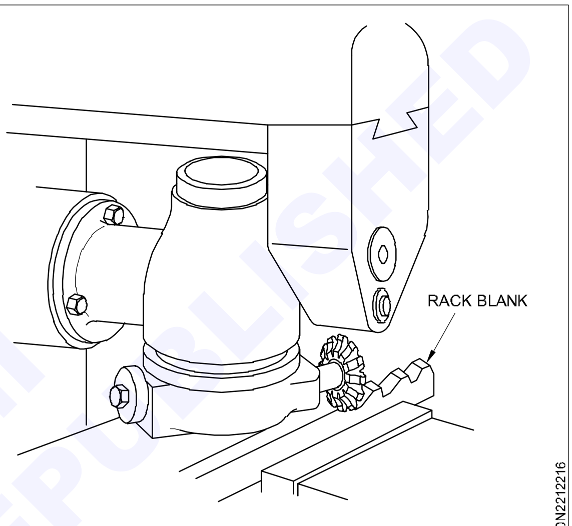
On left end of the table is fastened a bracket which carries a locking indexing wheel together with change gears for gearing to the table feed screw.

To index any required spacing, change gears are selected which will produce one or more complete turns of the indexing wheel.

This method is positive and much more reliable than setting the table to a graduated dial directly fitted to the table feed screw.

A universal or spiral attachment (Fig 6) can also be used to widen the scope of the rack indexing attachment to include operations such as milling the undercut teeth of a broach.

Fig 6



RACK MILLING WITH UNIVERSAL SPIRAL ATTACHMENT

Special vices with long jaws and bases are available for use in rack cutting operations.

Rack calculation

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

- **calculate the proportions of elements of a rack.**

A rack and a gear have the following common terms.

- Addendum
- Dedendum
- Whole depth
- Clearance
- Pressure angle

A rack may be a straight-toothed or a helical-toothed one. Calculation

Straight teeth rack

Addendum = m.

Dedendum = 1.25 m.

Whole depth = 2.25 m.

Linear pitch = πm

Tooth thickness = πm/2 = 1.5708 m.

Example

Data given

Module = 3

Number of teeth = 20

Addendum = m = 3 mm.

Dedendum = 1.25 m = 1.25 x 3 = 3.75 mm.

Whole depth = 2.25 m = 2.25 x 3 = 6.75 mm.

Linear pitch = πm = 3.14 x 3 = 9.42 mm.

Tooth thickness = πm/2 = 3.14 x 3/2 = 4.71 mm.

Length of rack = Linear pitch x Number of teeth

= 9.42 x 20 = 188.4 mm.

The tooth thickness in a rack need not be corrected by calculating the chordal thickness. It can be measured directly using a gear tooth vernier caliper.

The depth setting is equal to Addendum.

Helical rack

Data given

Module = 2.5

Helix angle β = 15°

Addendum = m = 2.5x1 = 2.5 mm

dedendum = 1.25 m = 1.25 x 2.5 = 3.12 mm.

Whole depth = 2.25 m = 2.25 x 2.5 = 5.62 mm.

Normal linear pitch = πm = 3.14 x 2.5 = 7.86 mm.

$$\text{Normal tooth thickness} = \frac{\pi m}{2} = \frac{7.86}{2} = 3.93\,\text{mm}$$

Linear pitch

$$= \frac{\pi m}{\text{Cos}\beta} = \frac{7.86}{\text{Cos}\,15^\circ} = \frac{7.86}{0.9659} = 8.137\text{mm}/ \cong 8.14\text{mm}$$

Selection of gear cutter and forms

Objectives : At the end of this lesson you shall be able to
- **determine the cutter number according to the teeth to be cut in a gear**
- **express the forms of gear cutter.**

Selection of cutter No

Cutting gear tooth of involutes profile 8 numbers of cutter are required to cut from a pinion of 12 teeth to rack and for cycloidal tooth profile 24 number cutter number are used for cutting different number of gear teeth. List of cutter with number of teeth are intended to cut are as follows. (Table 1 & Table 2)

Cutter for Involute and Cycloidal gear teeth

Table 1 - Involute gear

Cutter No.	No. of teeth cut
No.1	135 teeth to a rake
No.2	55 to 134 teeth
No.3	35 to 54 teeth
No.4	26 to 34 teeth
No.5	21 to 25 teeth
No.6	17 to 20 teeth
No.7	14 to 16 teeth
No.8	12 to 13 teeth

Table 2 - Cycloidal gear

Cutter No	No. of teeth cut	Cutter No	No. of teeth cut
No.A	12 teeth	No.M	27 to 29 teeth
No.B	13 teeth	No.N	30 to 33 teeth
No.C	14 teeth	No.O	34 to 37 teeth
No.D	15 teeth	No.P	38 to 42 teeth
No.E	16 teeth	No.Q	43 to 49 teeth
No.F	17 teeth	No.R	50 to 59 teeth
No.G	18 teeth	No.S	60 to 74 teeth
No.H	19 teeth	No.T	75 to 99 teeth
No.I	20 teeth	No.U	100 to 149 teeth
No.J	21 to 22 teeth	No.V	150 to 249 teeth
No.K	23 to 24 teeth	No.W	250 or more
No.L	25 to 26	No.X	Cuts a rack

The standard properties adopted by the Indian standard system (Module) and the American standard system (D.P)

The recommended series of modules are 1, 1.25, 1.5, 2, 2.5, 3, 4, 5, 6, 8, 10 ,12, 16 and 20 and diametrical pitches are 20, 16, 12, 10, 8, 7, 6, 5, 4, 3 2 ½ , 2, 1 ½, 1 ¼ and 1

Gear tooth vernier caliper

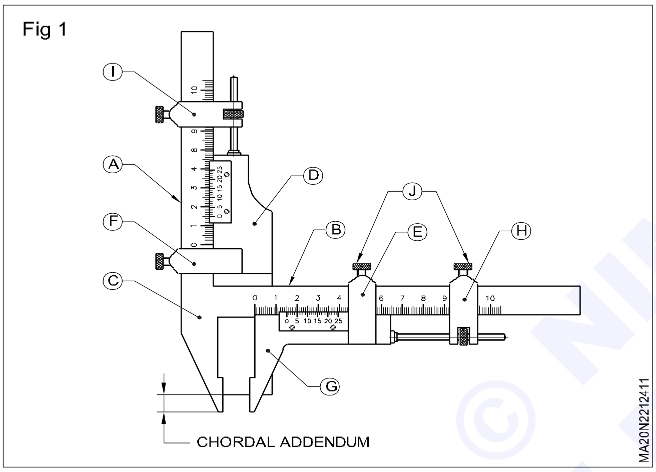
Objectives : At the end of this lesson you shall be able to
- **explain the parts of the instrument**
- **solve the problems to find chordal addendum (hac) and addendum height (ha).**

This instrument is very similar to the usual vernier calipers in principle but differs in construction.

The gear tooth vernier caliper shown in Fig 1 has a (L) shaped) frame which integrates the fixed jaw (C), the graduated vertical beam (A) and the graduated horizontal beam (B) to which sliding heads (E) and (F) with vernier scales are assembled.

The vertical sliding head carries a thin blade (D) and the horizontal sliding head carries the movable jaw (G).

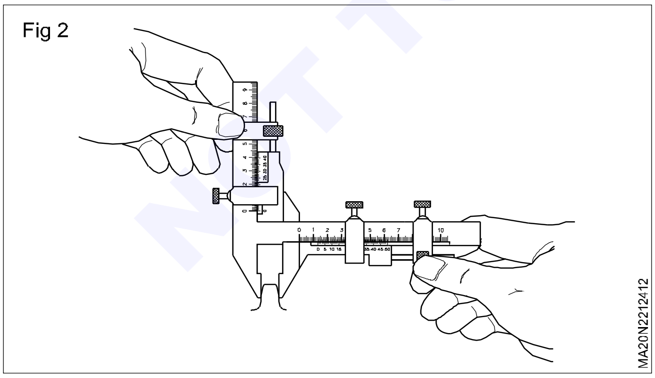
Fig 1



Both the slides are controlled by fine adjustment heads (H) and (I) and can be locked in position by screws (J).

While checking (Fig 2) the gear tooth, the vertical slide with blade (G) is used to set the addendum height (ac) and the horizontal slide with a movable jaw (H) is used to set the chodal tooth thickness (tc).

Fig 2



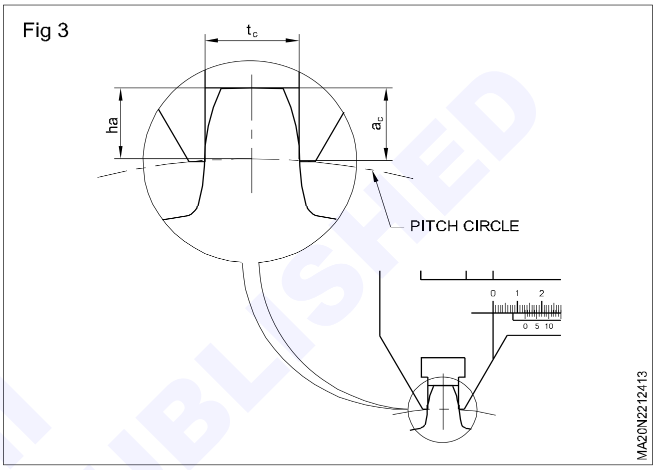
Measurements of gear tooth (Fig 3)

$$<AOB = R \sin\left(\frac{90}{Z}\right)$$

Chordal thickness = tc = $mZ \, \text{Sin}\left(\frac{90}{Z}\right)$

Chordal addendum ac = $m + \frac{mZ}{2}\left(1 - \text{Cos}\,\frac{90}{Z}\right)$

Fig 3



Where R = Radius of the pitch circle
Z = Number of teeth on the gear
ac = chordal addendum
tc = Chordal thickness

Example: To calculate the gear tooth venire settings for measuring a gear of 33 teeth 6 module.

$$tc = mZ \text{ x Sin}\left(\frac{90}{Z}\right)$$

$$= 6 \text{ x } 33 \text{ x}\left(\text{Sin}\frac{90}{33}\right)$$

$$= 198 \text{ x (Sin } 2º.43')$$

$$= 198 \text{ x } (0.0474) = 9.3852 \quad \therefore tc = 9.39mm.$$

Chordal thickness = 9.39mm.

$$ac = m + \frac{mZ}{2}\left(1 - \text{Cos}\,\frac{90}{Z}\right)$$

$$= 6 + \frac{6 \text{ x } 33}{2}\left(1\text{- Cos}\,\frac{90}{33}\right)$$

$$= 6 + 99 \text{ (1 - Cos } 2º.43')$$

$$= 6 + 99 \ (0.0011)$$

$$= 6 + 0.1089$$

$$= 6.1089 = 6.11$$

$\therefore$ ac = chordal addendum = 6.11mm.

Radius gauges and templates

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

Radius gauges are used to check internal and external radius of work pieces.

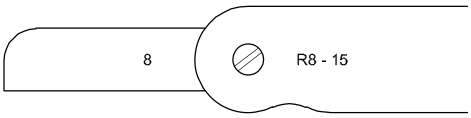
These gauges are made of high quality steel sheets, and are finished to accurate radius.

Radii of parts are checked by comparing the radius of the gauges.

Radius gauges are available in sets of several blades held in a holder. Each blade can be separately pulled out of the holder when wanted for use.

The size of the radius is marked on the individual blades of the gauges. (Fig 1)

Fig 1



These gauges are available in different combinations with internal and external radius. (Figs 2 and 3)

Individual gauges are also available for different radii. (Fig 4)

Before using radius gauges ensure the gauges are perfectly clean, remove burrs, if any, from the work pieces, check and make sure there is no damage to the profile of the gauge.

Fig 2

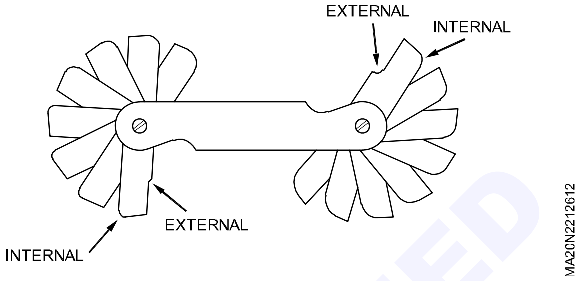


Fig 3

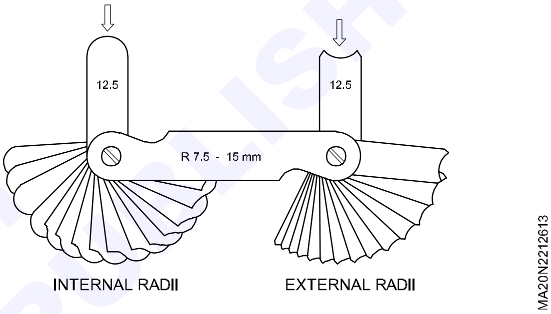
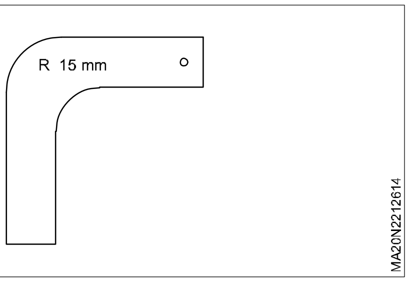


Fig 4

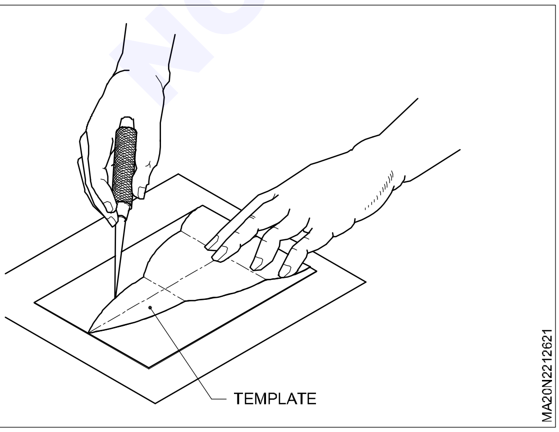


Templates (Profiles)

Objective : At the end of this lesson you shall be to
- **state the necessity of templates for marking and checking.**

A template is made of good quality steel. It is used to mark and check profiles (Fig 1)

Fig 1



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

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

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

Vertical milling machine

Objectives : At the end of this lesson you shall be able to
- **list the parts of a vertical milling machine**
- **describe the construction of a vertical milling machine**
- **state the uses of a vertical milling machine**
- **difference between horizontal and vertical milling machines.**

Parts, construction and functions

A vertical milling machine is similar to a horizontal milling machine in all aspects except the spindle position. The spindle is supported by a vertical head in the vertical plane. The vertical head is attached to the column. The spindle rotates about the vertical axis perpendicular to the table surface.

Parts

The parts of a vertical milling machine are shown in Fig 1.

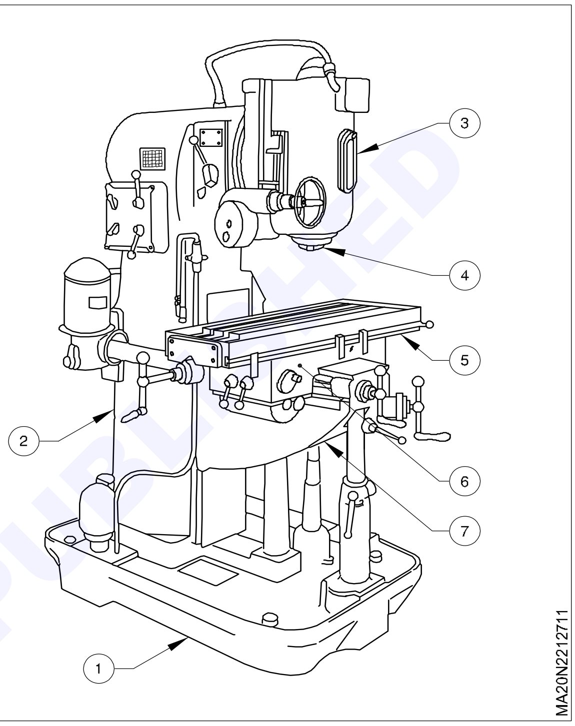
1 Base
2 Column
3 Vertical head
4 Spindle
5 Table
6 Saddle
7 Knee

The construction and function of all parts except the vertical head (3) and the spindle (4) are the same as plain or horizontal milling machines.

The vertical head (3) is attached to the column (2), it can be swivelled up to 45° in both left/right hand directions.

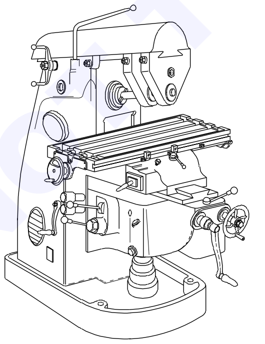
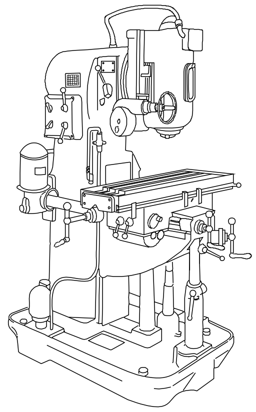
The spindle (4) is mounted in the vertical head (3) and rotates about a vertical axis. The spindle can be moved up and down even when the spindle is rotating. This arrangement is useful to give feed while boring.

Fig 1



A vertical milling machine is used for face milling, 'T' slot milling, angular milling, end milling, keyway milling etc.

Differences between horizontal and vertical milling machines

Horizontal milling machine	Vertical milling machine
 The spindle axis is in horizontal plane axis parallel to the table surface. The position of the spindle cannot be changed. Mainly, peripheral milling operation is done on this machine, using bore type cutters.	 The spindle is in the vertical plane and rotates about a vertical axis perpendicular to the table surface. The spindle can be moved up and down.

Angular milling cutters and swivelling vices are required to do angular milling. The operator will not have a good view of the milling cutter and the cutting zone. Normally, long arbors are used to mount the cutters. The tendency to chatter is more.	Face milling and end milling operations are done on this machine using end mills and face mills. Angular milling can be done just by swivelling the spindle head. The operator will have a better view of cutter and cutting zone. The cutters are mounted on stub arbors or directly on to the spindle. So the tendency to chatter is less.

Boring heads and methods of boring in vertical milling machine

Objectives : At the end of this lesson you shall be able to
- **name the different boring bars**
- **identify each type of boring bars**
- **state the uses of each type.**

Boring is done on the vertical milling machine by using boring bars with single point tool.

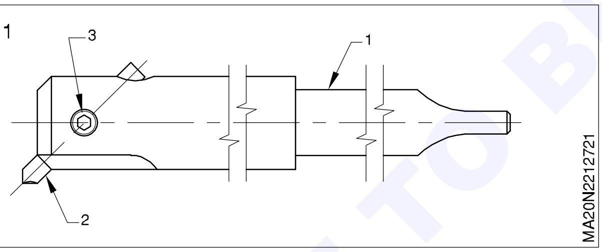
The different boring bars are:

- Solid boring bar
- Adjustable boring bar or micro boring bar
- Boring heads.

Solid boring bar (Fig 1)

It is the simplest form of a solid bar with an angular or straight hole at one end to mount the tool, a standard shank (1) (ISO or Morse taper) at the other end. The tool (2) is held in position by a locking screw(3). The tool is adjusted further at the end of every cut.

Fig 1



Adjustable boring bar (Fig 2)

This type of boring bar is capable of some fine adjustment of the tool within short ranges. The tool bits are generally of a throw away type. The tools bits are mounted on the cartridges (1) with micrometer collars (2) called micro cartridges for accurate adjustments. It is useful to maintain sizes of bores within close tolerances.

Boring heads (Fig 3)

A boring head is a rigid tool holder used to hold boring tools or boring bars. The boring heads are used to bore large holes. The range of adjustment varies from 100 to 500 mm. The boring tools can be mounted axially(1) or radically (2) in the boring head according to the size of the bore to be machined.

Fig 2

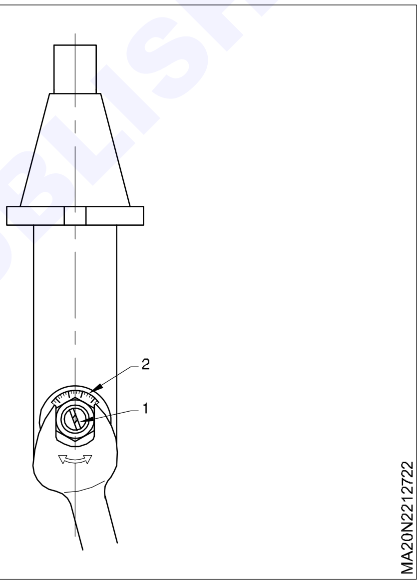
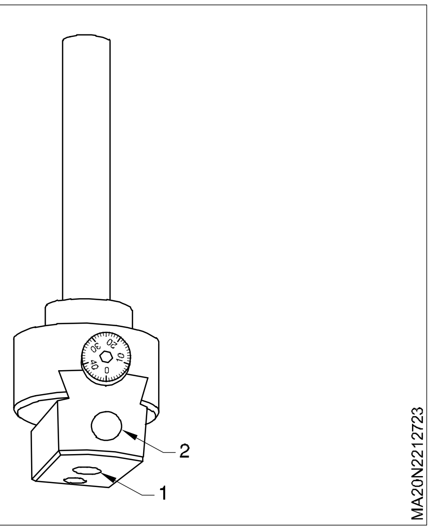


Fig 3



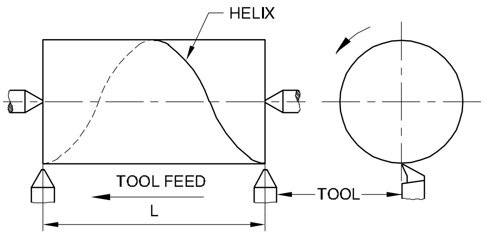
Helix and spiral introduction

Objectives : At the end of this lesson you shall be able to
- **define helix and spiral**
- **state the elements of a helix**
- **difference between a helix and a spiral.**

Helix

A helix is the line generated by the progressive rotation of a point around a cylinder. (Fig 1)

Fig 1



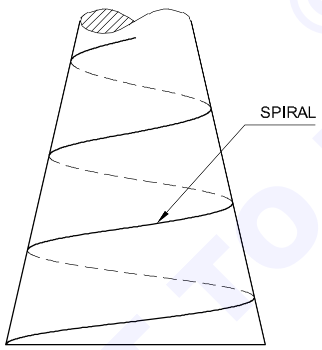
A HELIX WILL BE GENERATED IF THE WORK IS TURNED AS THE TOOL IS MOVED ALONG UNIFORMLY

The flutes on a drill or the threads on a bolt are examples of helies.

Spiral

A spiral is the path generated by the progressive rotation of a point moving along the surface of a cone is called conical spiral. (Fig 2)

Fig 2



A SPIRAL IS PRODUCED ON A CONICAL SURFACE

Threads on a wood screw and pipe threads are examples of conical spirals. While watch springs and scroll threads on a self centering lathe chuck are examples of plane or flat spirals.

In order to cut a helix or a spiral, any two of the following elements must be known.

1 The lead of the helix

2 The angle of the helix

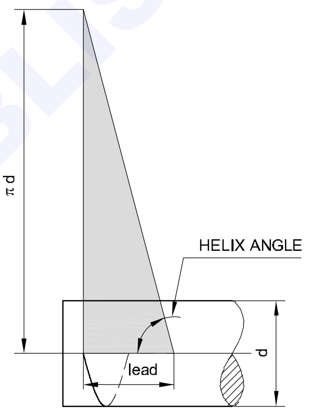
3 The diameter (or circumference) of the work piece.

   i The lead of a helix is the longitudinal distance, the helix advances axially in one complete revolution of the work.

   ii The angle of helix is formed by the intersection of the helix with the axis of the work piece.

The relationship of lead circumference and helix angle is shown in the Fig 3.

Fig 3



RELATIONSHIP OF LEAD, CIRCUMFERENCE AND HELIX ANGLE

The lead of a helix varies with

- The diameter of the work
- The angle of the helix

Differentiate between helix and spiral

Helix

A line traced on a cylindrical surface where all points of the surface area cut at the same angle.

Spiral

A simple curve in the plane which continuously winds about itself either into some point or out from some point.

Spiral milling calculations

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

- **state the applications of the helix**
- **calculate the angle of helix for both metric and the British systems of dimensions**
- **determine the direction of swivel of the table with reference to the hand of helix**
- **calculate change gears for the required lead.**

Applications of helix

Helical grooves are used as flutes on cutting tools like reamers, milling cutters etc.

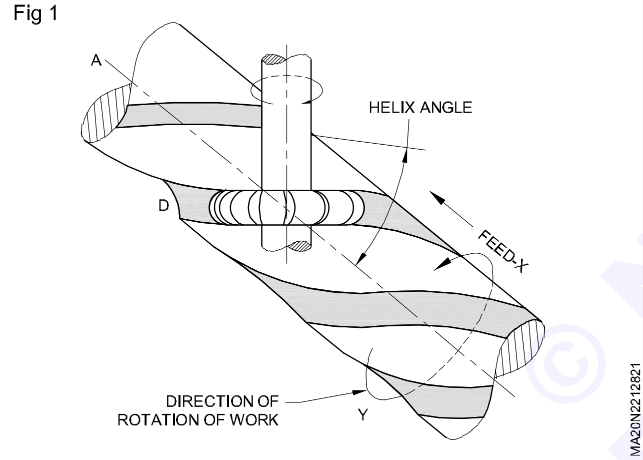
Helical grooves cut on shafts to lubricate its bush bearings.

Flat spirals known as across are used on a lathe's self-centering chucks to move the jaws for clamping the work.

Determining the helix angle

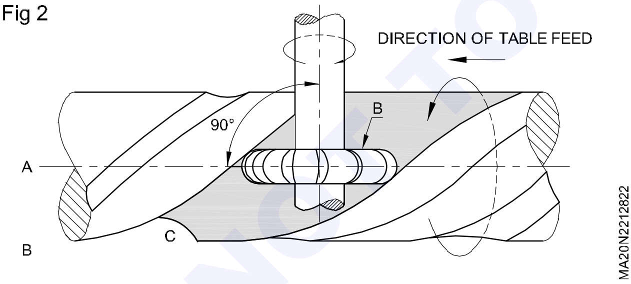
To ensure that a groove of the same contour as the cutter is produced, the table must be swung to the angle of the helix. (Fig 1)

Fig 1



Note that when the table is not swung (Fig 2), a helix having the proper lead but an improper contour will be generated.

Fig 2



To determine the angle of swivel of the table, it is necessary to calculate the tangent of the helix angle.

Tangent of helix angle

$$= \frac{\text{Circumference of work}}{\text{lead of helix}}$$

$$= \frac{\pi \times D}{\text{lead of helix}}$$

Example (British system)

To what angle must the milling machine table be swivelled to cut a helix having a lead of 10.882 in a piece of work of diameter 2 inches.

Solution

Tangent of helix angle

$$= \frac{\pi \times D}{\text{lead of helix}}$$

$$= \frac{3.1416 \times 2}{10.8882}$$

$$= 0.57739$$

Helix angle $= 30^{\circ}$.

Example (Metric)

To what angle must a milling machine table be swivelled to cut a helix having a lead of 450 mm on a workpiece of diameter 40 mm.

Tangent of helix angle

$$= \frac{3.1416 \times D(\text{mm})}{\text{lead of helix}(\text{mm})}$$

$$= \frac{3.1416 \times 40}{450}$$

$$= 0.2792$$

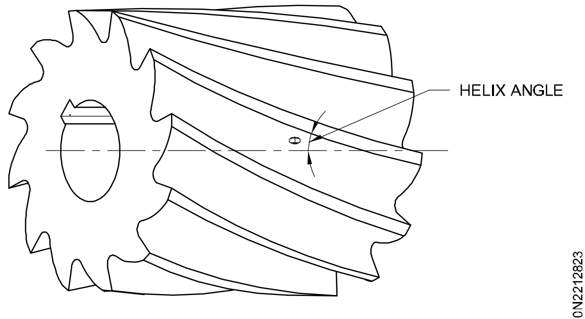
$$= 15^{\circ} - 36'.$$

Determining the direction to swing the table

In order to determine the hand of a helix, hold the cylinder on which the helix is cut in a horizontal plane with its axis running in a right-left direction.

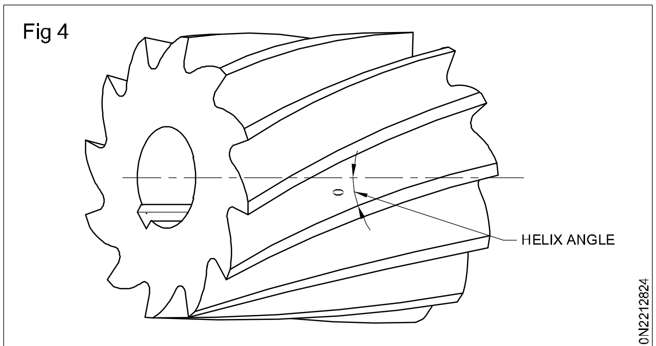
If the helix slopes down to the right, it is a right hand helix (Fig 3). A left hand helix slopes down to the left. (Fig 4)

Fig 3



RIGHT HAND HELIX

Fig 4



LEFT HAND HELIX

When a left hand helix is to be cut, the table of the milling machine must be swivelled in a clockwise direction (operator standing in front of the machine facing the spindle of machine.

A right hand helix may be produced similarly by moving the right hand end of the table, towards the column or by moving it in a counter-clockwise direction.

Calculating change gears to produce the required lead

To cut a helix, it is necessary to have the work move longitudinally and rotate at the same time.

The amount the work (ie., table) travels lengthwise as the work revolves one complete revolution is called the lead of helix. The rotation of the work is caused by gearing the auxiliary worm shaft of the dividing head to the lead screw of the machine-table.

Inch calculations

To cut a helix on an inch milling machine, it is necessary first to understand how to calculate the required change gears for any desired lead.

Assume that the dividing head auxiliary worm shaft is geared to the table lead screw with equal gears (for example both having 24-tooth gears). The dividing head ratio is 40:1, while a standard milling machine lead screw has 4 threads per inch (TPI). The lead screw, as it revolves one turn, would revolve the dividing head spindle one-fortieth of a revolution. In order for the dividing head spindle to revolve one turn, it would be necessary for the lead screw to revolve 40 times.

Machine lead

Thus the table would travel 40 x 1/4 inch or 10 inch while the work revolves one turn. Therefore, the lead of a milling machine is said to be 10 inch when the lead screw. (4 TPI) is connected to the dividing head (40:1 ratio) with equal gears ie.,1:1 ratio. In calculating the change gears required to cut any lead, the following formula may be used.

$$\text{Gear ratio} = \frac{\text{Lead of helix}}{\text{Lead of machine(10 inch)}}$$

$$= \frac{\text{Product of driven gears}}{\text{Product of driver gears}}$$

The ratio of gears required to produce any lead on a milling machine having a lead screw with 4 TPI is always equal to a fraction having the lead of the helix for the numerator and 10 for the denominator.

The preceding formula may be inverted if preferred.

$$\text{Gear ratio} = \frac{\text{Lead of the machine}}{\text{Lead of the helix}}$$

$$= \frac{\text{Product of driver gears}}{\text{Product of driven gears}}$$

Example-1

Calculate the change gears required to produce a helix having a lead of 25 inches on a piece of work. The available change gears have the following number of teeth. 24, 24, 28,32, 40, 56, 64, 72, 86, 100 ie., std. set of gears.

Solution

$$\text{Gear ratio} = \frac{\text{Lead of helix (driven gears)}}{\text{Lead of machine (driver gears)}}$$

$$= \frac{25}{10}$$

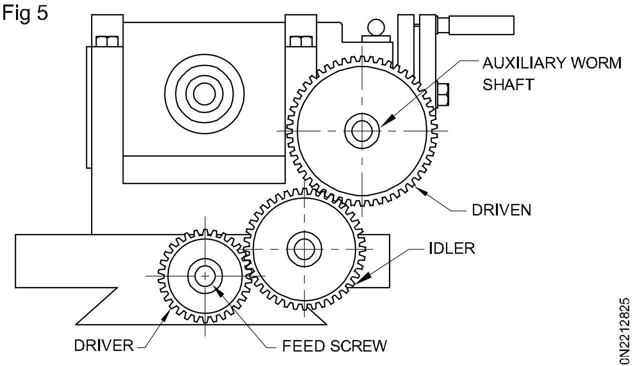
Since 10 and 25 tooth gears are not supplied with standard dividing heads, it is necessary to multiply the 25:10 by any number that will suit the change gears available.

$$\text{Gear ratio} = \frac{25}{10} \times \frac{4}{4}$$

$$= \frac{100\,\text{(Driven gears)}}{40\,\text{(Driver gears)}}$$

As both 100 tooth and 40 tooth gears are available, a simple gear train may be used. (Fig 5)

Fig 5



SIMPLE GEAR TRAIN

Example-2

Determine the change gears required to produce a helix having a lead of 27 inches using a std. set of change gears.

Solution

$$\text{Gear ratio} = \frac{\text{Lead of helix(driven gears)}}{\text{Lead of machine(driver gears)}} = \frac{27}{10}$$

Since there are no gears in the set which are multiples of both 27 and 10, it is impossible to use simple gearing. Compounding gearing must, therefore, be used, and it because necessary to factorise the fraction.

Gear ratio

$$= \frac{27}{10} \text{ x } \frac{3}{2} \text{ x } \frac{9 \text{ (driven)}}{5 \text{ (driver)}}$$

This does not change the value of the faction.

$$\frac{3 \text{ x } 16}{2 \text{ x } 16} = \frac{48}{32}$$

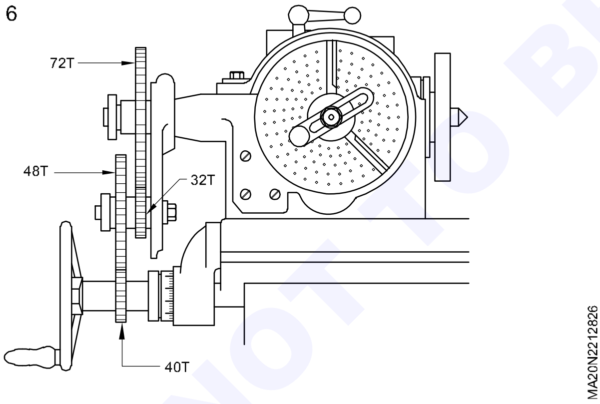
$$\frac{9 \text{ x } 8}{5 \text{ x } 8} = \frac{72}{40}$$

$$\text{The gear ratio} = \frac{48}{32} \text{ x } \frac{72\text{(driven gears)}}{40\text{(driver gears)}}$$

Therefore the driven gears are 48 and 72 and the driver gears are 32 and 40.

The gears would be placed in the train as follows. (Fig 6)

Fig 6



Gear on worm 72 (driven)

First gear on stud.32 (driver)

Second gear on stud 48 (driven)

Gear on lead screw 40 (driver)

The preceding order is not absolutely necessary the two driven gears may be interchanged and/or the two driver gears may be interchanged, provided a driver is not interchanged with a driven gears.

Metric calculations

Most of the standard milling machine lead screws have a 5 mm pitch and the dividing head has a ratio of 40:1. As the lead screw revolves one turn, it would rotate the dividing head spindle (and work) 1/40 turn to rotate one full turn, the lead screw must make 40 complete revolutions. Therefore, the lead of the machines would be 40 times the pitch of the lead screw.

For metric calculations the change gears required is calculated as follows.

$$\text{Gear ratio} = \frac{\text{Lead of helix(mm)}}{\text{Lead of machine(mm)}}$$

$$= \frac{\text{Product of driven gears}}{\text{Product of driver gears}}$$

The normal change gears in a set are 24, 24, 28, 32, 40, 44, 48, 56, 64, 72, 86, and 100 (12 gears).

Example

Calculate the change gears required to cut a helix having a lead of 500 mm on a workpiece using a standard set of gears. The milling machine lead screw has pitch of 5 mm.

$$\frac{\text{Driven gears}}{\text{Driver gears}} = \frac{\text{Lead of helix(mm)}}{\text{Pitch of lead screw}} \text{ x } \frac{1}{40}$$

$$= \frac{500}{5 \text{ x } 40} = \frac{500}{200} = \frac{5}{2} = \frac{5 \text{ x } 20}{2 \text{ x } 20} = \frac{100}{40} = \frac{\text{Driven}}{\text{Driver}}$$

Driven gear = 100, Driver gear = 40.

Direction of spindle rotation

For cutting a right-hand helix, the gear on the lead screw and the worm shaft must revolve in the same direction and for cutting the left hand helix, the spindle must revolve in the opposite direction that of the gear on the lead screw.

Use of idler gears

The idler in this case acts neither as a driven nor as a driver gear, and is not considered in the calculation of the gear train. It acts merely as a means of changing the direction of rotation of the dividing head spindle. It should also be noted that the direction of spindle rotation for simple gearing will be opposite to that for compound gearing if idler gear is not introduced in the gear train.

Short lead helix

When it is necessary to cut leads smaller than those given in most machinery handbooks, it is advisable to disengage the dividing head worm and worm wheel and connect the change gears directly from the table lead screw to the dividing head spindle rather than to the worm shaft. Consider that the machine is geared to cut a lead of 4,000 inches and if we connect the worm shaft and the lead screw, the same gears would produce a lead of 1/40 x 4,000 inch or 0.1000 inch instead of 4,000.

Helical gear - application elements and calculation

Objectives : At the end of this lesson you shall be able to
- **state the applications of helical gear**
- **state the advantages and disadvantages of helical gears**
- **define the important elements of helical gear**
- **calculate the proportion of elements for both metric and inch pitches**
- **select the correct cutter number for milling a helical gear.**

Like spur gears, the helical gears are employed to transmit motion between parallel shafts. These gears can also be used for transmitting motion between non-parallel, non-intersecting shafts.

Helical gears employed to transmit motion between parallel shafts are called parallel helical gears.

Helical gears employed to transmit motion between non-parallel and non-intersecting shafts are called crossed helical gears.

The advantages of helical gear over spur gears are:

1 smooth operation because of sliding action

2 greater load carrying capacity because more than one tooth are in contact

3 noiseless running since there is no hammering action while running.

The disadvantage of helical gearing is that it generates axial thrust forces. These can be taken care of by proper bearing mountings or by using a double helical gear drive.

Application of a helical gears

Helical gears are used for high speed running and heavy duty gear drives.

Elements of a helical gear

The definitions of basic nomenclature and gear tooth terminology are the same in spur and helical gears.

The special features which characterise and differentiate a helical gear from a spur gear are explained here-under.

Lead

The distance between the two intersections of a helix by a straight generator of the pitch cylinder on which it lies.

Helix angle

The acute angle between the tangent to the helix and the straight generator of the pitch cylinder on which it lies.

The formula for the helix angle is given below:

$$\tan\beta = \frac{\pi D}{l}$$

where β = the helix angle

D = pitch diameter of the gear

l = lead of the helix.

Helical gear calculations

Helical gear calculations are based upon the normal circular pitch and not upon the circular pitch measured in the plane of rotation.

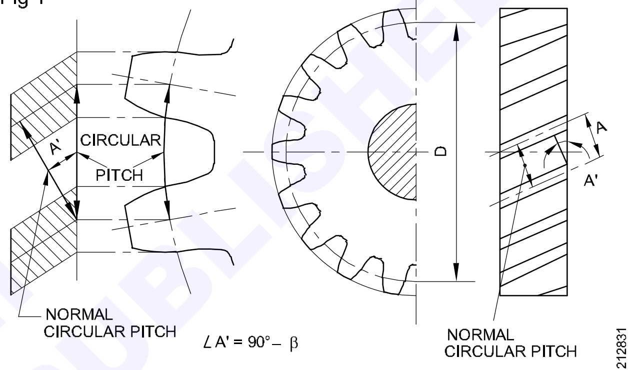
Helical gearing requires a thorough knowledge of all the aspects of spur gear as well as the formula and procedures used in helical milling.

In helical gear cutting the two new terms that we come across are the normal circular pitch (NCP) and normal diametric pitch.(NDP)

The NCP is the distance from a point on one tooth to a corresponding point on the next tooth, measured on the pitch circle at right angles to the face of the tooth.

Circular pitch (CP) is the length of the arc between similar points on the faces of adjacent teeth measured on the pitch cylinder in a plane perpendicular to the axis of rotation. (Fig 1)

Fig 1



RELATIONSHIP BETWEEN THE CIRCULAR PITCH AND THE NORMAL CIRCULAR PITCH

Normal circular pitch of helical gear changes with the helix angle α.

Normal circular pitch = CP_n = CP Cos α

$$= \frac{\pi \text{ x} \cos\alpha}{DP}$$

Normal diametric pitch

$$DP_\eta = \frac{\pi}{CP_\eta} = \frac{DP}{\cos\alpha} = \frac{N}{PD\cos\alpha}$$

Normal module

$$m_\eta = \frac{CP_\eta}{\pi} = \frac{\cos\alpha}{DP} = \frac{d^1 \text{ x} \cos\alpha}{N}$$

$$= m \cos\alpha$$

Where

d^1 = pitch diameter,

N = Number of teeth.

It should be noted that a right angle triangle is formed, with a line representing the CP as the hypotenuse and a line representing the NCP as the side opposite.

By simple geometry, it may be proved that the angle ‘A’ is equal to the A1, which is the angle of the helix. Therefore, in the triangle in Fig 1.

$$\text{Cosine of Angle A} = \frac{NCP}{CP}$$

Therefore, the relationship between NCP and CP is exactly proportional to the length of these lines.

The number of teeth in a helical and a spur gear of the same size and pitch will also be proportional to the length of these lines.

It is obvious that the CP will increase as the helix angle increases; therefore, the greater the helix angle, the fewer the teeth in a helical gear as compared to a spur gear of the same pitch diameter.

Since most gears is calculated on the DP system, it will be necessary to convert the CP into DP terms.

Referring to the spur gear formulae

$$DP = \frac{3.1416}{CP} \quad \text{or} \quad CP = \frac{3.1416}{DP}$$

Since normal diameter pitch (NDP) bears the same relation to NCP as DP does to CP, the following formulae will apply:

$$NDP = \frac{3.1416}{NCP}$$

$$NCP = \frac{3.1416}{NDP}$$

$$\text{Cos helix angle} = \frac{NCP}{CP}$$

This may be converted to DP as follows

$$\text{Cos helix angle} = \frac{\frac{3.1416}{NDP}}{\frac{3.1416}{DP}}$$

$$= \frac{3.1416}{NDP} \times \frac{DP}{3.1416}$$

$$= \frac{DP}{NDP}$$

Therefore Cos helix angle

$$= \frac{DP}{NDP}$$

Other formulae may be derived as follows

$$NDP = \frac{DP}{\text{Cos helix angle}}$$

$$DP = NDP \times \text{Cos helix angle}$$

Remember that, in a job drawing, always normal pitch is given.

Cutter Selection

The type of cutter used for cutting helical gear is the same as for spur gears. However, the DP of a spur gear and the gear-cutter now becomes the NDP of the helical gear. The thickness of the cutter at the pitch line for cutting helical gears should be equal to one-half of the NCP.

When a spur gear is being cut, the number of the gear cutter used is dependent on the number of teeth being cut. However, when a helical gear is being cut, this table does not apply since the shape of the tooth would be changed because of the helix angle of the gear teeth. To determine the proper cutter for machining a helical gear, it is necessary to consider the virtual number of teeth, if it would be a spur gear being cut and the helix angle of the tooth.

The cutter number may be determined by dividing the number of teeth (N) by the cube of the Cosine of the angle.ie.,

$$\text{Cutter number} = \frac{N}{(\text{Cos}\,\alpha)^3}$$

Example

Determine the cutter number required to cut 38 teeth on a helical gear having an angle of 45°.

Number of teeth for which to select the cutter

$$= \frac{38}{(0.7071)^3}$$

$$= \frac{38}{0.3534} = 108$$

Therefore cutter number 2 would be used since it cuts from 55 to 134 teeth.

Helical gear calculations and formulae

Many of the calculations and formulae that apply to spur gears also apply to helical gears.

It is good to remember the following when making helical gears.

1 The DP of a spur gear is the NDP of a helical gear

2 The NDP of the helical gear is the DP of the cutter

3 The cutter number must be calculated using the formula.

$$\text{Number of teeth for which to select a cutter} = \frac{N}{(\text{Cos}\,\alpha)^3}$$

The alignment chart (Fig 2) will be found convenient when it is necessary to determine cutter number for helical gears.

By means of a straight edge, align the actual number of teeth in the left hand column with the helix angle in the right hand column.

Read the cutter number from the centre column where the straight edge crosses the line.

Metric helical gear proportions

Most metric module helical gear calculations are similar to those for DP helical gears and module spur gears.

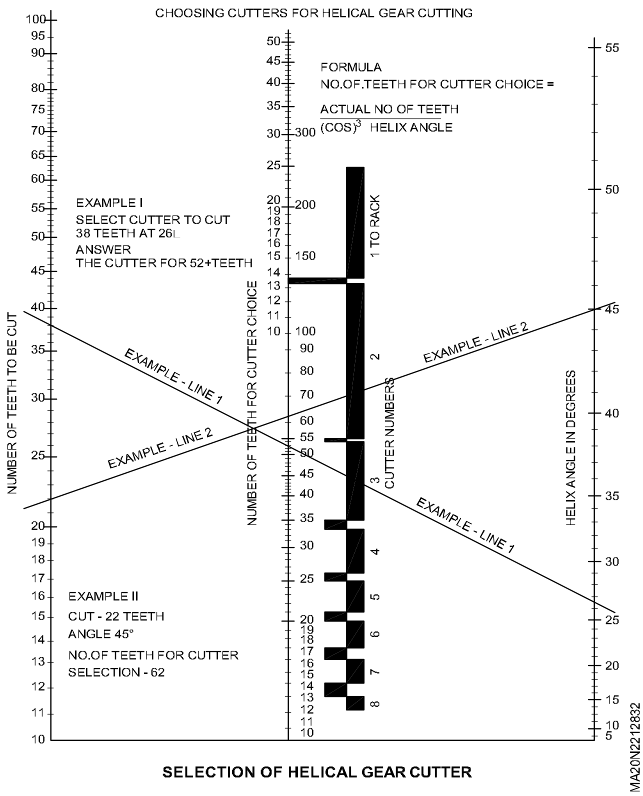
There are, however, certain changes which must be understood before any calculations are attempted. Refer to Fig (1) to understand more clearly the following terms.

– Circular pitch (CP)

– Normal circular pitch (nCP)

The real (pitch) module (M) is calculated on the pitch circle.

Fig 2



SELECTION OF HELICAL GEAR CUTTER

Example-1

It is required to cut a helical gear of 5 nM having 19 teeth. The pitch diameter (PD) of the gear is 100 mm. Pitch of the lead screw is 5.0 mm. The type of helix to be cut is R.H.

Calculate

1 Real module (M)
2 Normal module (nM)
3 Circular pitch (CP)
4 Normal circular pitch (nCP)
5 Addendum (A)
6 Outside diameter (OD)
7 Whole depth of tooth (WD)
8 Helix angle (α)
9 Lead (–)
10 Indexing movement
11 Gear-train
12 Cutter number

Solution

1 Real module (M) $= \frac{PD}{N}$

$$= \frac{100}{19} = 5.26\text{mm}$$

2 Normal module (nM) = 5

3 Circular pitch (CP) = πM
= 3.1416 x 5.26
= 16.52 mm

4 Normal circular pitch (nCP)
= πnM
= 3.1416 x 5
= 15.70 mm

5 Addendum (A) = M
= 5 mm

6 Outside dia. (OD) = PD + 2nM
= 100 + 10
= 110 mm

7 Whole depth (WD) = 2.25 x nM
= 2.25 x 5
= 11.25 mm

8 Helix angle (Cosα) $= \frac{nM}{M}$

$$= \frac{5}{5.2632} = 0.950$$

α = 18°12'

9 Lead (–) = PD x π x Cot$\angle\alpha$
= 100 x 3.1416 x 3.0415
= 955.5 mm

10 Indexing $= \frac{40}{N}$

$$= \frac{40}{19} = 2\frac{1}{19}$$

2 turn of crank + 1 space in the 19 hole circle.

11 Gear train

Lead of the machine = 5 x 40
= 200.0 mm

Lead of the job = 955.5 mm

$$= \frac{\text{Driver}}{\text{Driven}} = \frac{\text{Lead of machine}}{\text{Lead of the job}}$$

$$= \frac{200}{960} = \frac{20}{96} = \frac{4x5}{12x8}$$

$$= \frac{(4x6)x5x8}{(12x6)x8x8} = \frac{24x40}{72x64}$$

24 T and 40 T are driver gears

72 T and 64 T are driven gears

12 Cutter number

$$= \frac{\text{Number of teeth}}{(\text{Cos }\alpha)^3}$$

$$= \frac{19}{(\text{Cos}18°12')^3} = \frac{19}{(.9500)^3}$$

$$= \frac{19}{.8574} = 22.16 \text{ say } 22T$$

Select cutter No. 5

Setting of the table

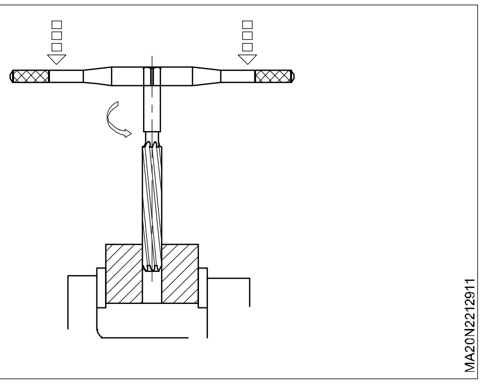
To cut R.H. Helix. Push right side of the table to 18º12'.

Reamers - types, elements and uses

Objectives : At the end of this lesson you shall be able to
- **state the uses of reamers**
- **distinguish between hand and machine reamer**
- **name the elements of reamer and state their functions.**

A reamer is a multipoint 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

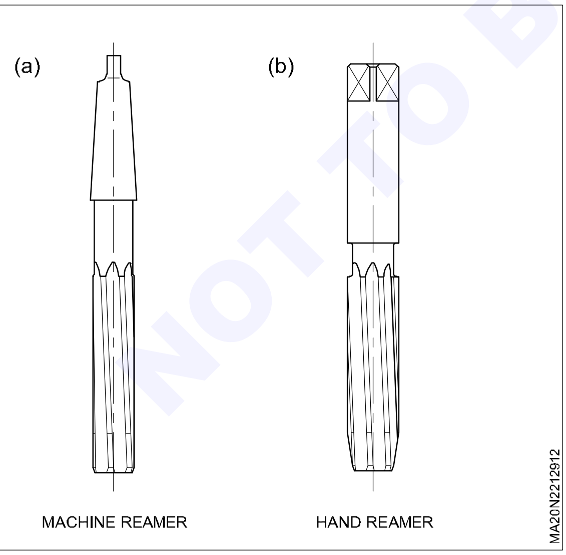
dimensional accuracy to close limits.

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 (a and b)

Fig 2



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

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

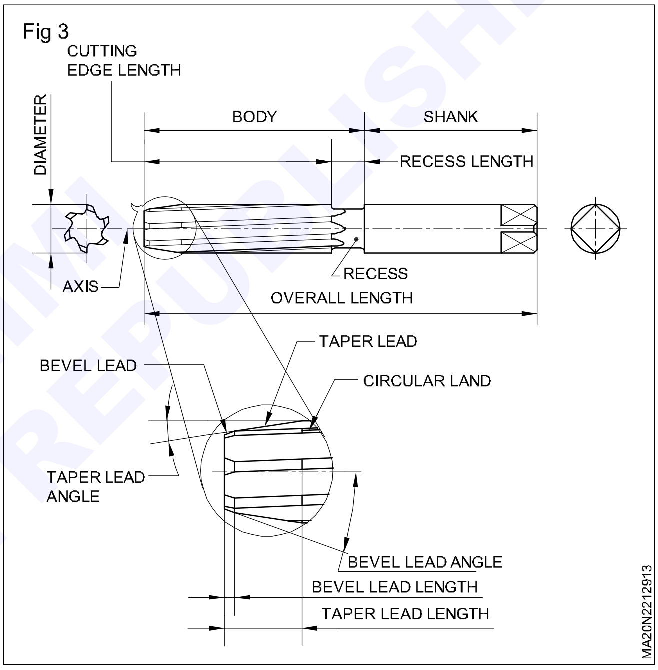
Machine reamers are provided with morse taper shanks for holding on machine spindles.

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

Elements of a hand reamer

The parts of a hand reamer are listed hereunder. Refer to Fig 3.

Fig 3



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 of the shank.

Recess

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

Shank

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

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 hole. 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

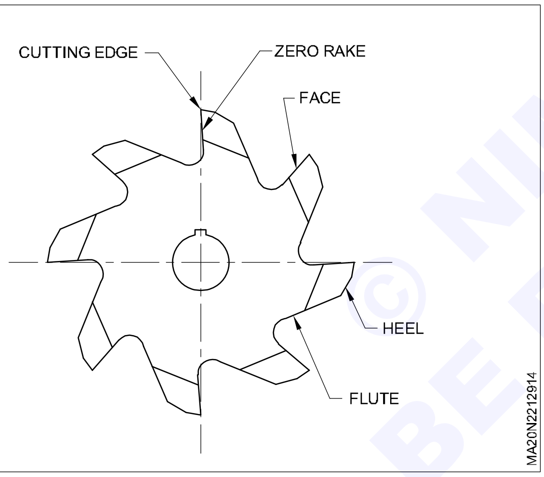
The angle formed by the cutting edges of the taper and the reamer axis.

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 4)

Fig 4



Heel

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

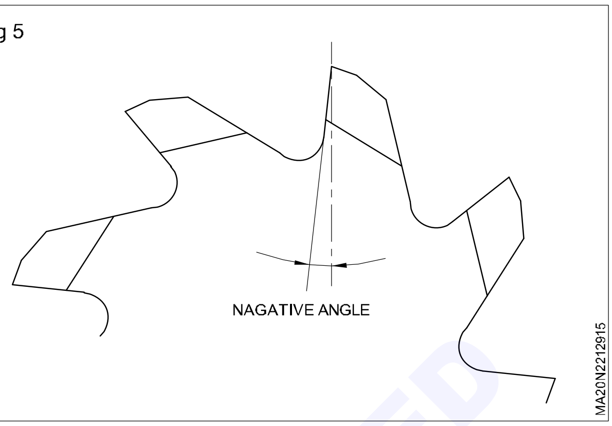
Cutting edge

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)

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)

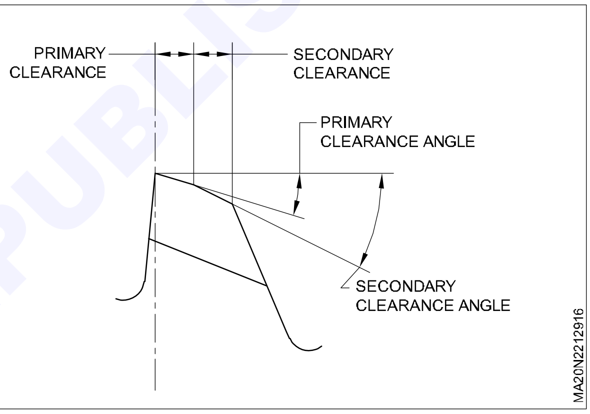
Fig 5



Rake angles

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

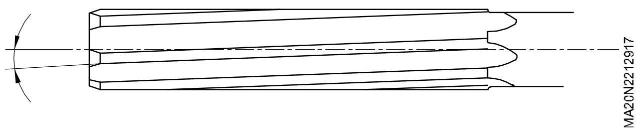
Fig 6



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 7)

Hand reamers

Objectives : At the end of this lesson you shall be able to
- **state the general features of hand reamers**
- **identify the types of hand reamers**
- **distinguish between the uses of straight fluted and helical fluted reamers**
- **name 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.

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 1

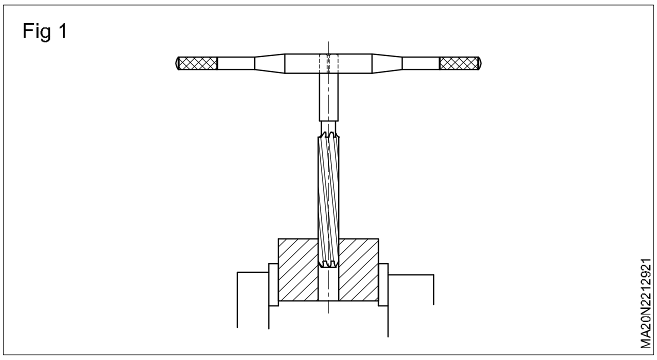
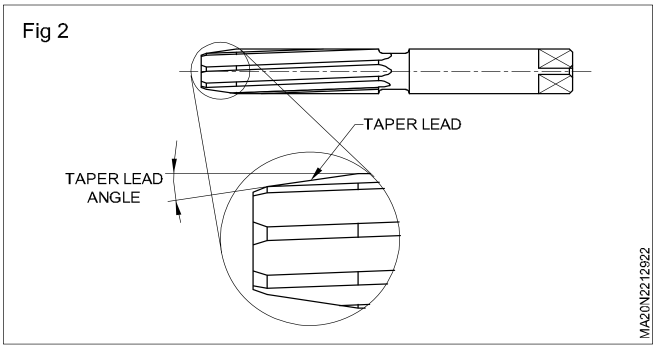


Fig 2

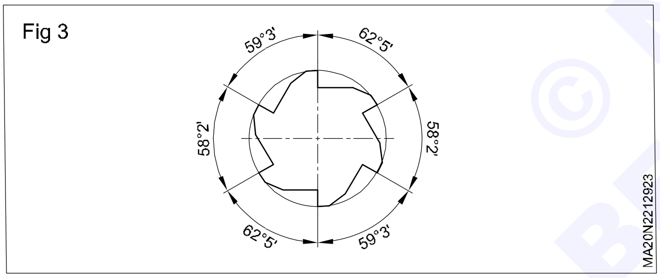


Most hand reamers are for right hand cutting.

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.

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 4

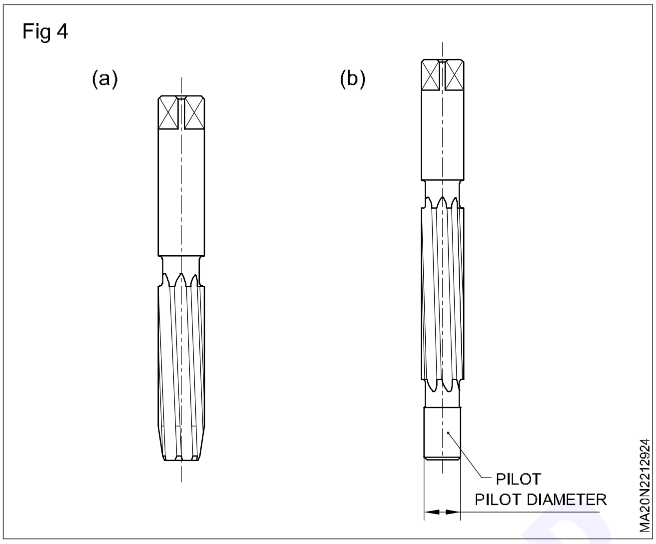
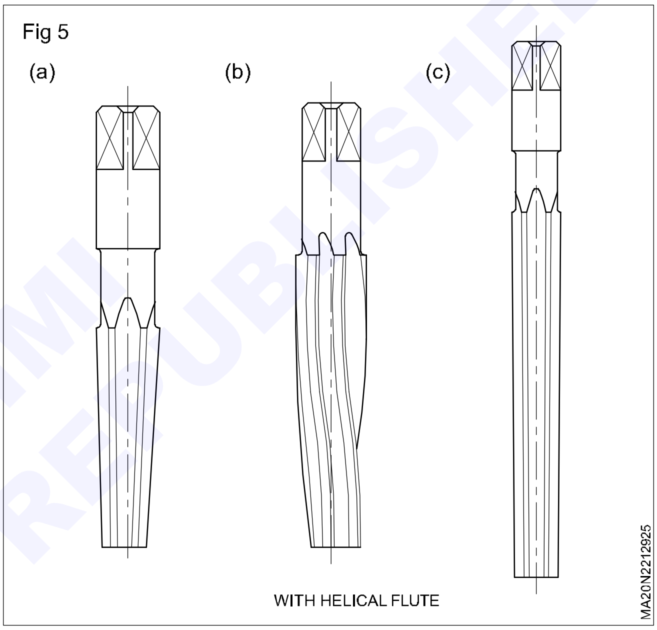
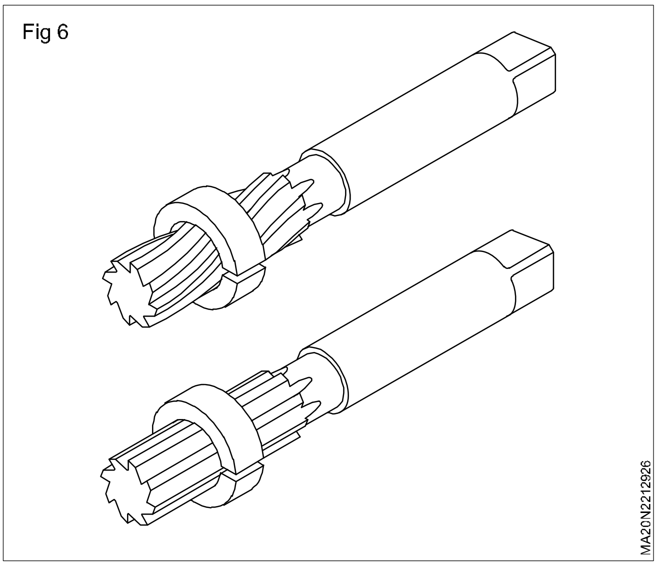


Fig 5



Socket reamer with parallel shank (Fig 6)

Fig 6



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.

Taper pin hand reamer

This reamer has tapered cutting edges for reaming taper holes to suit taper pins. A taper pin reamer is made with a taper of 1 in 50. These reamers are available with straight or helical flutes.

Use of straight and helical fluted reamers

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.

Specifications of a reamer

To specify a reamer the following data is to be given.

Type

Flute

Shank end

Size

Example

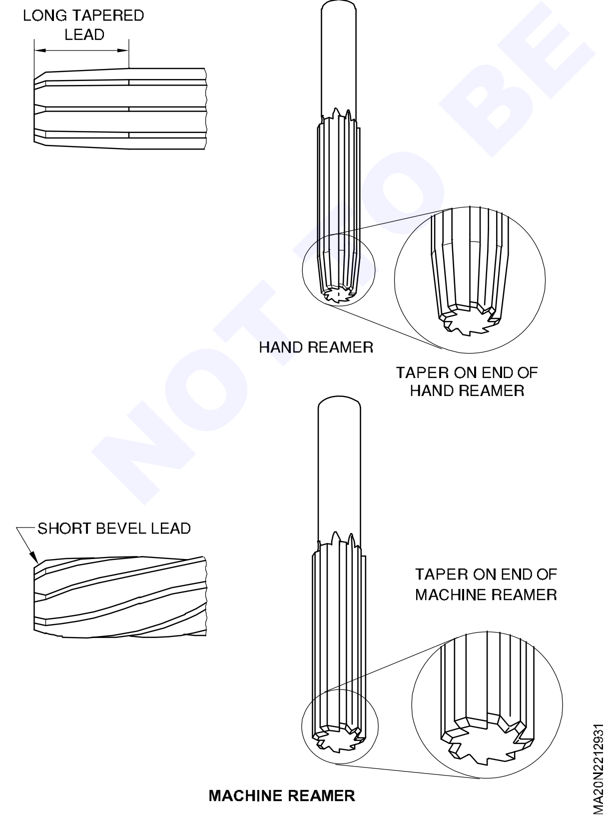
Hand reamer, Straight flute, Parallel shank of ∅ 20 mm.

Machine reamers

Objectives : At the end of this lesson you shall be able to
- **distinguish between the features of a hand reamer and a machine reamer**
- **name the different types of commonly used machine reamers, and explain them.**

Reamers are used to finish previously drilled holes accurately to size and 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.

Fig 1



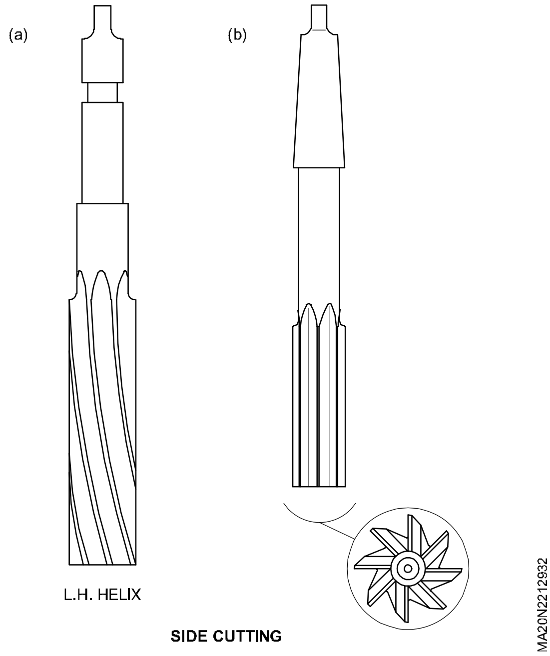
The shank end of the hand reamers will be square to facilitate reaming using tap wrenches. Machine reamers of small diameters will have parallel shank and the larger reamers are provided with taper shanks.

Types of machine reamers

Solid fluted machine reamer (jobber reamer)

This is identical 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 (2a))

Fig 2



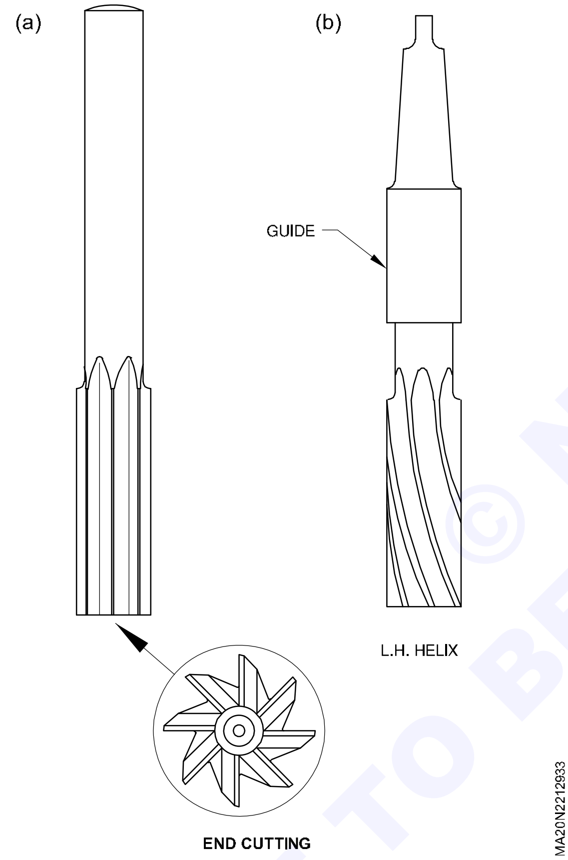
Chucking reamers (Fig (2b))

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.

Rose reamer/rose chucking reamer (Fig (3a))

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 3



Machine jig reamers (Fig (3b))

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.

Reamers of this type are available with rear guides only or with front and rear guides. (Fig 4) They can produce very accurate holes if the spindle and bushes are aligned accurately. While reaming deep holes it is better to select the jig reamers with guides on both ends.

When reaming is carried out immediately after drilling in the same setting, renewable bushes are used on the jig. (Fig 5)

Fig 4

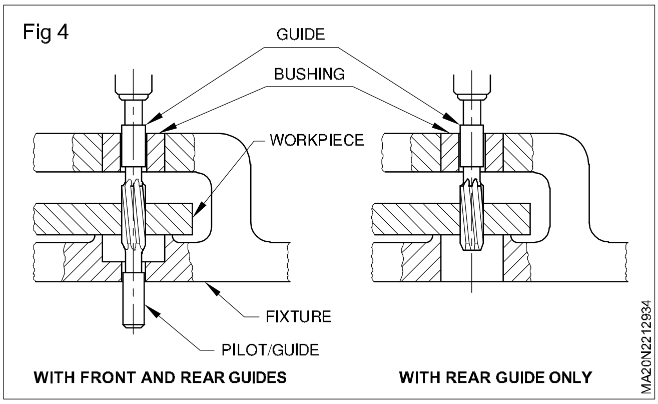
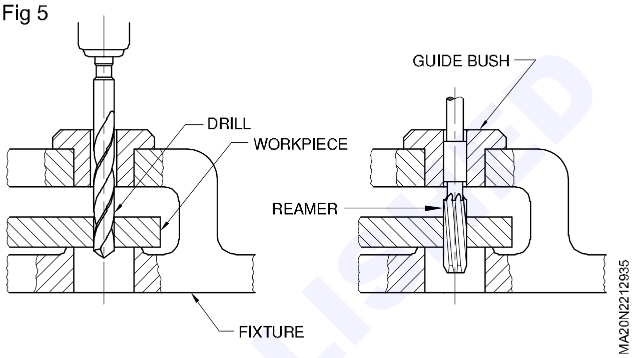


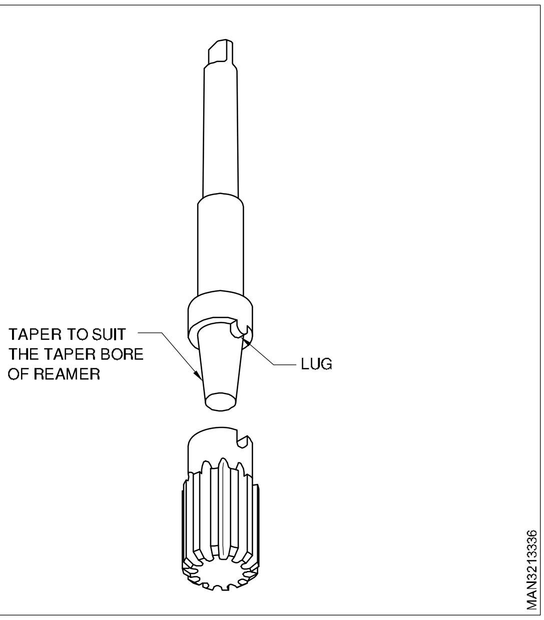
Fig 5



Shell reamer (Fig 6)

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.

Fig 6

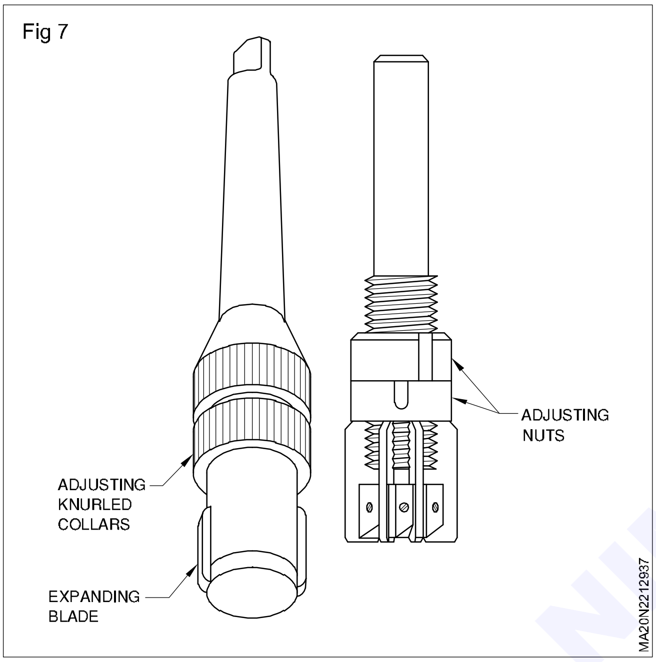


Adjustable machine reamer

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.

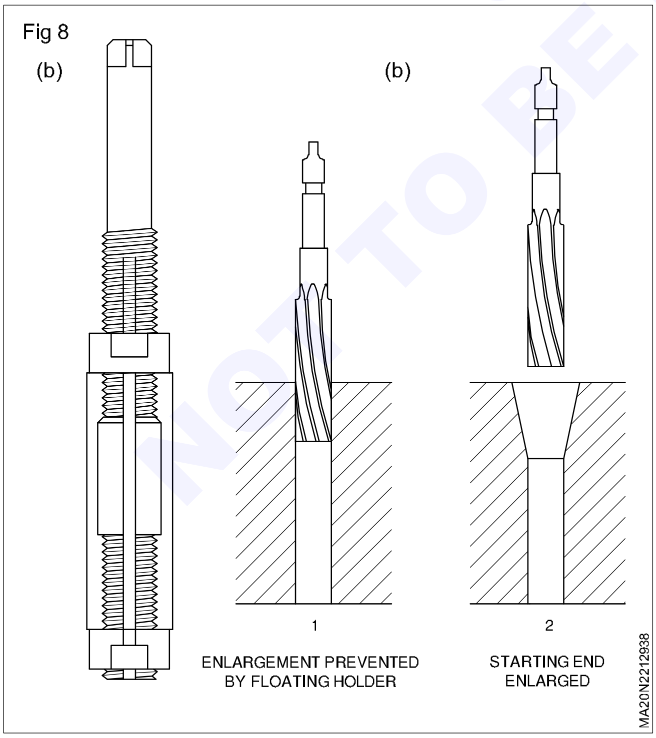
Adjustable reamers have adjustable insert blades. (Fig 7) When worn out or damaged, the blades can be easily replaced. These reamers are not meant to produce holes of different sizes. However, they can be used to increase the holes slightly.

Fig 7



Similarly, adjustable reamers are also available for hand reaming. (Fig 8a)

Fig 8

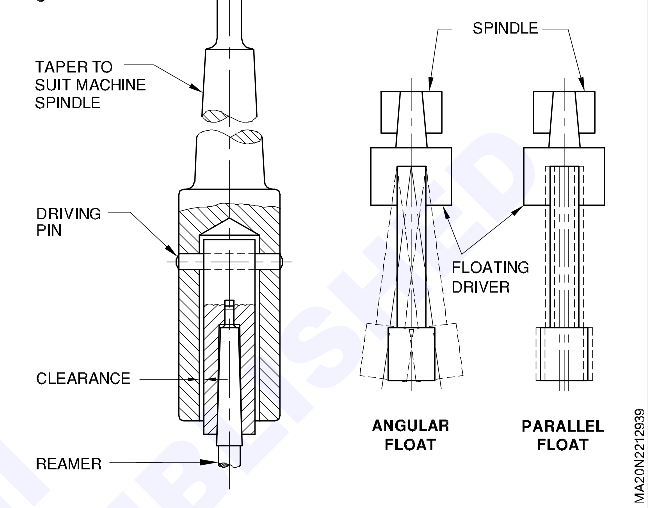


The size of the reamer can be adjusted by moving the blades in the tapered using the nuts provided on both end of the blade.

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 8b -2) This can be avoided by the use of reamers with floating holders (Fig 8b-1). Floating holders compensate minor discrepancies in the axis alignment. Floating holders are available with angular floats and parallel floats. (Fig 9)

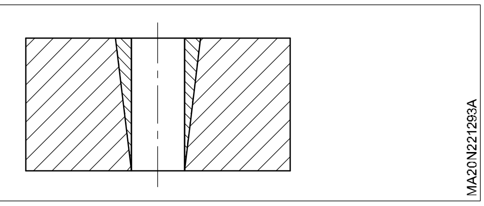
Fig 9



Taper reamers

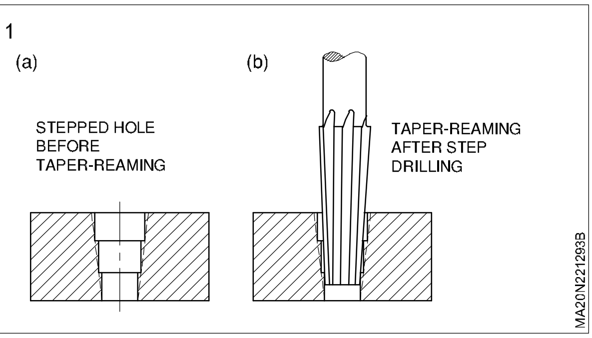
Taper machine reamers are manufactured in all standard tapers and with tapered shank. They can be mounted directly in the spindle of the machine. For taper reaming the diameter of the hole drilled is slightly smaller than the finished diameter of the small end of the taper. While reaming, the taper reamer will have to remove more material at the big end and less material at the small end. (Fig 10) While cutting, the entire length of the reamer will be in contact with the workpiece centres . This can cause chatter marks and poor finish.

Fig 10



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 11)

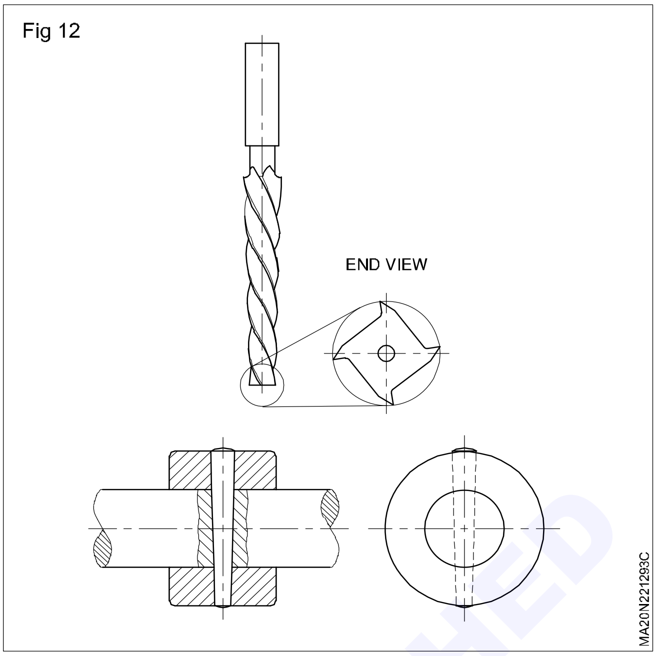
Fig 11



Taper pin machine reamers (Fig 12)

These reamers are used for reaming taper holes needed for fitting taper pins.

Fig 12



Calculation for cutting reamer

Objectives : At the end of this lesson you shall be able to
- **describe the calculations for cutting reamer**
- **explain the procedure to cut helical flute reamer.**

Calculations for cutting reamer

Reamer Flutes are in the form of straight or helical. For cutting Helical fluted Reamer.

Calculate Lead of helix, swivel of table and gear ratio to cut a helix.

It is necessary to move the work Longitudinally and rotate simultaneously.

Reamers are available in helix of (5 - 10º) normally.

Example

Calculate to cut a six fluted of 10º Helix angle for 20 mm Dia Hand Reamer. The available standard change gears are with number of teeth 24, 24, 28, 32, 40, 44, 48, 56, 64, 72, 86 and 100. (1 set of 12 gears)

Dia = 20 mm

Helix angle = 10º

tan10º = 0.1763

$$\text{Lead of helix} = \frac{\pi D \text{ (circumference of work)}}{\text{Tangent of helix angle } (\tan 10^\circ = 0.1763)}$$

$$\text{Lead of helix } = \frac{3.142 \times 20}{0.1763}$$

$$= 356.43 = 360 \text{ mm}$$

$$\text{Change gear ratio} = \frac{\text{Lead of Machine}}{\text{Lead of helix}} = \frac{\text{Driver}}{\text{Driven}}$$

Lead of Machine = Pitch of table lead screw x ratio of index head

Ratio of index head = 1 : 40

= 5 x 40

= 200 mm

$$\text{Change gear ratio} = \frac{200}{360} = \frac{200}{360} = \frac{5}{9}$$

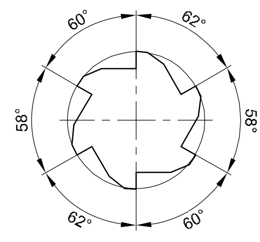
$$\frac{\text{Driver}}{\text{Driven}} = \frac{5 \times 8}{9 \times 8} = \frac{40}{72} = \frac{\text{gear on spindle}}{\text{gear on worm shaft}}.$$

As both 40 tooth and 72 tooth gears are available in standard set of change gears. A simple gear train may be used.

Procedure to cut helical flute reamer

Turn the required diameter Reamer blank and mount the blank between centres on universal milling machine table. Fix un equal double angle cutter or suitable Reamer flute form cutter. Most Reamer having uneven spacing of teeth. (Fig 1)

Fig 1



Rule for angular indexing

To find the index crank movement, divide the angle by 9 if it is expressed in degrees or by 540 if it is expressed in minutes or by 32400 if it is expressed in seconds.

Index crank movement =

$$\frac{\text{Angular displacement of work in degree}}{9}$$

$$\text{OR} = \frac{\text{Angular displacement of work in minute}}{540}$$

$$\text{OR} = \frac{\text{Angular displacement of work in seconds}}{32{,}400}$$

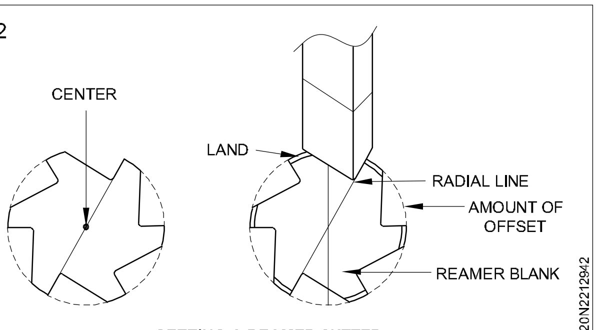
$$\text{Turns of index crank} = \frac{40}{N} = \frac{40}{6} = 6\frac{4}{6} = 6\frac{2}{3} = 6\frac{12}{18}$$

General Rule for angular indexing, one rotation covers 9º. In 18 hole circle between hole angle covers 30 minute or 0.5 degree.

62º - 58º - 60º - 62º - 58º - 60º

First position the Milling Cutter seats centre line of Reamer blank cut first (Fig 2) groove and Index for cutting second groove Turn 6 full rotation and 16 holes in 18 hole circle.

Fig 2



SETTING A REAMER CUTTER

For cutting Third groove Turn 6 full rotation and 8 holes in 18 hole circle.

For cutting Fourth groove Turn 6 full rotation and 12 holes in 18 hole circle.

For cutting Fifth groove Turn 6 full rotation and 16 holes in 18 hole circle.

For cutting Sixth groove Turn 6 full rotation and 8 holes in 18 hole circle.

For Return to First groove Turn 6 full rotation and 12 holes in 18 hole circle.

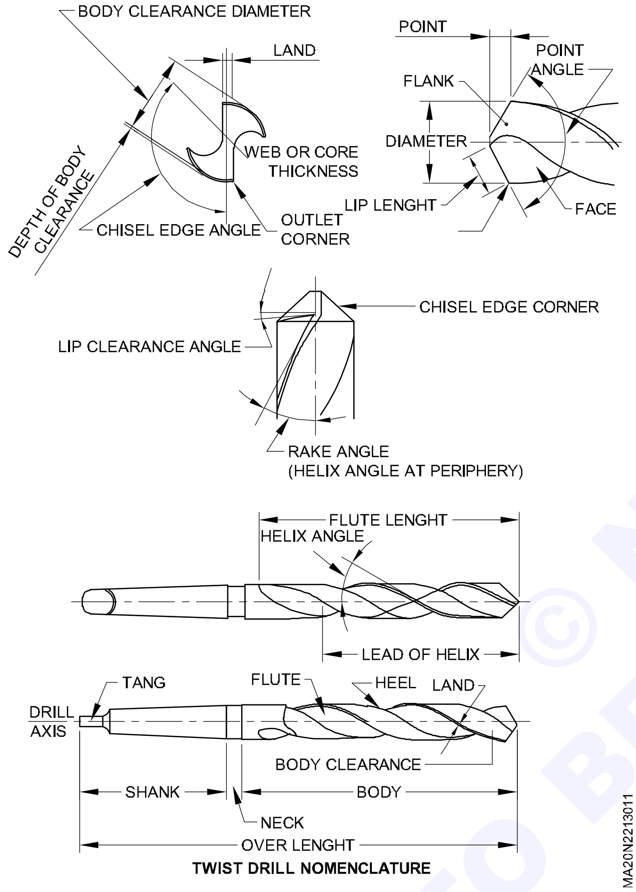
Twist drill - nomenclature

Objective : At the end of this lesson you shall be able to
- **list the elements of drills, and nomenclature of drill.**

Twist drill nomenclature

The following are the nomenclature, definitions and functions of the different parts of drill illustrated in Fig 1.

Fig 1



TWIST DRILL NOMENCLATURE

Twist drill elements

The following are the twist drill elements.

Axis

The longitudinal centre line of the drill.

Body

That portion of the drill extending from its extreme points to the commencement of the neck, if present, otherwise extending to the commencement of the shank.

Body clearance

That portion of the body surface which is reduced in diameter to provide diametric clearance.

Chisel edge

The edge formed by the intersection of the flanks. The chisel edge is also sometimes called dead centre. The dead centre or the chisel edge acts as a flat drill and cuts its own hole in the work piece. A great amount of axial thrust is required to cut a hole by the chisel edge. In some drills chisel edge is made spiral instead of a straight one. This reduces the axial thrust and improves the hole location. Chances of production of oversize holes is also reduced.

Chisel edge corner

The corner formed by the intersection of a lip and the chisel edge.

Face

The portion of the flute surface adjacent to the lip on which the chip impinges as it is cut from the work.

Flank

That surface on a drill points which extends behind the lip to the following flute.

Flutes

The groove in the body of the drill which provides lip

The functions of the flutes are

1 To form the cutting edges on the point.
2 To allow the chips to escape.
3 To cause the chips to curl.
4 To permit the cutting fluid to reach the cutting edges.

Heel

The edge formed by the intersection of the flute surface and the body clearance.

Lands

The cylindrically ground surface on the leading edges of the drill flutes. The width of the land is measured at right angle to the flute helix. The drill is full size only across the lands at the points end. Land keeps the drill aligned.

Lip (cutting edge)

The edge formed by the intersections of the flank and face. The requirements of the drill lips are:

1 Both lips should be at the same angle of inclination with the drill axis, 59° for general work.
2 Both lips should be of equal length.
3 Both lips should be provided with the correct clearance.

Neck

The diametrically undercut portion between the body and the shank of the drill. Diameter and other particulars of the drill are engraved at the neck.

Outer corner

The corner formed by the intersection of the flank and face.

Point

The sharpened end of the drill, consisting of all that part of the drill which is shaped to produce lips, faces, flanks and chisel edge.

Right hand cutting drill

A drill which cuts when rotation is counter-clockwise direction viewed on the points end of the drill.

Shank

That part of the drill by which it is held and driven. The most common types of shank are the taper shank and the straight shank. The taper shank provides means of centering and holding the drill by friction in the tapered end of the spindle.

Tang

The flattened end of the taper shank intended to fit into a drift slot in the spindle, socket or drill holder. The tang ensure positive drive of the drill from the drill spindle.

Web

The central portion of the drill situated between the roots of the flutes and extending from the points toward the shank; the point end of the web or core forms the chisel edge.

Drill angles

Following are the drill angles which are ground on a twist drill for efficient removal of metal.

Chisel edge angle

The obtuse angle included between the chisel edge and the lip as viewed from the end of the drill. The usual value of this angle varies from 120° to 135°

Helix angle or rake angle

The helix or rake angle is the angle formed by the leading edge of the land with a plane having the axis of the drill. If the flute is straight, parallel to the drill axis then there would be no rake. if the flute is right handed then it is positive rake; and if it is left handed then the rake is negative. The usual value of rake angle is 30° although it may vary up to 45° for different materials. Smaller the rake angle. greater will be the torque required to drive the drill at a given feed.

Point angle

This is the angle included between the two lips projected upon a plane parallel to the drill axis and parallel to the two cutting lips. The usual point angle is 118° but for harder steel alloys, the angle increases.

Lip clearance angle

The angle formed by the flank and a place at right angles to the drill axis. The angle is normally measured at the periphery of the drill. Lip clearance is the relief that is ground to the cutting edges in order to allow the drill to enter the metal without interference. The lip clearance angle should increase towards the centre of the drill than at the circumference. This is due to the fact that different points on the drill cutting edge follow different helical paths. Any points on the cutting edge at the circumference moves through a smaller helical angle than a point on the cutting edge near the centre. This happens to be such due to the lead of the helix being same in each case and hence the clearance angle given to the drill cutting edge should increase towards the centre. The clearance angle is 12° most cases. The clearance angle should be minimum to add rigidity and strength to the cutting edge.

Designation of drill

In Indian standard system, twist drills are designated by the series to which they belong, the diameter, the I.S number and the material of the drill. Unless mentioned in the designation, it should to presumed that the drill type is N and point angle 118°. The drills are made in three types, namely, normal (N), hard (H), and soft (S). These designation are based on the material to be cut and design requirements of the drill. Thus a parallel shank twist drill of long series, 10 mm dia, conforming to I.S standard, made of carbon steel, of type S and point angle 80° is designated as: Parallel shank dwell twist drill (Long) 10.00-IS : 599-CS-S80.

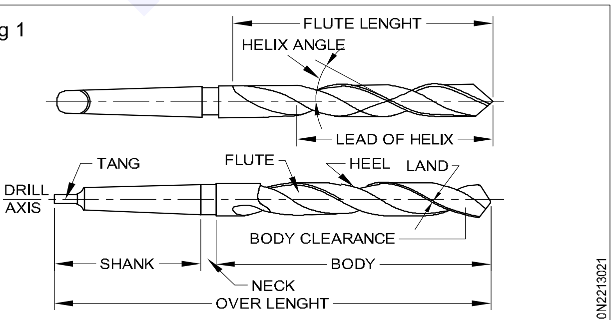
Twist drill - cutter selection & calculation

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

- **determine the size of the drill flute cutter.**

Drill flute cutter

Fig 1



TWIST DRILL NOMENCLATURE

Drill flute cutter size

Cutter No.	Over (Inch)	To (Inch)
1	1/8	3/16
2	3/16	5/16
3	5/16	7/16
4	7/16	11/16
5	11/16	1
6	1	1-1/2
7	1-1/2	2-1/8
8	2-1/8	3

Drill bit dimensions (are in mm)							Cutter Dimensions (are in inch)			
Diam D	Total Length	No. of Morse	Flute length	Web thickness	Width of land	Total clearance ZT	W	V	R	F
12	194	1	125	1.6	1.5	1.8	0.40	0.262	0.37	0.229
15	220	2	145	2.3	1.6	1	0.50	0.378	0.432	0.297
19	245	3	160	2.5	1.9	1.2	0.60	0.394	0.506	0.396
25	270	4	175	3.5	2.2	1.4	0.80	0.525	0.675	0.475

Calculation for drill bit

- Length of job (L) = 27 mm
- Diameter of job (D) = 25 mm
- Morse taper shank (no.4)
- Lead of machine (40 T_1) = 40 x 5 = 200 mm
- Whole depth = 0.44 x D 0.44 x 25 = 11 mm
- Offset for rake (0.635 mm/3.175mm) x D = (0.635/ 3.175) 25 = 5 mm
- Helix angle β(R/H)=25º

$$\text{Gear ratio} = \frac{Dr}{Dn} = \frac{\text{Lead of machine}}{\text{Lead of job}}$$

$$= \frac{200}{175} = \frac{200}{7 \times 25}$$

$$= {}^{8}\!/_{7} = \frac{64}{56}$$

- Indexing = 40/N = 20
- Flute length = 175 mm

Safety in CNC turning centers

Objectives : At the end of this lesson you shall be able to
- **state the points for personal safety and the material handling**
- **state the safety points for the safe machine operation on CNC Turning centers.**

1 General Safety

- Always be ALERT while machining on CNC machine.
- Know / Familiarize yourself with the machine before attempting to set (and) operate.
- First know all the emergency switches location before proceeding with operation.
- Do not press any switch / button / key unless you know fully well about the function of the same.
- Do not poke your head inside the machine when the auxiliary is on or in auto mode for any inspection /setting.
- Main switch should be OFF during cleaning the machine.
- Do not use compressed air for cleaning.
- Do not remove / adjust any safety / limit / proximity switches.
- Do not use nonstandard tools/holders.

Safety check on the machine - before start

- Check the voltage and current.
- Check the lubricant tank oil level.
- Check the hydraulic tank oil level.
- Check the clamping devices.
- Check the clean of the machines.
- Check the clamping stock.

Safety check on the machine - after start

- Check the main pressure of the hydraulic/pneumatic system.
- Check the clamping pressure of the hydraulic system for chuck / fixture / tail stock
- Check the chuck function.
- Lubricate the clamping device if any manual.
- Move the slides to Ref. points in return mode.
- Jog the slides to and fro and check the sliding movement.
- Check the centralized lubricant system by pressing app. Switches.
- Check the coolant supply by pressing app. switches.
- Check the selectable zero offset entries.
- Check the tool offset entries.
- Check the program and the app. Zero and tool offset.
- Check the Feed override position.
- Check the speed override position.
- Check for the alarm if any and set it right.
- Check the tool fits.
- Check the tools position with reference to program.
- Check the tool indexing / tool care.

Safety while machining

- Check the program control levels.
- (Single block, Dry run, skip optional stop etc.)
- Be careful Dry run should not be activated unless it is warranted.
- While in auto if - speed, feed is missing input manually.
- If coolant is not activated through program, manually activated the coolant switches.
- Try to cut the job in single block when your are first time proving the program.
- Open the door at the end of machining preferably in Jog mode.
- Clean with brush, the tools and the job. Avoid compressed air for cleaning.

2 Safety - while working on machining centre

- First check the operations of the machine then try to machine a work piece, preferably a trail without job/tools is advisable to avoid unexpected accident.
- Check first thoroughly the work zero point, tool offsets etc. These two checks are very important.
- Check speed and feed in the program otherwise it may cause vibrations, loosening of the fixture etc. and end up in an accident.
- Avoid changing the factory set parameters; setting wrong and unknown parameter may damage the machine.
- Immediately after switching ON the power do not touch any key on the MDI panel until the position display appears on the screen. Some of the keys are dedicated to maintenance (or) other special operations.

- Unexpected pressing of these keys may end up abnormal condition.
- Operations and programming manuals are general which describes several machines function. Therefore some functions may not be available to your machines. Hence check the specifications.
- Some functions may be implemented at the request of the consignees. These functions may be referred with the machine manual.

3 **Safety - related to programming**

- **Coordinate system setting**

  Incorrect coordinate system setting may damage the machine tool, workpiece etc, establish the coordinate system correctly in the program with reference to the machine setting.

- **Position by nonlinear interpolation**

  When performing positioning by non linear, Interpolation between the start and end points. The tool path should be carefully confirmed before performing the program.

- **Function involving 4th axis**

  When programming the 4th axis pay careful attention to the speed of the rotation axis. Incorrect programming of the rotational axis may cause the check to loose the grip and caused damage the machine tool, tool etc.

- **Inch/metric conversion**

  Switching between inch and metric inputs does not convert the measurement units of data such as the work piece origin, offset, current position etc. Hence before starting the machining determine which measurement units are being to be used and check the values accordingly.

- **Stroke check**

  Perform the manual reference position return as required. Stroke check is not possible before manual reference position return. When the stroke check is disabled alarm is not issued even if the stroke limit is exceeded.

- **Tool offset check**

  If the called tool is not considering the tool offset value may cause heavy damage to the tool job,machine - hence check in the program the tool offset, CRC values etc.

- **Plane selection**

  The machine behaves unexpectedly in wrong movement if the plane selection is wrongly selected in the program.

Safe material handling

Materials handling equipment

Materials handling equipment is a mechanical equipment used for the movement, storage, control and protection/ protecting of materials, goods and products throughout the process of manufacturing, distribution, consumption and disposal.

Different types of material handling equipment

- Tools
- Vehicles
- Storage units
- Appliance and accessories

Racks

Pallet racks, drive - through or drive - in racks, push back racks, and sliding racks.

Truck / Trolley

Conveyor system

- Fork lift
- Cranes
- Pallet truck

Basic of CNC technology

Objectives : At the end of this lesson you shall be able to
- **describe basic of CNC Technology**
- **state the advantages of CNC Turning**
- **state the different between CNC and conventional machine**
- **concept of accuracy and repeatability.**

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

Evolution of automation

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

- Mechanisation started in 1870 at the beginning of industrial revolution with simple production machines.
- In 18th Century fixed automatic mechanism and transfer lines came into existence for faster output and shorter production time.
- Simple automatic control machines and copying machines were invented in the later part of the 18th century. After 1950 the industrial automation was started In this second phase of the industrial automation/ revolution, workers, instead of physically performing all the task are placed in the control of the machines.
- The extension of NC was computerised numerical control (CNC) machine tools in which computer (Micro Processor) is included as an integral part of the control system.
- The latest of the above is Computer Integrated Manufacturing (CIM) which includes battery of CNC machines, with flexible modules, for manufacturing tool head changers, automatic material handling system like AGV's etc with minimum number of operating personals.

Fundamentals of NC controls

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

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

Advantage of NC machine are

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

There are many disadvantage of NC system:

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

Applications of CNC

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

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

Present CNC technology

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

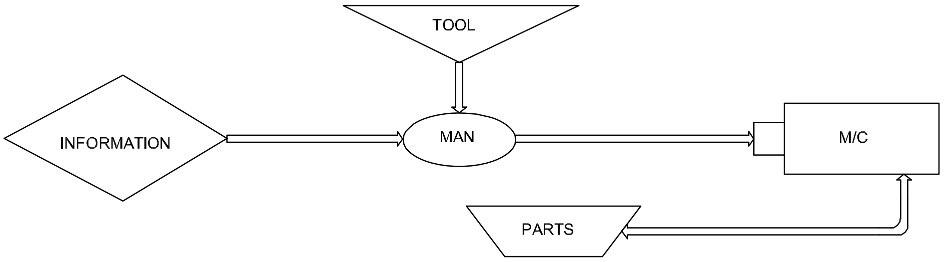
User defined parametric programming Standard Cycles like stock removal, drilling milling pattern etc are now a day's standard component of the systems.

Some latest controls are having "DOS" front end with CAD/ CAM facility in which one can design a component and get the Computer Assisted Part Programe (CAPP) and proving the component on the machine control itself without wasting much time and money. Modern machine tools have multi spindle with a spindle speed of 75000 rpm; Cutting feed rate of 5000 mm/min, and rapid traverse of 40,000 mm/min. Use of multi various sensing elements with adaptive controls remote diagnostics system makes the machine more versatile and free from accidents.

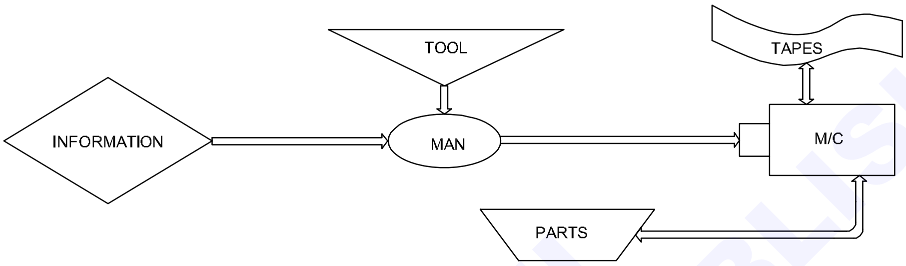
The silent and salient use of computers in factory automation and in factory management will boost the quality and quantity in production, which in turn will definitely change the lifestyle of the people in future.

Pictorial representation of conventional machine numerical control, auto tool changer and computer numerical control are shown below

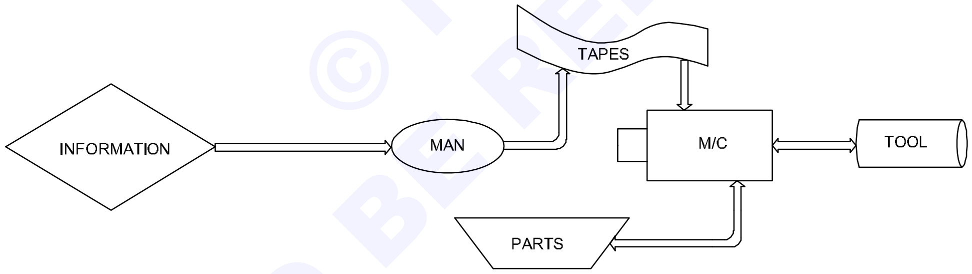
CONVENTIONAL



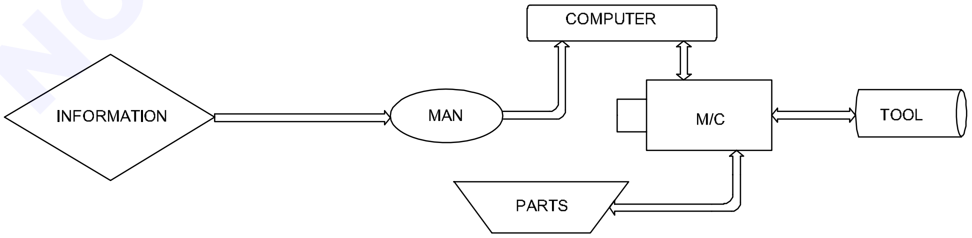
NUMERICAL CONTROL



AUTO TOOL CHANGERS



COMPUTER NUMERICAL CONTROL



MA20N2313111

Difference between CNC and conventional lathes

Objectives : At the end of this lesson you shall be able to
- **state the difference between conventional lathe and CNC lathe**
- **state the advantages of CNC machine.**

Difference between conventional and CNC

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

Advantages of CNC machines over conventional lathes

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

Disadvantages of CNC machines

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

Concept of positioning accuracy and repeatability

Two important specifications of CNC machines are Positional accuracy and repeatability.

Positional accuracy: The difference in the positional coordinates to which a machine actually moves and the theoretical coordinates. Example, you have commanded the machine to move to position X=10.58mm. When the machine moves, it actually reaches 10.57mm. So the positional accuracy in this case is difference between actual and theoretical values, i.e. 10.58-10.57=0.01mm. Generally, positional accuracy is not calculated by a single measurement. Multiple readings are taken when a machine moves to multiple programmed point and all deviations recorded. The statistical average value all these deviations is the positional accuracy of the machine. It is usually 0.020mm for normal CNC machines and can go up to 0.003mm for Jig Boring machines.

Repeatability: It is the reproducibility of the values when the machine is asked to move to a targeted point multiple times and the difference in various values is the repeatability. Eg you ask machine to move to target point 10.58mm, 5 times. the actual values to which the machine moves are 10.57, 10.568, 10.571, 10.567 and 10.572mm. Repeatability in this case is difference between to extreme positions, i.e. 10.572-10.567=0.005. In reality, repeatability is calculated using multiple readings over multiple points.

- Repeatability is more important than positional accuracy.
- Typically, for a CNC machine, Repeatability=0.5 x positional accuracy.

Positional accuracy and repeatability is measured separately for each individual axis of the machine (X, Y, Z etc.). This measurement is done by an instrument called Laser Interferometer.

Repeatability

The total deviation of positions achieved through repeated commands to the same point. Typical value is ±5 μ m.

Spindle nose runout

The eccentricity with which the spindle rotates about its own axis. Typical value is 10 μ m

Runout of revolving tailstock center

The eccentricity with which the center rotates about its own axis. Typical value is 15 μ m

Coaxiality between spindle and tailstock

The distance between the axes of the spindle and tailstock. Typical value is ± 5 μ m.

Perpendicularity of spindle axis to turret X movement

Typical value is ± 5 μ m over 300 mm.

Parallelism of spindle axis to turret Z movement

Typical value is total deviation of 20 μ m over 300 mm.

Control system makes

The machine and the control system are usually manufactured by different companies. Machine manufacturers make only the machines and buy the control system from other Companies which only make the control system, motors and drives.

Fanuc - Fujitsu, Japan

Sinumeric - Siemens, Germany ,Acramatic - Vickers,USA

Allen - Bradley - Allen Bradley, USAGE - General Electric, USA

Mazatrol - Mazak, Japan

CNC lathe machine elements and its functions

Objectives : At the end of this lesson you shall be able to
- **state the CNC lathe and its type**
- **list the part of the CNC machine**
- **state the functions of the parts in CNC machine.**

CNC lathe

Computerized numerically controlled, or CNC lathe machines are most commonly used because they are fast, accurate and the most advanced type of lathe. CNC lathes have at least the ability to drive the cutting tool under g-code control over 2 axes, referred to as X and Z. They may have a considerable amount of other functionality as well and there are many variations on lathes .

Types of CNC lathe machine

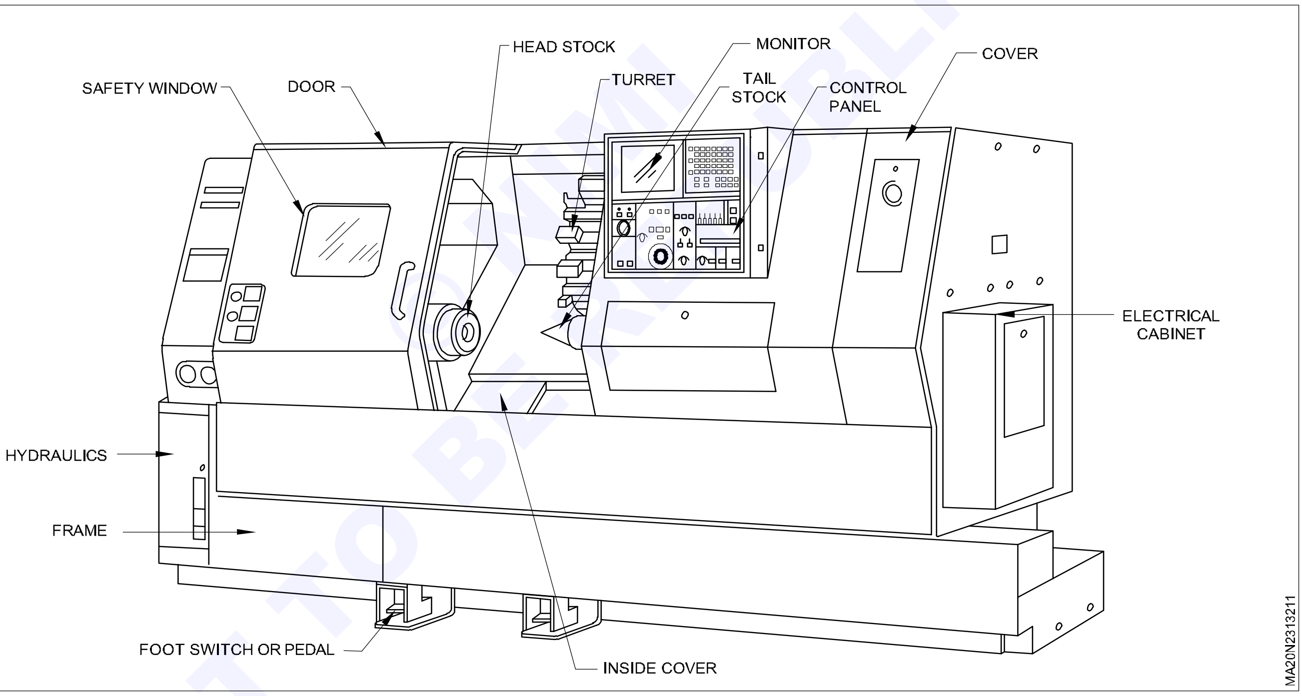
The types of CNC lathe can be determined by the number of movement axes:

- 2-axis - can perform outer/inner diameter machining, facing, drilling, and tapping.
- 3-axis - adds a live tool system for milling, boring, etc.
- 4-axis - typically adds a second tool carrier or turret.
- 5 & 6-axis - adds an additional Y, and occasionally a B-axis, to either or both of the tool carriers.

Specialised CNC lathes with more than 6 axes are called Multi-Spindle Machines and these are also available with advanced features and flexibility.

Parts of CNC machine is shown in Fig 1.

Fig 1



Bed (Fig 2)

The bed is the casting which has the guideways for the Z axis. Beds can be flats or slant. A slant bed aids chip disposal, since the chips fall down to the bottom due to gravity. A conveyor system for removing chips in CNC machine. It is used to convey material (Chips) from one end to another. Conveyor systems are required in all the field.

The slant bed of a CNC turning centre

Work holding in CNC turning centre

A three-jaw power-actuated chuck in a CNC turning centre. This setup is using soft jaws that were machined to match the outside diameter of the work piece. (Fig 3)

Fig 2

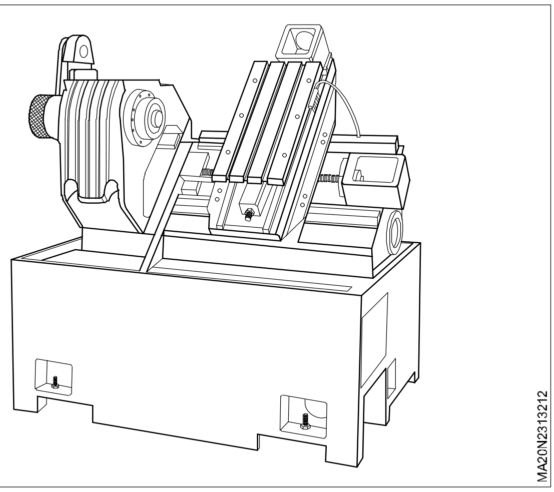
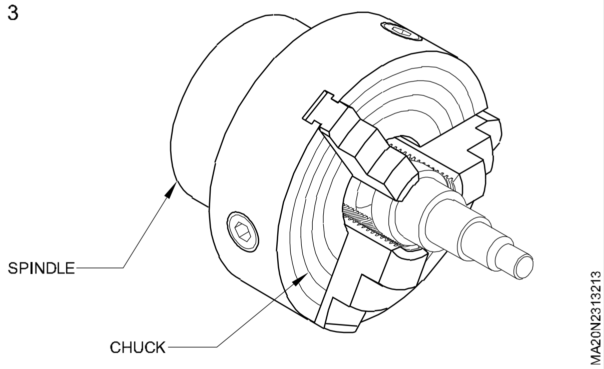
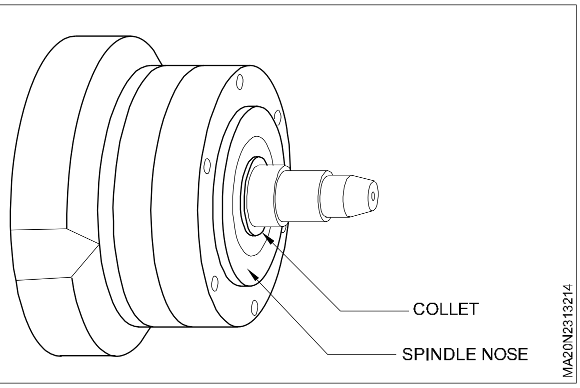


Fig 3



Collet chuck for work holding (Fig 4)

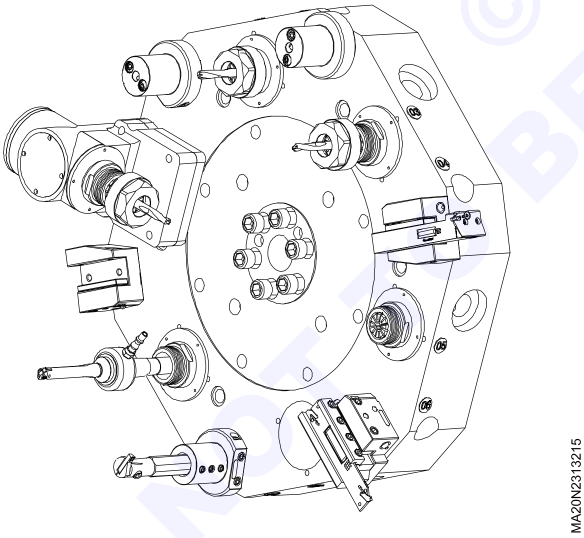
Fig 4



Turret

The turret typically has 8 or 12 tools mounted, each in its own station. When a tools is selected in the program, the turret rotates about its axis to bring the selected tool into the active position. (Fig 5)

Fig 5

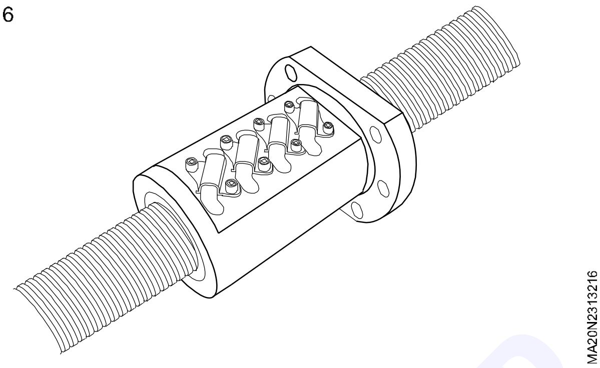


Ball screw (Fig 6)

The recirculating ball screw and nut system are widely used on CNC machine tools because of the following advantages.

- Low frictional resistance.
- Low drive power requirement.
- Little temperature rise.
- Less wear and hence longer life.
- No stick-slip effect.
- High traverse speed.
- High efficiency.

Fig 6



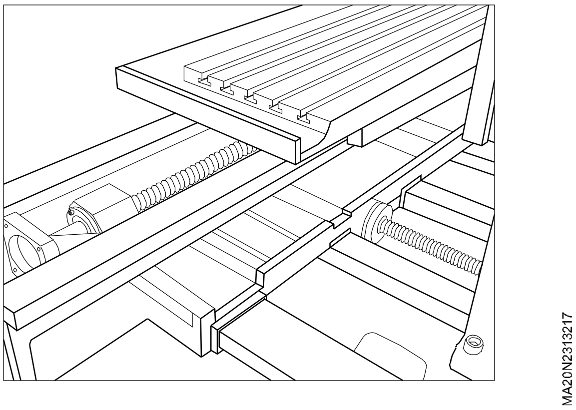
Guide ways

- To control the direction or line of action of the carriage or the table on which a tool or a work piece is held.
- To absorb all the static and dynamic forces.
- The shape and size of the work produced depends on the accuracy of the movement and on the geometric and kinematic accuracy of the guide way.
- The geometric relationship of the slide and the guide way to the machine base determines the geometric accuracy of the machine.
- Kinematic accuracy depends on the straightness, flatness and parallelism errors in the guide ways.

Guide ways are primarily two types:

Friction guide ways: (Fig 7)

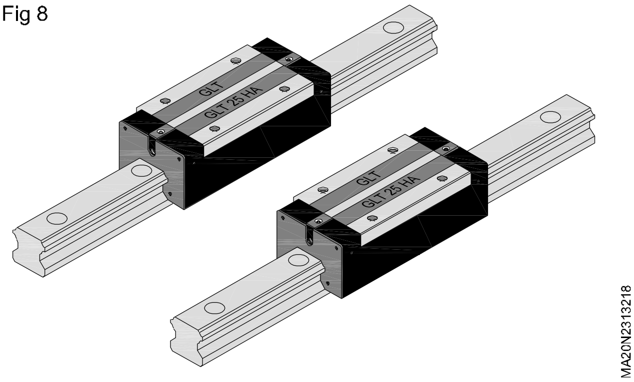
Fig 7



- They are most widely applied in conventional machine tools due to their low manufacturing cost and good damping properties.
- These guide ways operate under conditions of sliding friction and do not have a constant coefficient of friction.
- **Types :** vee guide ways, Flat guide ways, Dove tail guide ways, Cylindrical guide ways, Combination of flat & vee guide ways.

Anti-friction linear motion (LM) guide ways (Fig 8}

Fig 8



- These are used on CNC machine tools to Reduce the amount of wear.
- Improve the smoothness of the movement.
- Reduce friction & heat generation.
- Antifriction guide ways are used to over come the relatively high coefficient of friction in metal-to-metal contacts and resulting the limitations addressed in the above list.
- They use rolling elements in between the moving and the stationary elements of the machine.

Advantages of anti-friction guide ways:

- Low frictional resistance.
- No stick- slip.
- Ease of assembly.
- Commercially available in ready-to-fit condition.
- High load carrying capacity.
- Heavier preloading possibility.
- High traverse speeds.
- Lower damping capacity is the only main disadvantage.

Other guide ways:

- Hydrostatic guide ways: Very thin film of oil under high pressure separates the surface of the slide.
- Frictional wear and stick-slip are entirely eliminated.
- Aerostatic guide ways: The slide is raised on a cushion of compressed air which entirely separates the slide and guide way surfaces. E.g. Coordinate measuring machine (CMM).

Feed drives:

- On a CNC machine, the function of feed drive is to provide motion to the slide as per the motion commands.
- Since the degree of accuracy requirements are high, the feed drive should have high efficiency and response.
- It consists of servo motor and mechanical transmission system.

Servo motors:

- Commonly used feed drive motors for CNC machines are direct current (dc) servo motors and alternating current (ac) servo motors.“Initially DC servo motors and drives were used most commonly on all CNC machines. But later, Ac servos has become more popular due to cost comparison, less maintenance, better response and a higher reliability.

Controls, software and user interface:

- CNC controls are the heart of the CNC machines.
- The early CNC controls were developed for simple applications in turning, machining centers and grinding.
- But with increased capabilities on modern machine tools such as higher spindle speeds, higher rapid traverses and more no. of axes, CNC systems have been developed to meet these needs.
- Multi tasks - run one program while programming and simulating a second task which maximizes the m/c use.
- The new controllers offer advanced graphic interfaces, program simulation and some cutter selecting capabilities.

Spindle (Figs 9 & 10)

Spindle is the main part of the machine tool. It receives power from the drive unit and delivers it to the job/work piece.

Fig 9

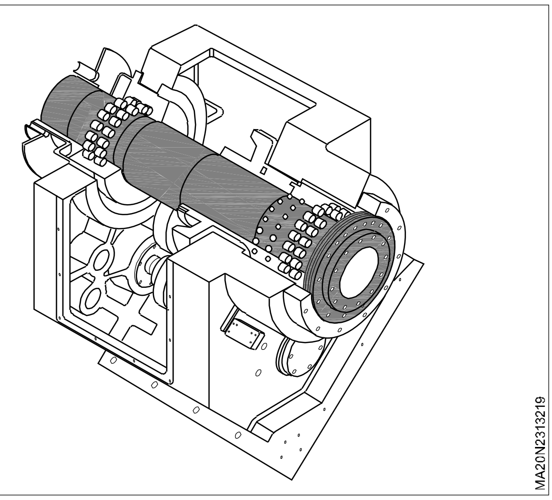
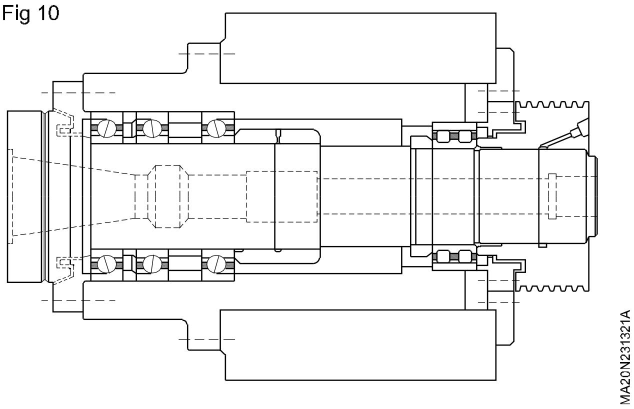


Fig 10



The function spindles are as follows

1 To deliver power to the job/tool.

2 To hold the job/tool

3 To centre the job/tool

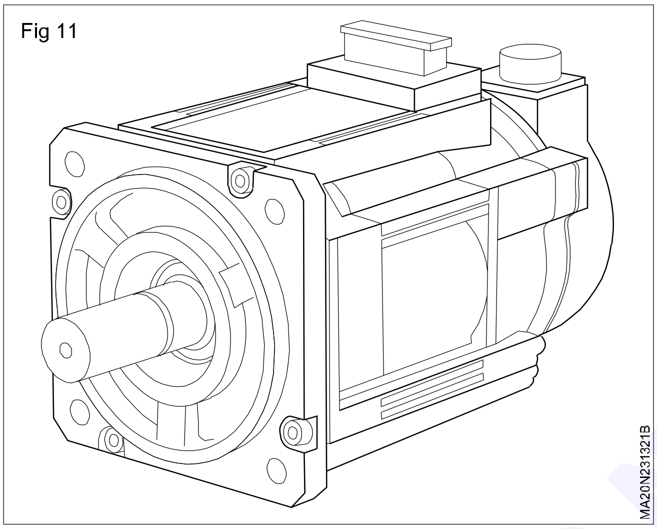
Types of spindle drive motors (Fig 11) - Normally more number of spindle drive motors are available. The following two are widely used

1 Separately excited DC.shunt motor, and

2 Three phase AC induction motor

Normally DC spindle drive motors are used in machine tools. But due to the availability of microprocessor based technology, AC drives are preferred. (Fig 11)

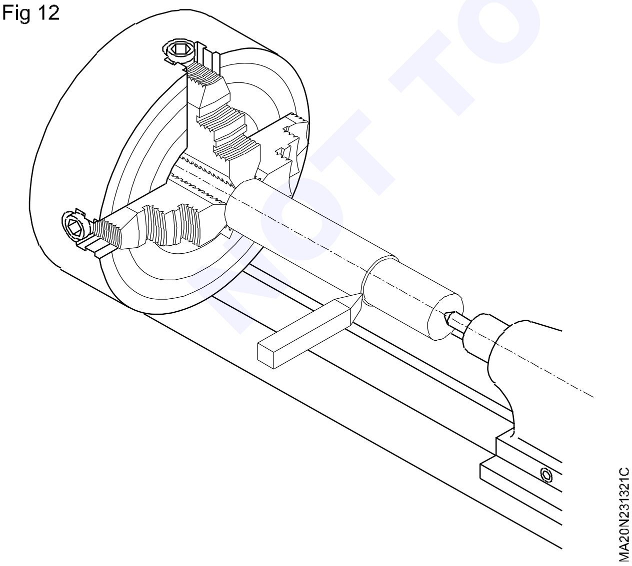
Fig 11



Tail stock (Fig 12)

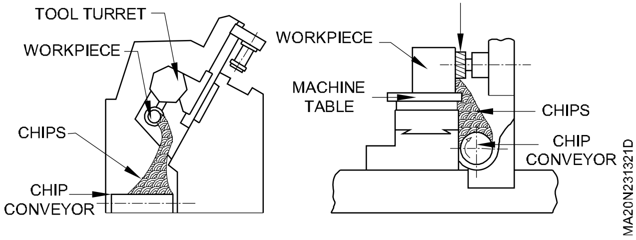
The tailstock holds the free end of long parts which cannot be held in the chuck alone. The quill can be moved in or out through program control on some machines. The body can also be moved through program control on some machines

Fig 12



Chip conveyors in CNC machine tools (Fig 13)

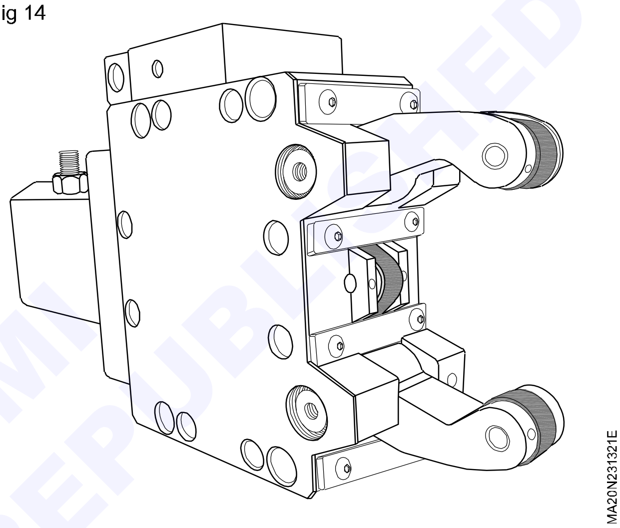
Fig 13



Steady rest (Fig 14)

The steady rest prevents long parts from deflecting when cutting forces are exerted. The steady can be opened and closed through proper arm control on some machines.

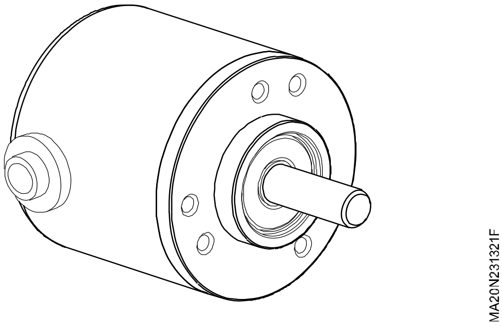
Fig 14



Encoder (Fig 15)

An encoder is a sensing device that provides feedback. Encoder convert motion to an electrical signal that can be read by some type of control device in a motion control system. The encoder sends a feedback signal that can be used to determine position, count, speed or direction.

Fig 15

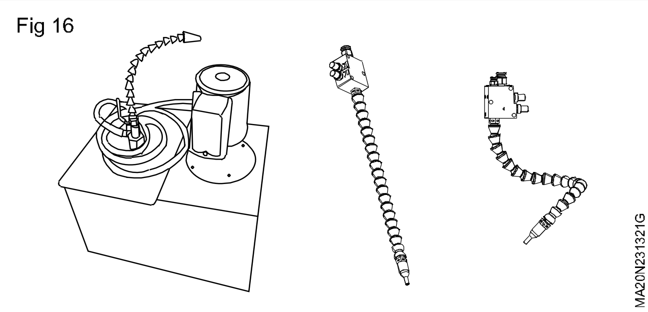


Coolant system (Fig 16)

Flood type coolant system

Coolant system provided with the machine is designed to supply the coolant through the flexion to the cutting zone.

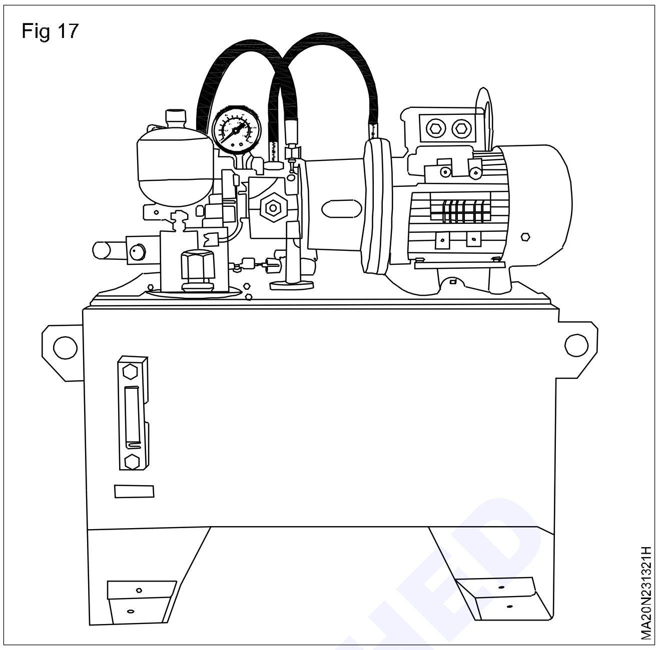
Fig 16



Hydraulic system of CNC machine (Fig 17)

CNC machine is such type of machine where several jobs are executed in compact area and in less time. In such scenario hydraulic system play a big role to exercise several functions hydraulic pressure in CNC machine is developed by hydraulic power pack which is a combination of several hydraulic components.

Fig 17



Feed back control system and interpolations

Objectives : At the end of this lesson you shall be able to
- **explain about feed back system**
- **state the closed loop and open loop control**
- **define the interpolation**
- **state the purpose of interpolation.**

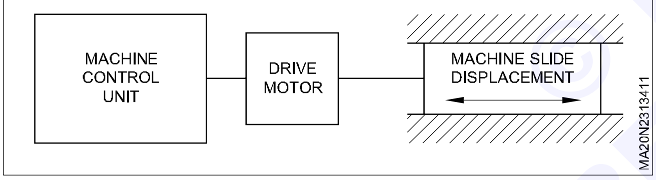
Feed back system

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

Open loop control system (Fig 1)

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

Fig 1



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

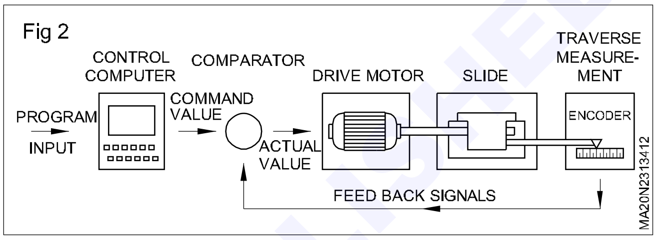
Closed loop control (Fig 2)

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

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

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

Fig 2



Interpolation

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

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

Definition

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

The parameters supplied may be

1 Radius
2 Start and end point of a curve
3 Radius and centre of a circle
4 Gradient angle for a line.

Types of interpolations

Interpolations are classified as

1 Linear interpolation
2 Circular interpolation
3 Helical interpolation
4 Parabolic interpolation
5 Logarithmic interpolation
6 Exponential interpolation of these the linear and circulate interpolators are commonly employed.

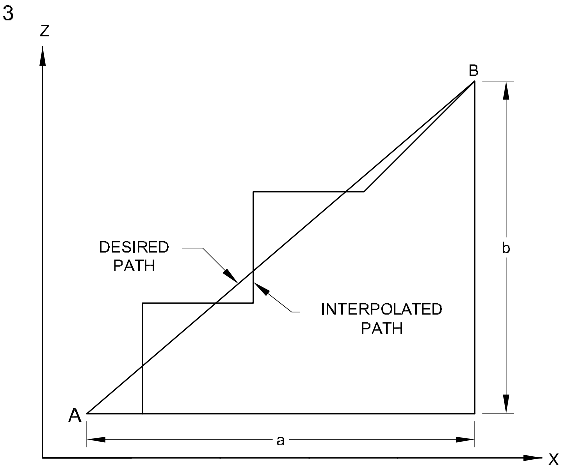
Linear interpolation

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

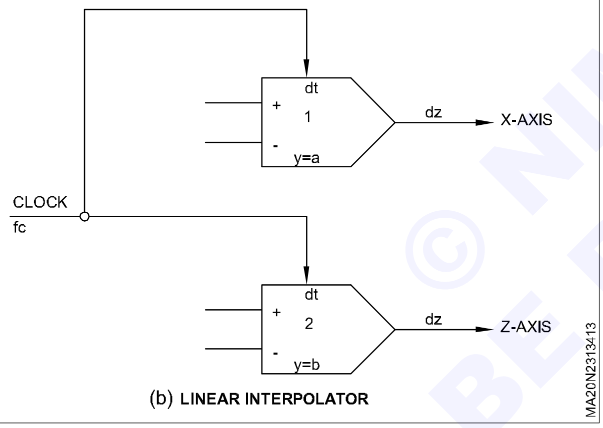
This is done in two or three dimensions.

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

Fig 3



(a) STRAIGHT LINE INTERPOLATION



(b) LINEAR INTERPOLATOR

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

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

Circular interpolation

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

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

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

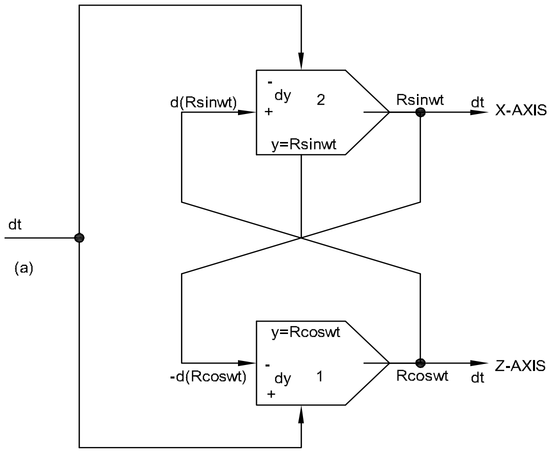
Two Digital Differential Analysers (DDA) are required for circular interpolation.

Advantages of circular interpolation are

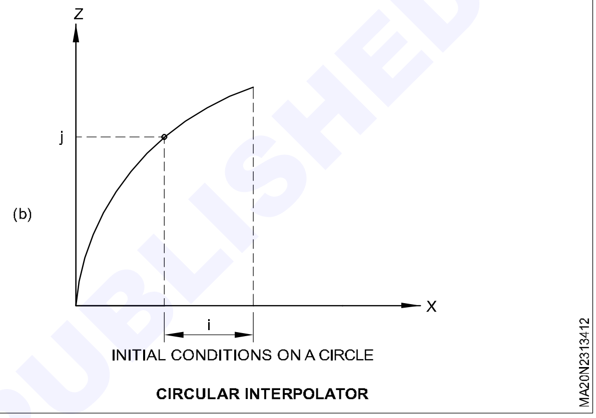
Better surface finish

Greater accuracy

Fig 4



LINEAR INTERPOLATOR



CIRCULAR INTERPOLATOR

Less total machining time

Lower working costs.

Circular interpolation

The circular interpolation

Code G02 (clockwise)

Code G03 (anti - clockwise)

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

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

The following assignment applies

I for the X - axis

K for the Z - axis

Circular Interpolation with mixed programming

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

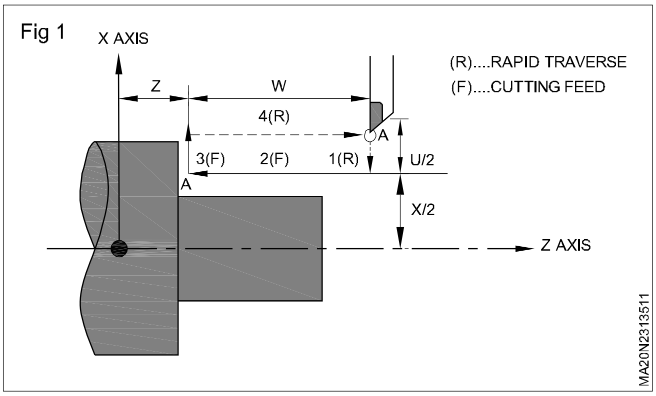
This is a prime example of where the mixed co-ordinate programming of the interpolation parameters in absolute dimensions comes in useful.

Machining operation and tool path

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

Straight cutting cycle (Fig 1)

Fig 1



Operations A straight cutting cycle performs four operations:

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

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

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

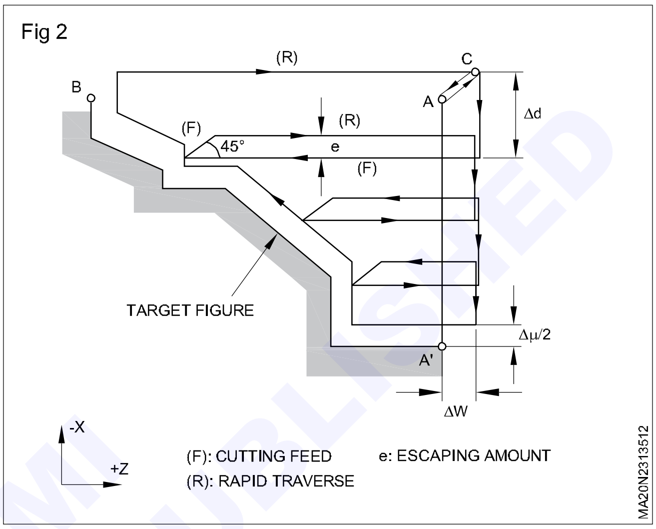
4 Operation 4 moves the tool to the start coordinate of the first axis on the plane (start Z-coordinate for the ZX plane) in rapid traverse. (The tool returns to the start point (A).

Stock Removal in Turning (Fig 2)

Operations

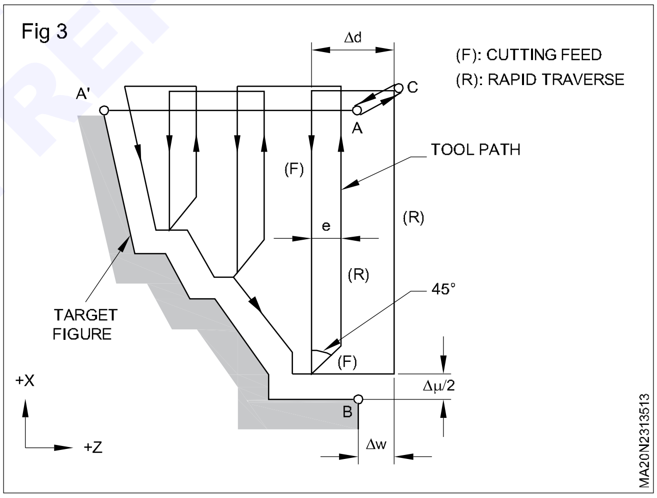
When a target figure passing through A, A', and B in this order is given by a program, the specified area is removed by △d (depth of cut), with the finishing allowance specified by △u/2 and △w left. After the last cutting is performed in the direction of the second axis on the plane (X-axis for the ZX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

Fig 2



Stock Removal in Facing (Fig 3)

Fig 3



Cutting path in stock removal in facing

Operations

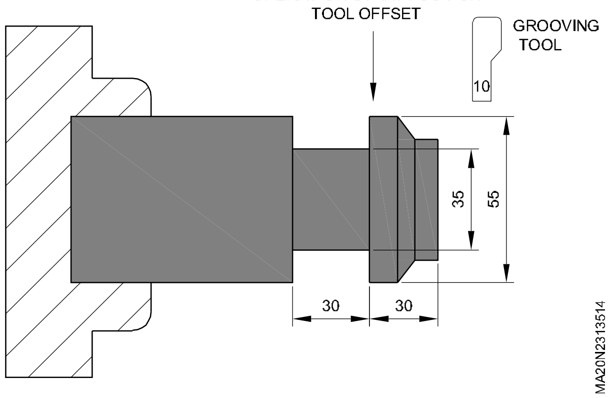
When a target figure passing through A, A', and B in this order is given by a program, the specified area is removed by △d (depth of cut), with the finishing allowance specified by △u/2 and △w left.

Grooving tool path (Fig 4)

Example program

N70 G75 R1 ;

Fig 4



N80 G75 X35 Z-60 P3000 Q9000 ;

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

Multiple Threading Cycle (Figs 5&6)

Fig 5

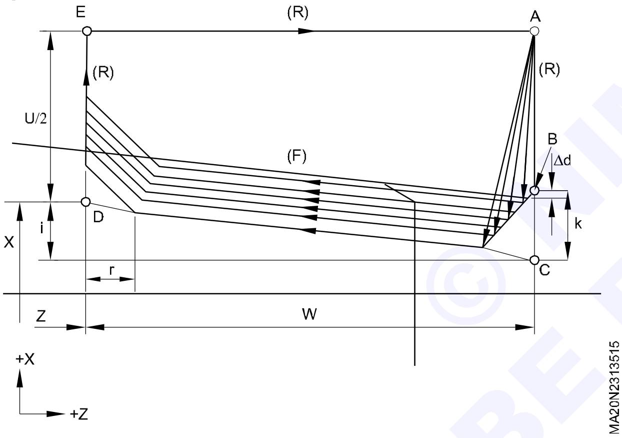
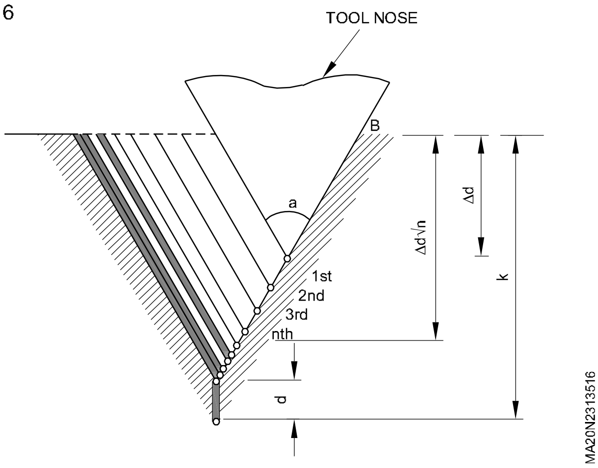


Fig 6



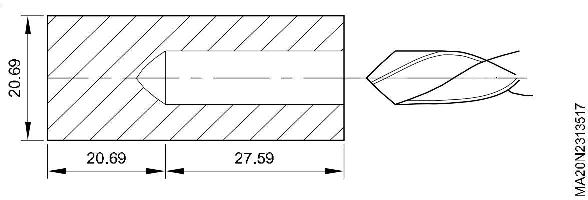
Operations

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

Face drilling (Fig 7)

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

Fig 7



Concepts of coordinate geometry, machine geometry and CNC zero points

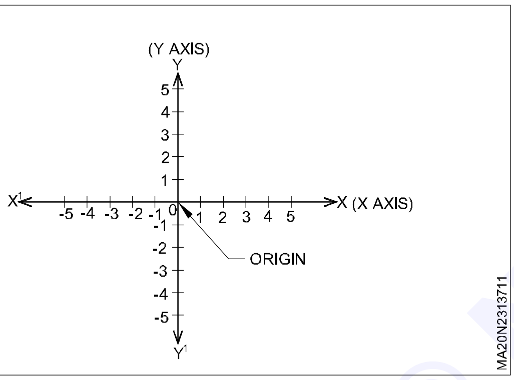
Objectives : At the end of this lesson you shall be able to

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

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

The concepts of Co-ordinate Geometry were developed by Rene Descartes. He is a French mathematician and philosopher and he established an association between algebraic equations and geometric curves. (Fig 1)

Fig 1



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

The horizontal number line XX1 is known as X-axis and the vertical number line YY1 is known as Y-axis.

The point where XX1 & YY1 intersecting each other is called the origin, and is denoted by 'O'.

In this plane the positive numbers lie on the directions OX is called the positive direction of the X-axis, similarly OY is the positive Y-axis respectively. Also the negative numbers lie on the directions OX1 is called the negative directions of the X-axis, similarly OY 1 is the negative Y-axis respectively.

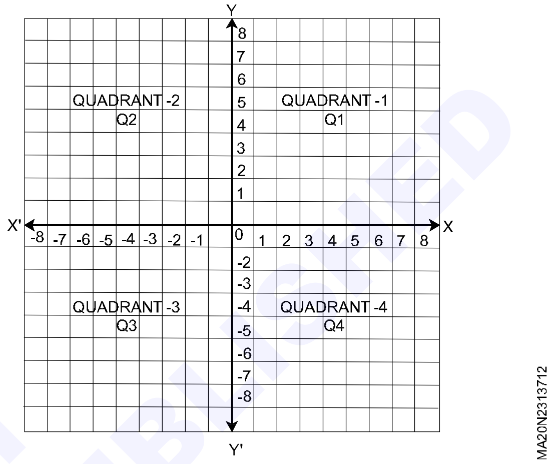
The plane here is known as the Cartesian plane or co-ordinate plane or XY-plane. The X & Y axes are known as coordinate axes.

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

Coordinates of a Point (or) Locating of coordinate points

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

Fig 2



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

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

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

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

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

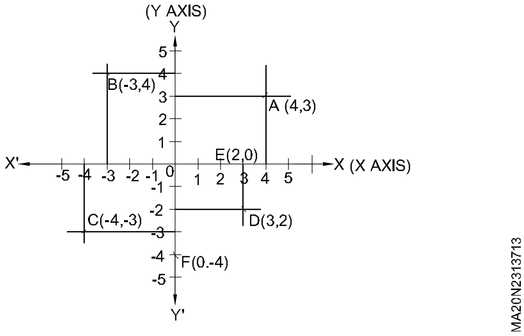
Y coordinate - The Y-coordinate of a point is the distance from origin to foot of perpendicular on Y-axis. The y-coordinate is also known as the ordinate.

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

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

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

Fig 3



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

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

Hence the coordinates of A are (4,3)

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

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

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

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

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

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

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

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

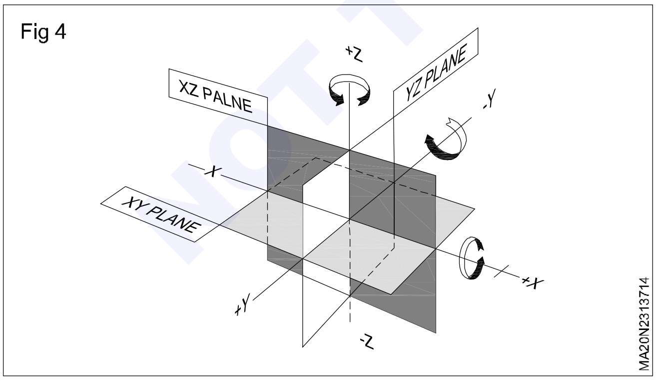
The point "F" is at a distance of -4 units from the X-axis and at a distance zero from the Y-axis. Therefore, the x-coordinate of "F" is 0 and y-coordinate is -4.

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

Machine geometry

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

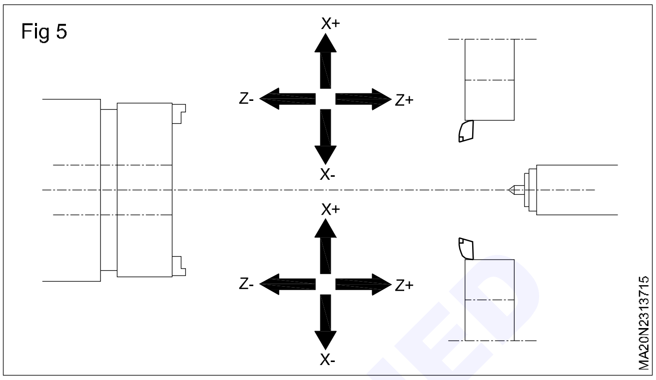
Fig 4



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

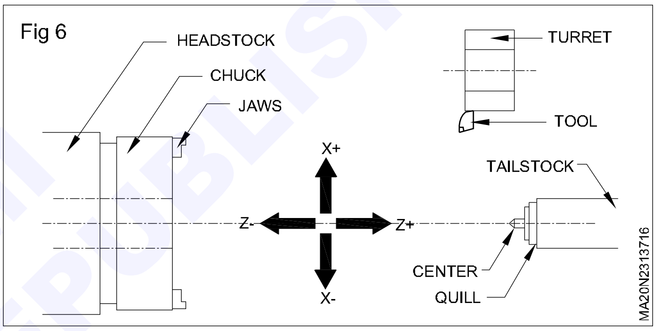
What is much more common for CNC lathes in industry, is the double orientation of XZ axes. CNC lathes are separated as front and rear lathes. An example of a front lathe is similar to the conventional engine lathe. All slant bed lathe types are of the rear kind. Identification of axes in industry have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

Fig 5



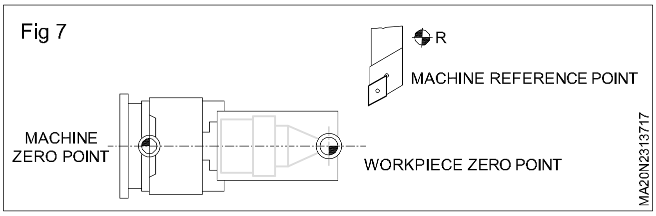
The rear bed axes orientation (Fig 6)

Fig 6



Cartesian coordinate system is used to describe the position of a 2D or 3D point in space. In Polar coordinate system a point is located by its distance to the point of origin and its angle to a specified axis. CNC Lathe Machine's coordinate system is used to ensure that machine is able to read the assigned coordinates correctly to indicate the position of the work piece. Key parts of the system are (Fig 7)

Fig 7



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

Machine zero point

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

Work zero point

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

Reference point

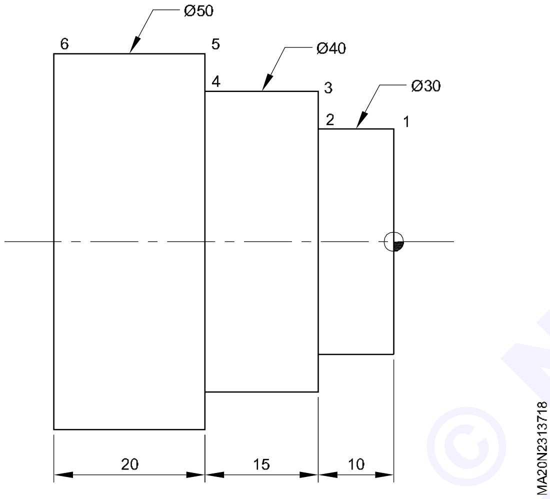
Reference point of any CNC machine has been selected at a specific fixed point during the initial machine design, by the machine design engineers. It is a fixed point, located within machine travel limits, and its actual position does not normally change. This point (position) is typically called the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

Programming method

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

Absolute method (Fig 8)

Fig 8



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

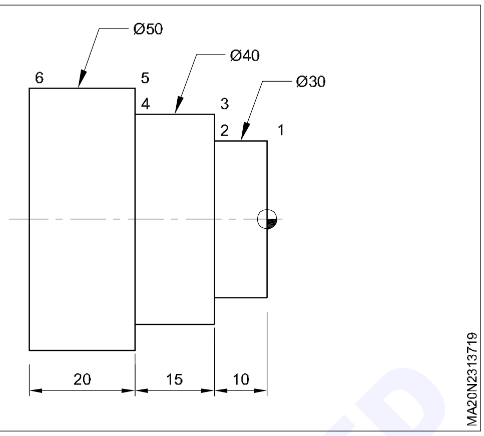
At the travel commands for tool are mean their coordinate value from the work piece zero point (X0, Z0).

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

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

Incremental method (Fig 9)

Fig 9



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

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

Example

In the Fig 9 the points, 1 to 6 can be specified as follows in incremental dimension programming.

Position	U	W
1	30.0	0.0
2	0.0	-10.0
3	10.0	0.0
4	0.0	-15.0
5	10.0	0.0
6	0.0	-20.0

Programming sequence, ISO G codes and M codes

Objectives : At the end of this lesson you shall be able to
- **state the CNC program**
- **list the three types of CNC program**
- **explain the ISO G and M codes**
- **describe the structure of the CNC program.**

Program

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

Types of CNC Programming

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

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

Conversational or "shop-floor" programming is an increasingly popular method that involves graphic and menu-driven functions. Operators input commands directly into the CNC machine, just like in manual programming.

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

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

Preparation of part programming

1 Block numbers / sequence number (N words)

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

2 Preparatory Function (G-words)

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

3 Dimension words (X, Y & Z words)

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

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

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

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

- Decimal point should be not be allowed e.g. x = 7.875 will be represented as X07875 in a five system i.e. the last three digits are used for the decimal part of the number. Some machines allow omission of leading zeros, hence the same can be represented as X7875.
- It is recommended that dimensions should be expressed in mm.
- All angular dimensions should be expressed as a decimal fraction of a revolution.
- In absolute system, all dimensions should be positive.

– In incremental system the '+','-'sign represent the direction of motion.

4 Feed Rate Word (F - word)

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

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

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

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

6 Tool selection word (T - word)

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

7 Miscellaneous words (M-words)

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

8 End of Block (EOB)

It identifies the end of instruction block.

G and M codes (G-codes)

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

List of G codes

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

List of M Codes

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

List of G codes

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

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

List of M codes

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

Part program

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

Example

O1203;

N1;

G28 U 0.0 W 0.0;

G50 S 1200 T 0300;

________________;
________________;
________________;

M01;

Part program

N2;

G28 U 0.0 W 0.0;

G50 S 1200 T 0200;

________________;
________________;
________________;
________________;

M01;

M30;

Part program

Decimal point input

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

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

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

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

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

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

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

X 50 = 0.05mm

X 500 = 0.5mm

X 5000 = 5.0mm

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

Note

1 micron=0.001mm

1 mm=1000 microns

1 inch=25.4mm

1 sec=1000 millisec

Structure or format of a part program

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

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

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

	%	(Program start)
	O3642	(Program number)
Blocks	N010 ---------	

	N100 M02;	(Program end)

Program number

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

Sequence number (N-word)

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

Coordinate function

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

Comments

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

Example

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

Table common word addresses used in word address format

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

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

M - Cods (Miscellaneous Function)

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

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

M01 - Optional stop

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

M02 - Program end

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

M03 - Spindle on clockwise

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

M04 - Spindle on counter clockwise

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

M05 - Spindle stop

By programming ‘M05’ the spindle rotation is stopped.

M08 - Coolant on

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

M09 - Coolant off

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

M30 - Program end & rewind

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

G - Codes (preparatory functions)

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

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

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

G00 - Rapid traverse

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

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

Format

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

G00 - code used for the following operations

1 Machining start

Making the tool approach the work piece.

2 During machining

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

3 Machining end

Separating the tool from the work piece.

G01 - Linear interpolation (straight cutting)

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

Format

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

Application

a Facing
b Grooving etc.

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

Application

a. Straight turning
b. Drilling etc

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

Application

a. Taper turning
b. Chamfering

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

Function F

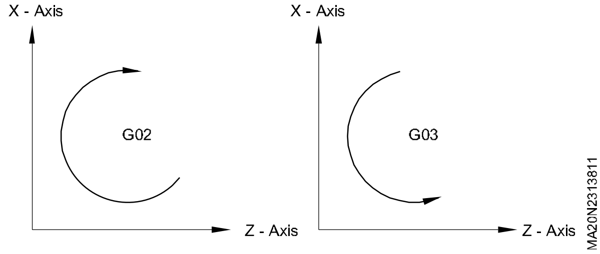
The feed rate is used to move the tool from one point to another point with constant feed rate. Feed is normally is given mm/rev. or mm/min. The rapid traverse rate and feed rate both are controlled by feed override switches in the machine panel.

Example

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

Circular interpolation (Fig 1)

Fig 1



G02-Circular interpolation clockwise direction

G03-Circular interpolation Anti clockwise direction

Format

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

OR

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

Where

XZ - End point of Arc

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

R - Radius of the arc

F - Feed

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

Example

G02 X 40.0 Z-5.0 R 5.0 F 0.1

G03 X 40.0 Z-5.0 R5.0 F 0.1

Where, R=Radius

G04-Dwell

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

Format

G04 (U, X, P) time

Example

G04 U 1.0 (Dwell of 1.0 second)

Note

Decimal point is not available in 'P'

Ex. Dwell of 2.5 seconds.

G04 U 2.5

G04 X 2.5

G04 P 2500

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

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

1 G28 U0

2 G28 W0

3 G28 U0 W0

G30 - Second reference return

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

1 G30 U0

2 G30 W0

3 G30 U0 W0

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

1 G50 X---Z---;

Ex. G50 X 300.0 Z 150.0;

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

Ex. G50 X300.0 Z 150.0 S 3000

G96-Constant Surface Speed Control (Cutting Speed Specification)

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

The G96 is used when the cutting speed is specified.

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

Calculation for cutting speed

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

Where

V = cutting speed

D = Diameter of the work piece in 'millimeter'

N = spindle speed in rpm

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

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

Ex. G97 S300 M03.

With this spindle rotates at 300 rpm.

For the following should use G97 always

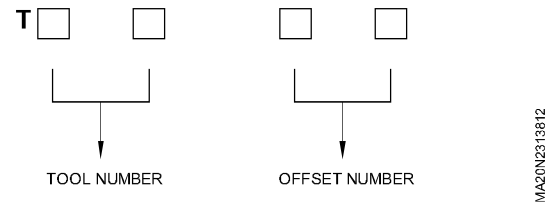
a Threading

b Tapping

c Drilling etc

Tool function (Fig 2)

Fig 2



Address: T

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

Format

Example : T01 01

Tool Number

The left most two digits specify the number of tool.

Offset Number

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

Types of offset

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

- **explain the different types of offset used in CNC.**

Types of Offsets

There are two types offsets:

1 Wear offset

2 Geometrical offset

1 Wear offset

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

2 Tool Geometry offset

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

Tool geometry offset (Fig 3)

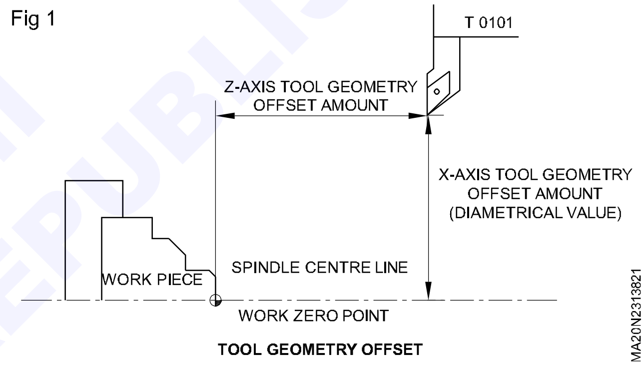
This offset amount is not need to be cancelled after every tool use because the next input of tool geometry offset cancels former offset automatically.

Tool wear offset

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

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

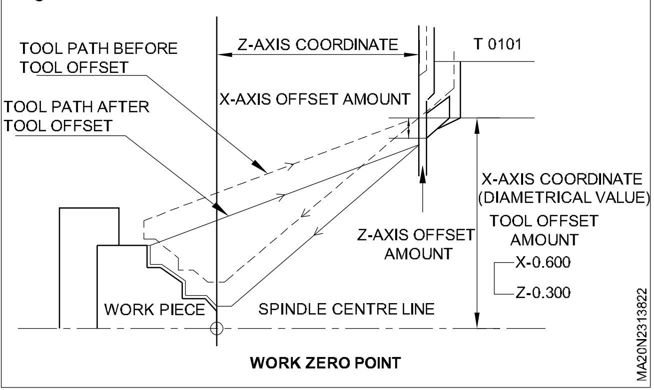
Fig 1



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

Procedure for setting work coordinate system (Fig 4)

Fig 2



Step 1 Mark sure that the component is securely clamped.

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

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

Step 4 Enter S500 M03;

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

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

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

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

Step 9 Now switch off the spindle.

Step 10 Press MENU offset. The wear geometrical and work shift are displayed on CRT.

Step 11 Now, press GEOM soft key and position the cursor using cursor movement buttons to be required offset number corresponding to the tool used.

Step 12 Press measure(m) key and press Z. Enter Zero(MZ0) press measure (m) key.

Step 13 Now rotate the spindle in appropriate direction and machine on OD

Step 14 Do not move X axis.

Step 15 Take Z away from the job.

Step 16 Stop the spindle.

Step 17 Press MENU OFFSET

Step 18 Press 'GEOM' soft key.

Step 19 Position the cursor to the required tool offset number.

Step 20 Press M….X….

Step 21 Input "The OD dimension measured.
The X-offset for the said tool is set.

Step 22 Repeat the procedure for all tools.

Program execution in different modes

Objectives : At the end of this lesson you shall be able to
- **state the function of the MDI mode operation**
- **list the use of DNC operation**
- **explain the purpose of the automatic mode operation**
- **state the single block mode of operation.**

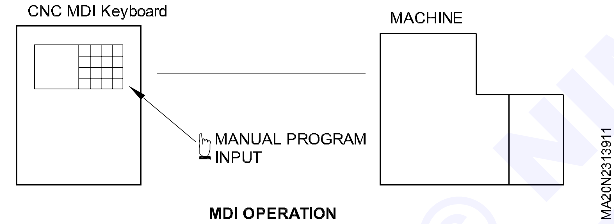
Automatic operation is to operate the machine according to the created program. It includes memory, MDI, and DNC operations

After the program is once registered in memory of CNC, the machine can be run according to the program instructions. This operation is called memory operation.

MDI operation

After the program is entered, as a command group, from the MDI keyboard, the machine can be run according to the program. This operation is called MDI operation. (Fig 1)

Fig 1



MDI OPERATION

DNC operation

The DNC (Direct Numerical Control) machine can be operated by reading a program directly from an external input/output device, without having to register the program in CNC memory. This is called DNC operation.

Automatic operation

Select the program used for the work piece. Ordinarily, one program is prepared for one work piece. If two or more programs are in memory, select the program to be used, by searching the program number (Fig 2)

Start and stop

Pressing the cycle start push button causes automatic operation to start. By pressing the feed hold or reset push button, automatic operation pauses or stops. By specifying the program stop or program termination command in the program, the running will stop during automatic operation. When one process machining is completed, automatic operation stops. (Fig 3)

Fig 2

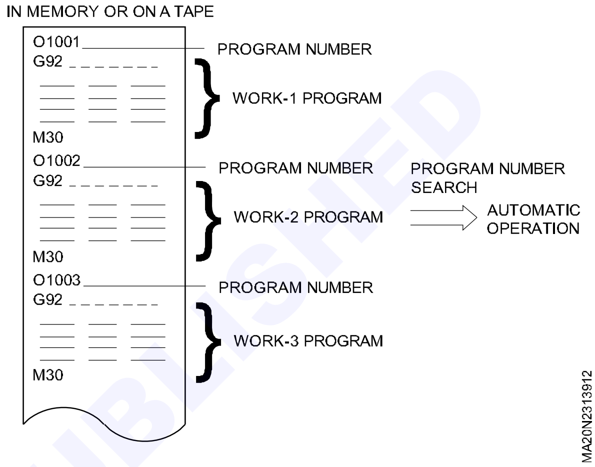
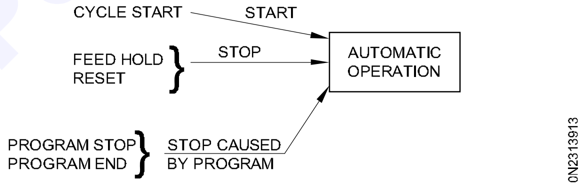


Fig 3

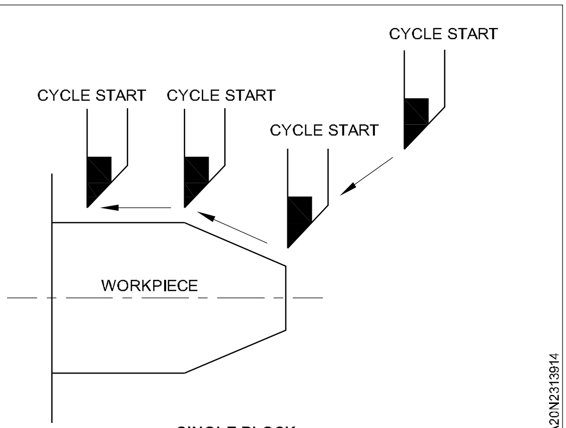


START AND STOP FOR AUTOMATIC OPERATION

Single block

When the cycle start push button is pressed, the tool executes one operation then stops. By pressing the cycle start again, the tool executes the next operation then stops. The program is checked in this manner (Fig 4)

Fig 4



SINGLE BLOCK

Canned cycles

Objective: At the end of this lesson you shall be able to
- **list the canned cycles used in turning program**
- **describe the cycles used in turning program.**

Canned cycles

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

G70 - Finishing cycle (Fig 1)

Format:

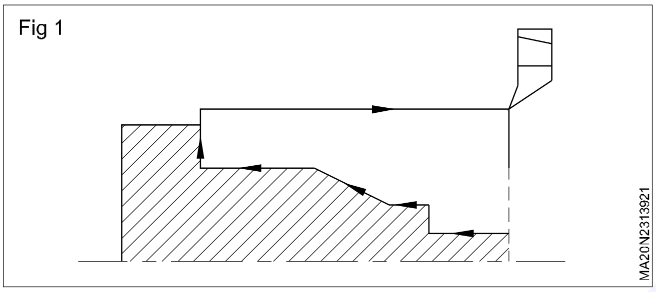
G70 P_Q_F_;

P : Starting Block Number

Q : Ending Block Number

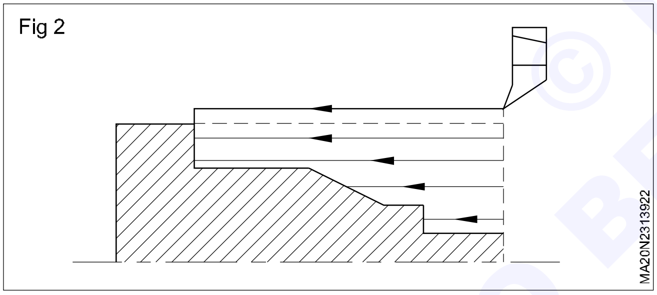
F : Finishing Feed

Fig 1



G71 - Turning cycle(Fig 2)

Fig 2



Format

G71 U_R_;

G71 P_Q_U_W_F_;

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

R : Relief Amount

P : Starting Block Number

Q : Ending Block Number

U : Finishing Allowance in X Axis

W : Finishing Allowance in z Axis

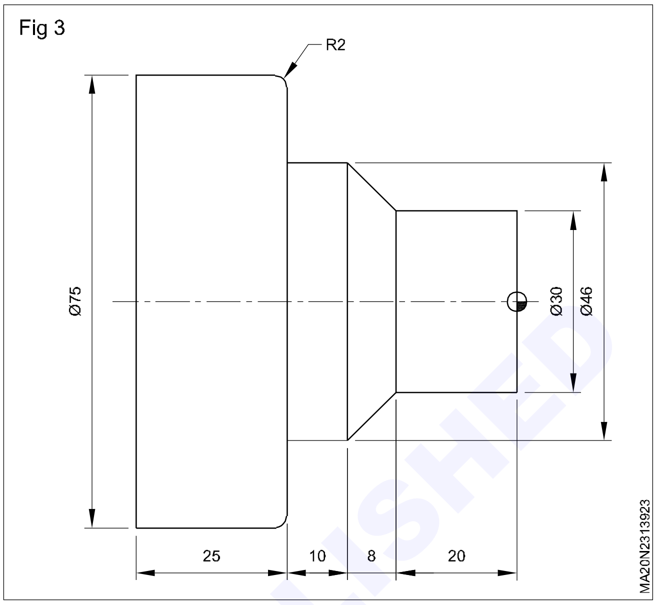
F : Feed

Example for G71 turning cycle

Blank size: ϕ 55 X 50

```
%
O0026 (Fig 3)
```

Fig 3



```
N1 ;
G28 U0 W0 ;
G92 S1200 T0300 ;
G96 S250 M03 ;
G00 X65.0 Z0.0 T0303 M08 ;
G01 X0.0 F0.1 ;
G00 X61.0 Z2.0 ;
G71 U1.0 R1.0 ;
G71 P10 Q20 U1.0 W0.5 F0.3 ;
N10 G00 X30.0 ;
G01 Z-20.0 F0.1 ;
X46.0 Z-28.0 ;
Z-38.0 ;
G1 X71 ;
G03 X75.0 Z-40 R2 ;
G01 Z-63 ;
N 20
G00 X85.0 Z2.0 M09 ;
G28 U0 W0 M05 ;
M01 ;
N2 ; (OD FINISHINING)
G28 U0 W0 ;
G92 S2600 T0700 ;
G96 S150 M03 ;
G00 X31.0 Z0.0 T0705 M08 ;
```

```
    G01 X0.0 F0.1 ;
    G00 X100.0 Z3.0 ;
    G70 P10 Q20 F0.12 ;
    G00 X100.0 Z2.0 M09 ;
    G28 U0 W0 M05 ;
    M30 ;
```

G72 - Facing cycle

Format

G72 W_R_;

G72 P_Q_U_W_F_;

W : Depth of cut per pass in Z Axis

R : Relief Amount

P : Starting Block Number

Q : Ending Block Number

U : Finishing Allowance In X Axis

W : Finishing Allowance In Z Axis

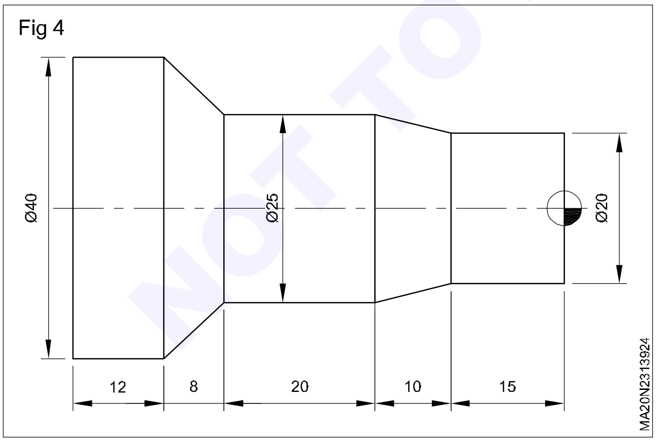
F : Feed

```
G00 X61.0 Z2.0 ;
    G71 U1.0 R1.0 ;
    G71 P10 Q20 U1.0 W0.5 F0.3 ;
    N10 G00 X30.0 ;
    G01 Z-20.0 F0.1 ;
    X46.0 Z-28.0 ;
    Z-38.0 ;
```

Example for G72 Facing cycle

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

Fig 4



```
    N1 ;
    G28 U0 W0 ;
    G50 S1300 T0200 ;
    G96 S150 M03 ;
    G00 X55.0 Z2.0 T0202 M08 ;
    G72 W1.0 R2 ;
    G72 P20 Q30 U1.0 W0.5 F0.3 ;
    N20 G00 Z-65.0 ;
    G01 X40 ;
    Z-53.0 ;
    X25.0 Z-45.0 ;
    Z-25.0 ;
    X20.0 Z-15.0 ;
    Z0.0 ;
    N30 ;
    G00 X55.0 Z2.0 M09 ;
    G28 U0 W0 M05 ;
    M01 ;
    N2 ; (OD Finishing)
    G28 U0 W0 ;
    G50 S2600 T0700 ;
    G96 S150 M03 ;
    G00 X55.0 Z2.0 T0707 M08 ;
    G70 P20 Q30 F0.12 ;
    G00 X65.0 Z2.0 M09 ;
    G28 U0 W0 M05 ;
    M30 ;
```

G74 - Peck drilling cycle

Format

G74 R_;

G74 Z___Q___F___;

R : Retract value

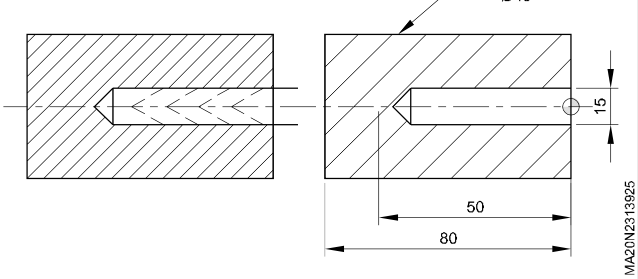
Z : Depth of the hole

Q : Depth of cut per pass in Microns

F : Feed rate

```
%
O0029 (Fig 5)
N1 ; (C.D)
G28 U0 W0 ;
```

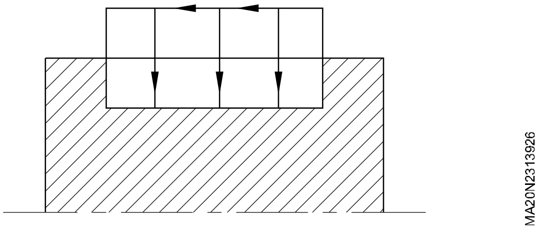
Fig 5



```
T0200 ;
G97 S1500 M04 ;
G00 X0.0 Z5.0 T0202 M08 ;
G01 Z-3.0 F0.12 ;
G00 Z5.0 M09 ;
G28 U0 W0 M05 ;
M01 ;
N2 ;
G28 U0 W0 ;
T0700 ;
G97 SI500 M04 ;
```

G75 - Grooving cycle (Fig 6)

Fig 6



Format

G75 R___;

G75 X____Z___P____Q___R___F___;

R : Relieving the tool (mm)

X : Groove diameter (mm)

Z : Groove length (mm)

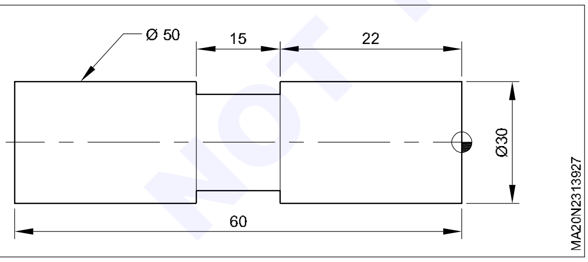
P : Depth of cut in X axis in microns (Radial value)

Q : Shift value in Z axis (microns)

F : Feed

```
%
O0030 (Fig 7)
```

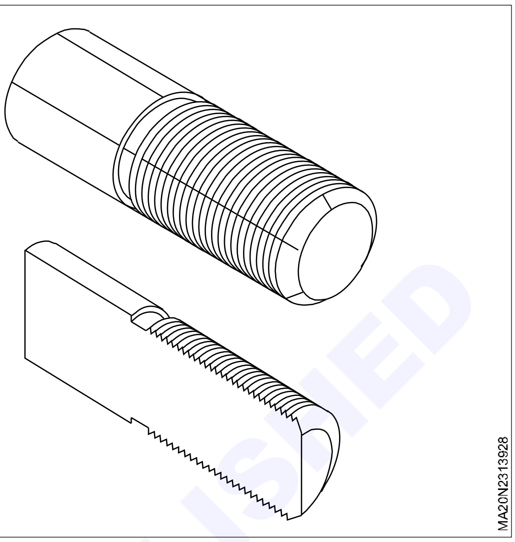
Fig 7



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

G76 - Multiple thread cutting cycle(Fig 8)

Fig 8



Format:

G76 P___Q____R___;

G76 X___Z___P___Q____F____;

Explanation for the cycle:

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

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

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

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

Z : Thread length

P : Height of thread (microns)

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

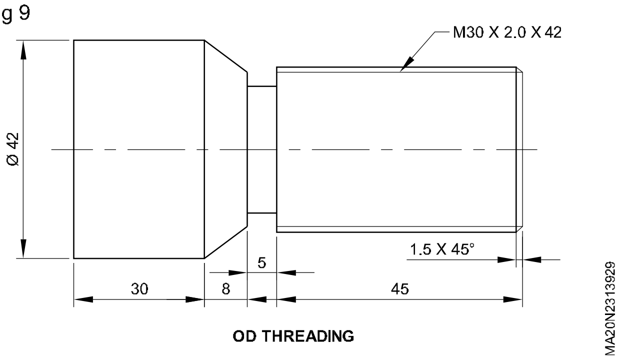
F : Feed (pitch of the thread)

Example

OD Threading

```
%
O0031 (Fig 9)
G28 U0 W0 ;
T0400 ;
G97 S600 M04 ;
G00 X32.0 Z5.0 T0404 M08 ;
G76 P030060 Q150 R20 ;
```

Fig 9



G76 X27.404 Z-42.0 P1298 Q300 F2.0 ;

G00 X32.0 ;

Z5.0 M09 ;

G28 U0 W0 M05 ;

M30 ;

Threading calculation

Minor dia, d = D - (2h)

(h = 0.649 p, for metric thread)

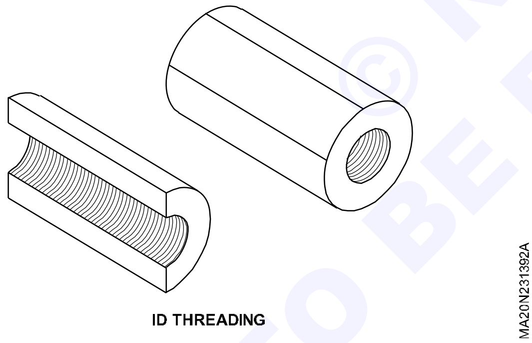
h = 0.649 2.0=1.298 mm.

d = 30 - (2 1.298)

= 27.404 mm.

ID Threading (Fig 10)

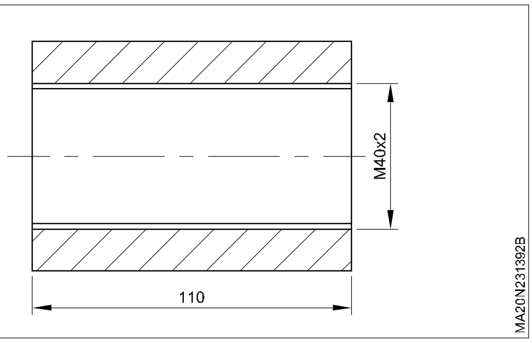
Fig 10



%

O0032 (Fig 11)

Fig 11



G28 U0 W0 ;

T0500 ;

G97 S600 M04 ;

G00 X38.0 Z5.0 T0505 M08 ;

G76 P030060 Q150 R20 ;

G76 X40 Z-110.0 P1298 Q300 F2.0 ;

G00 X38.0 ;

Z5.0 M09 ;

G28 U0 W0

M05;

M30;

Threading calculation

Minor dia, d = D - (2h)

(h = 0.649 P, for metric thread)

h = 0.649 2.0 = 1.298 mm

d = 40 - (2 X 1.298)

=37.404 mm.

Tool Nose Radius Compensation (TNRC)

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

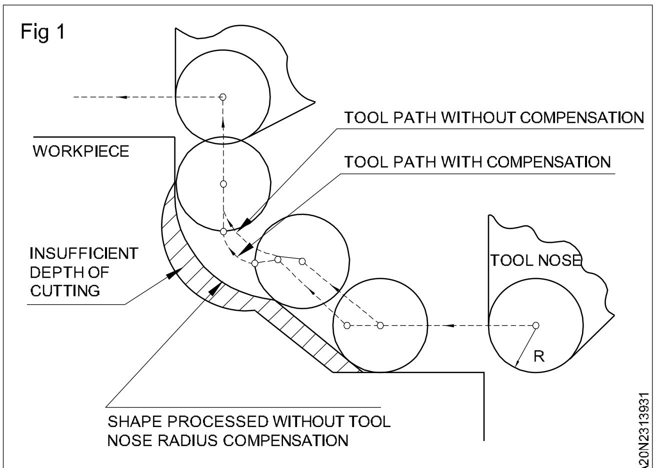
Overview of tool nose radius compensation (Fig 1)

It is difficult to produce the compensation necessary to form accurate parts when using only the tool offset function due to tool nose roundness in taper cutting or circular cutting. The tool nose radius compensation function compensates automatically for the above errors.

Imaginary Tool Nose (Fig 2)

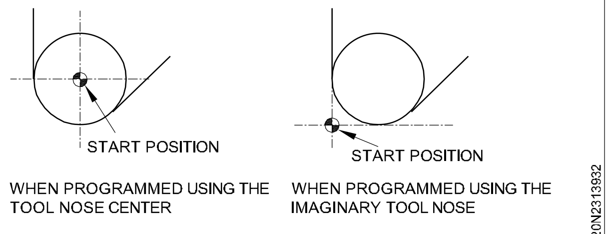
The tool nose at position A in following figure does not actually exist. The imaginary tool nose is required because it is usually more difficult to set the actual tool nose radius centre to the start position than the imaginary tool nose (Note). Also when imaginary tool nose is used, the tool nose radius need not be considered in programming. The position relationship when the tool is set to the start position is shown in the following figure.

Fig 1



TOOL PATH OF TOOL NOSE RADIUS COMPENSATION

Fig 2

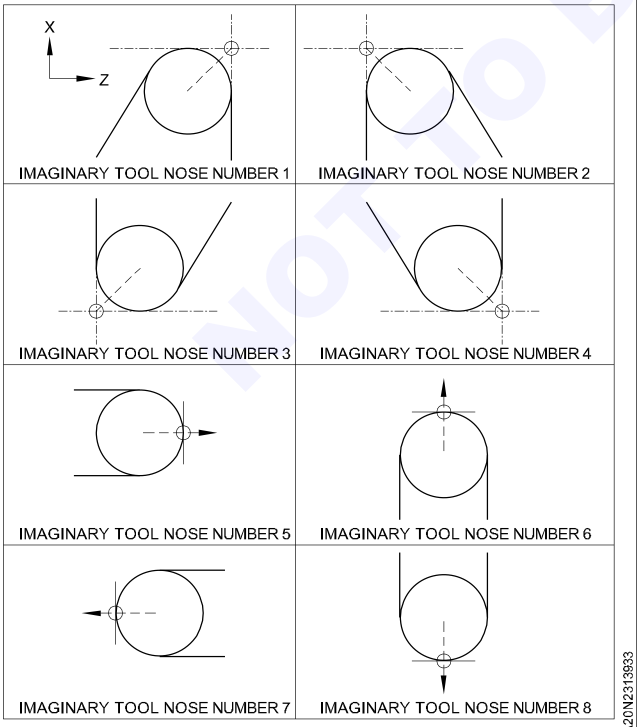


TOOL NOSE RADIUS CENTER AND IMAGINARY TOOL NOSE

Direction of Imaginary Tool Nose

The direction of the imaginary tool nose viewed from the tool nose centre is determined by the direction of the tool during cutting, so it must be set in advance as well as offset values. The direction of the imaginary tool nose can be selected from the eight specifications shown in the Fig 3 together with their corresponding codes. This Fig 3 illustrates the relation between the tool and the start position. The following apply when the tool geometry offset and tool wear offset option are selected.

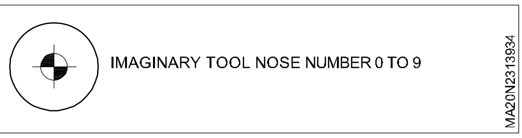
Fig 3



DIRECTION OF TOOL NOSE

Imaginary tool nose numbers 0 and 9 are used when the tool nose centre coincides with the start position. (Fig 4)

Fig 4



Offset Number and Offset Value are entered in tool geometry offset page. (Fig 5&6)

Fig 5

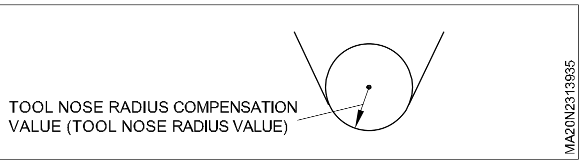
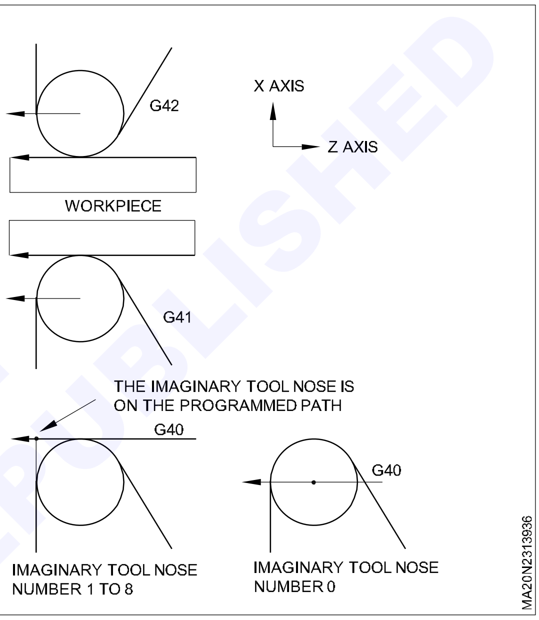


Fig 6



Position the cursor at appropriate place and input the values

The tool nose radius compensation value during execution is the sum of the geometry offset and the wear offset.

Work Position and Move Command

In tool nose radius compensation, the position of the work piece with respect to the tool must be specified (Fig 7&8)

Fig 7

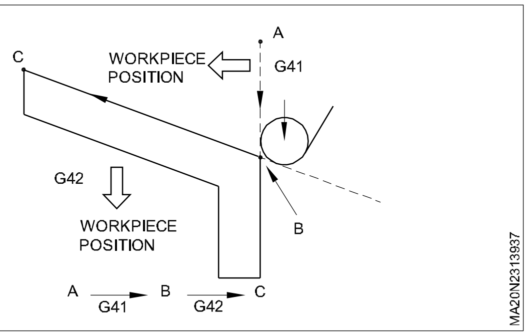


Fig 8

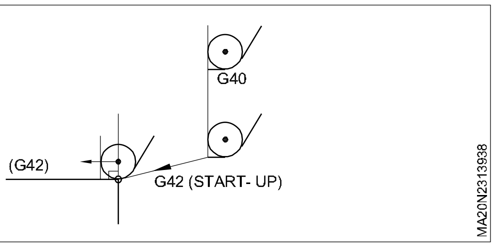


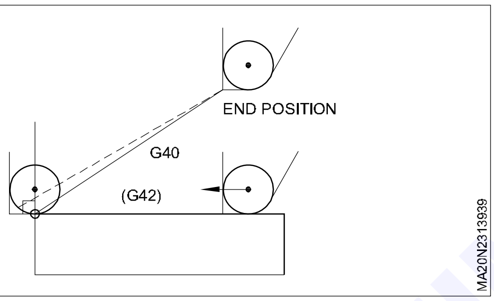
Table 1 - Tool geometry offset

Geometry offset number	OFGX (X-axis geometry offset	OFGZ (Z-axis geometry offset	OFGR (Tool nose radius geo-metry offset	OFT (Imaginary tool nose direction)	OFGY (Y-axis geometry offset amount)
G01	10.040	50.020	0	1	70.020
G02	20.060	30.030	0	2	90.030
G03	0	0	0.20	6	0
G04	:	:	:	:	:
G05	:	:	:	:	:
:	:	:	:	:	:

Tool movement when the work piece position changes

The work piece position against the toll changes at the corner of the programmed path as shown in Fig 9.

Fig 9



Although the work piece does not exist on the right side of the programmed path in the above case. the existence of the work piece is assumed in the movement from A to B. The work piece position must not be changed in the block next to the start-up block. In the above example, if the block specifying motion from A to B were the start-up block, the tool path would not be the same as the one shown.

Start-up

The block in which the mode changes to

G41 or G42 from G40 is called the start-up block.

G40 _ ;

G41 _ ; (Start-up block)

Transient tool movements for offset are performed in the start-up block. In the block after the start-up block, the tool nose centre is positioned Vertically to the programmed path of that block at the start position. (Fig 10)

Offset cancel

The block in which the mode changes to G40 from G41 or G42 is called the offset cancel block. G41 _ ; G40 _ ; (Offset cancel block) The tool nose centre moves to a position vertical to the programmed path in the block before the cancel block. The tool is positioned at the end position in the offset cancel block (G40).

Examples

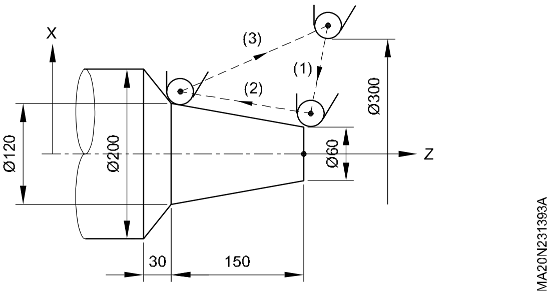
(G40 mode)

1 G42 G00 X60.0 ;

2 G01 X120.0 W-150.0 F10 ;

3 G40 G00 X300.0 W150.0 I40.0 K-30.0 ;

Fig 10



Cutting tool materials, inserts and tool holders for turning

Objectives : At the end of this lesson you shall be able to
- **explain the properties of cutting tool**
- **state the types of tool material.**

Cutting tool materials and their properties

Tool materials are the subject of intense development. They are the product of an evolvement that has taken place almost entirely during the twentieth century, and especially since the thirties. Machining which took one hundred minutes in 1900, today takes less than one minute. It is not an exaggeration to say that the evolvement of tool materials has been one of the major contributing factors that has helped to make the modern, efficient industrial world.

Today, there is a tool material to optimize every metal cutting operation-one that will cut a certain work piece, under certain conditions in the best way. Not only have completely new materials appeared but high speed steel, which was the major break through at the beginning of the century, has been developed to machine several times faster. It is however, the introduction and continuous improvement of hard materials that have really improved metal cutting during the recent decades.

Cutting tool properties

The most important properties required by a cutting tool material are,

Toughness

Ability to withstand the various cutting forces during machining.

Hardness

Ability to retain hardness under severe working conditions.

High resistance to wear

The material must withstand excessive wear even though the relative hardness of the tool materials changes.

Frictional coefficient

The frictional coefficient must remain low for minimum wear and reasonable surface finish.

Cost and easiness in fabrication

The cost and easiness of fabrication should have within reasonable limits.

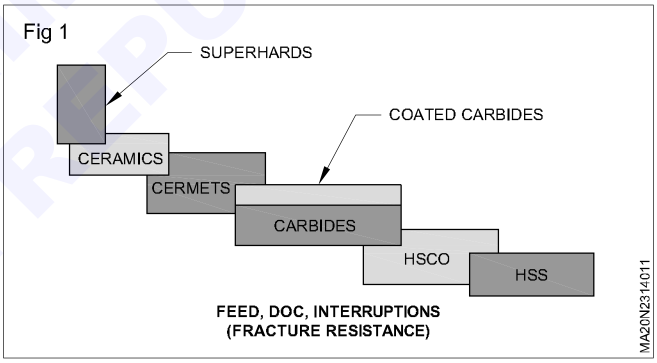
Evolution of cutting tools

- 1910-1920: High speed steel
- 1920's: Cemented carbide
- 1950's: Cermets (Tic -based)
- 1960's: Alumina-based ceramic
- 1970: CVD coated carbide
- 1980: First engineered carbide substrate (cobalt-enrichment)
- 1982: First SiAION ceramic
- 1985: First PVD coated carbide
- Mid 80's: Modern cermets (TiCN-based)
- Late 80's: SiC whisker reinforced Al_2O_3 ceramic
- Early 90's: Advanced Sialons
- Mid 90's: thin film diamond coated carbide
- Late 90's: PVD coated PCBN
- 2000: Advanced pre-coat & post-coat treatments

Cutting tool materials

Metal cutting environment (Fig 1)

Fig 1



- Heat (thermal deformation)
- Pressure (deformation, fracture)
- Wear (pure abrasion, chemical wear, notching)
- Interrupted cuts (thermal & mechanical cycling)

Types of tool materials

The selection of proper tool materials depends on the type of service to which the tool is subjected. The commonly used cutting materials are:

Carbon steels

- It is basically high carbon steel with percentage of carbon in the range of 0.8 to 1.5
- It may only be used in manufacture of tools operating at low cutting speed (12m/min).
- They are comparatively cheap, easy to forge and simple to harden.
- Disadvantage of carbon tool steel is their comparatively low heat and wear resistance.

High speed steel

- It is the general purpose metal for low and medium cutting speeds owing to its superior hot hardness and resistance to wear.
- HSS can operate at cutting speeds 2 to 3 times higher than for carbon steels and retain its hardness up to 900°C.
- Tungsten in HSS provides hot hardness and form stability. Molybdenum maintains keenness of the cutting edge. Cobalt makes the cutting tool more wear resistant.

Stellites

- Stellites is the trade name of a non-ferrous cast alloy composed of cobalt, chromium and tungsten.
- The range of elements in these alloys is 40% to 48% cobalt, 30% to 35%, chromium and tungsten.
- Stellites can be operated on steel at cutting speeds 2 times higher than for HSS.
- They are used for non metal cutting application such as rubbers, plastics etc.,

Carbides

- They are composed principally of carbon mixed with other elements.
- The basic ingredient of most carbides is tungsten carbide, which is extremely hard. Pure tungsten powder, is mixed under high heat(1500°C) with pure carbon in the ratio of 94% and 6% weight.
- The two types of carbides are the tungsten and titanium and both are more wear resistant.

Coated carbides

- The coated carbide has substrate and coating layer
- Substrate-for toughness having hard material and soft material (cobalt + carbide)
- Coating -Layer of carbide (very hard)
- Perform well on all work material
- Better impact strength to resist fracture
- Allow good coating adhesion

Ceramics

- The latest development in the metal cutting tool uses aluminium oxide, generally referred to as ceramics.
- Compacting aluminium oxide powder in a mould at about 280 kg/sq.cm or more makes ceramic tools. The part is then sintered at 2200 °C. This method is known as cold pressing.
- Ceramic tool material are made in the form of tips that are to be clamped on metal shanks
- The tools have low conductivity and extremely high compressive strength, but they are quite brittle and have a low bending strength.
- They can with stand temperature up to 1200°C and can be used at cutting speeds 4 times that of carbide and up to 40 times that of HSS.
- To give them increased strength often ceramic with metal bond know as cermets is used.
- Heat conductivity of ceramics being very low the tools are generally used without coolant.

Cermets

- Cermets -Ceramics and Metal

Characteristics of cermets

- High Hardness
- High Hot Hardness
- Resist oxidation
- Low Friction

Advantage of cermets

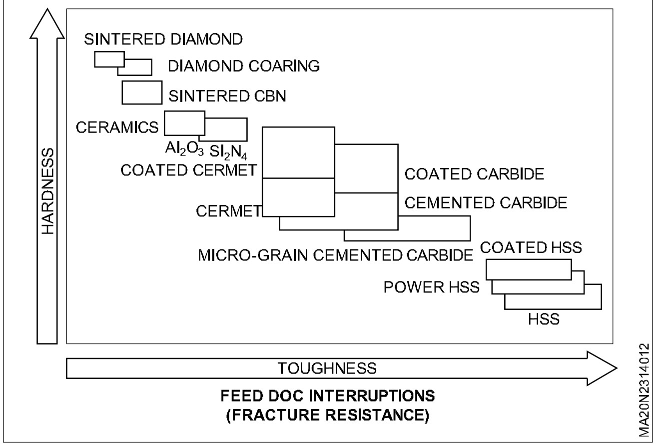
- High efficiency
- Long life
- Large batch
- Avoid Build Up Edge
- Surface Finish Control
- Cermets Have the properties of higher cutting speed and wear resistance which enables hard part turning.

Diamond

- The diamond is the hardest known material and can be run at cutting speed about 50 times greater than that of HSS tool and 5 to 6 times of tool life than carbide.
- Diamond is incompressible, readily conducts heat and has low coefficient of friction.
- Diamond are suitable for cutting very hard material such as glass, non-ferrous materials, plastics etc.,
- For poly crystalline diamond (PCD) the tool life is 30 times of carbide.

The following picture shows the comparison of the various materials in terms of their properties. (Fig 2)

Fig 2

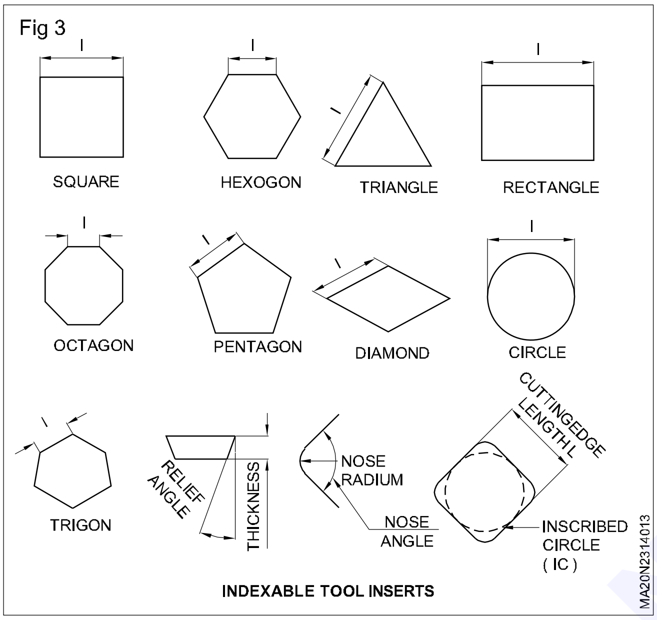


Tool inserts

Carbides and other harder tool materials are very costly. Moreover, they cannot be machined. So, only tool tips are made for such materials using powder metallurgy technique. In this method, the tool material is taken in a powder form. It is mixed with a suitable binder (in powder form) and compressed in the shape of an insert.

Inserts are available in various shapes such as triangle, square, rectangle, pentagon, hexagon, octagon, diamond shaped and circle. They cannot be resharpened, but they have a number of cutting edges. (Fig 3)

Fig 3



INDEXABLE TOOL INSERTS

Inserts are produced in various sizes and thicknesses. Smallest possible size is chosen to produce the desired depth of cut. Thickness of an insert affects its strength. Hence, for a large depth of cut and feed, a thicker insert is chosen.

ISO standard is commonly followed for specifying inserts. An example is CNMG120408. The first letter, C in this case, indicates the shape of the insert. The common types are:

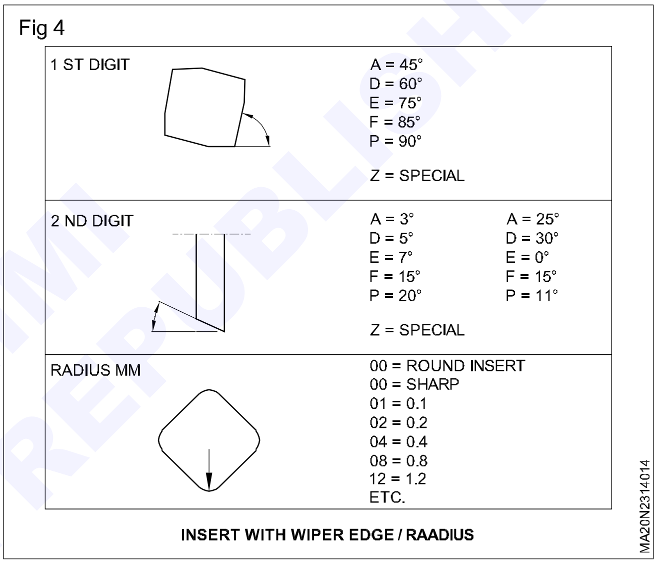
Symbol	Shape
S	Square
T	triangular
H	hexagonal
O	octagonal
P	pentagonal
L	rectangular
R	round
A, B, K	parallelogram (nose angles 85°, 82° and 55° respectively)
C, D, E, F, M, V	Diamond shaped or rhombic (nose angles 80°, 55°, 75°, 50°, 86°, 35 ° respectively)
W	Trigon (nose angle 80°)

The second letter specifies the relief angles (Fig 9)

Symbol	Relief Angle
N	0°
A	3°
B	5°
C	7°
P	11°
D	15°
E	20°
F	25°
G	30°

The third letter specifies tolerances on various dimensions (Fig 4) (e.g., thickness) of the insert. The different tolerance classes are A, F, C, H, E, G (absolute values) and J, K, L, M, N, U (tolerance values depend on the diameter of the inscribed circle of the insert).

Fig 4



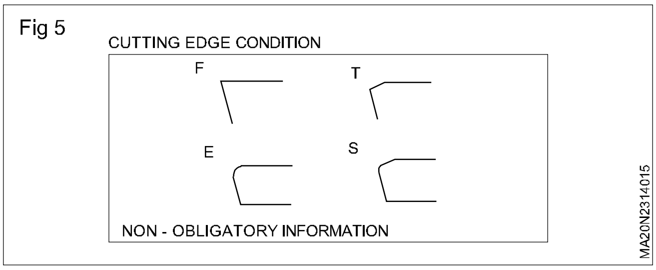
INSERT WITH WIPER EDGE / RAADIUS

Significance of the fourth letter

The fourth letter describes the overall geometrical features of the insert (refer table). For example, an insert may or may not have a hole at the centre. The hole may be cylindrical or cylindrical with single or double countersink. The insert may or may not have a chip-breaker. The chip-breaker may be single-sided or double-sided.

– **Cutting edge condition** (Fig 5)

Fig 5



F for sharp,

T for chamfered,

E for honed and

S for chamfered and honed.

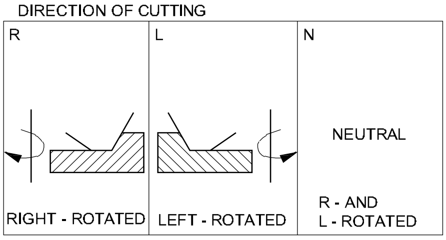
This information , however, is non-obligatory.

- **Cutting direction** (Fig 6)

  L for machining with left - rotated (CCW) spindle (M04),

  R for machining with right - rotated (CW) spindle (M03) and N for both left-and right - rotated.

Fig 6



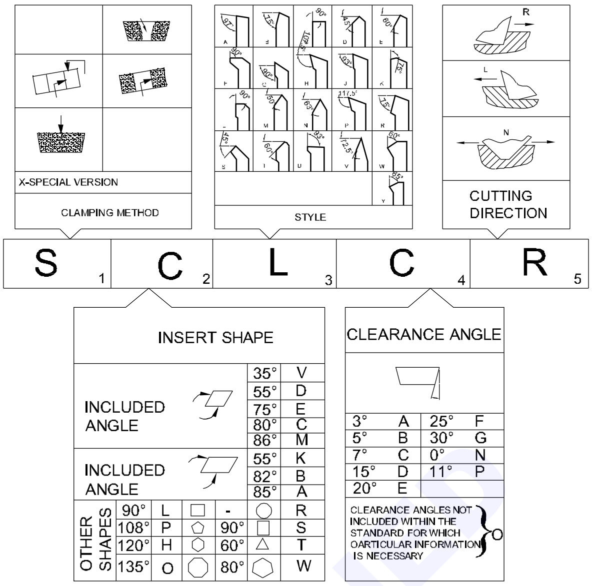
CUTTING EDGE CONDITION AND DIRECTION OF ROTATION OF MILLING INSERTS

Symbol	Hole	Shape of Hole	Chip-Breaker
N	without	no hole	without
R			single-sided
F			double -sided
A	with	cylindrical	without
M			single-sided
G			double -sided
W		cylindrical with 40°-60° countersink	without
T			single sided
Q		cylindrical with 40°-60° double countersink	without
			double-sided
B		cylindrical with 70°-90° countersink	without
H			single-sided
C		cylindrical with 70°-90° double countersink	without
J			double-sided
X		special shape	

The appropriate character designations are appended to the right, after the radius specification.

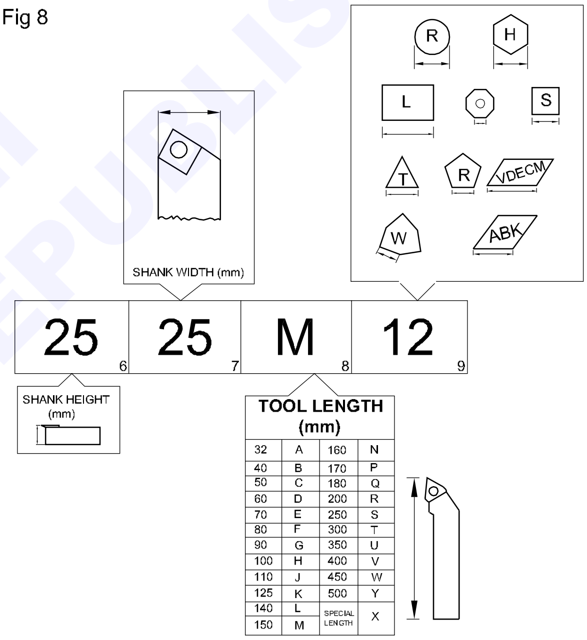
Tool holders for lathe: There is an ISO designation system for tool holders also, to suit various types of inserts. The first five characters describe insert clamping method, compatible insert shape, insert holding style of the tool holder (side cutting edge angle/end cutting edge angle and straight shank/offset shank), clearance angle and cutting direction, respectively. Next four digits specify shank height and shank width in mm (two digits for each). Tool length is specified next, by a character code. The next and the last two digits specify cutting edge length in mm. Manufacturers, specific codes may be appended in the end. (Figs 7&8)

Fig 7



ISO DESIGNATION FOR LATHE TOOL HOLDERS

Fig 8



ISO DESIGNATIOM FOR LATHE TOOL HOLDERS

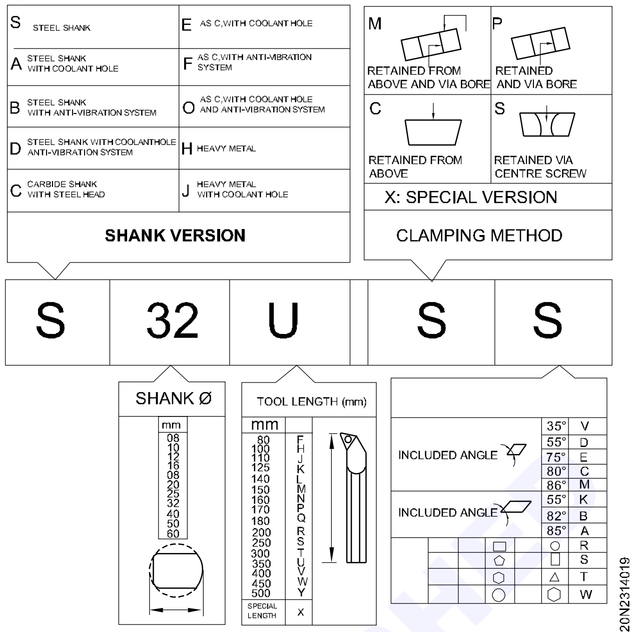
Insert clamping method (S)	Retained via central screw
Compatible insert shape (C)	80° diamond
Style of the tool holder body (L)	-5° side cutting edge angle, 5° end cutting edge angle and offset shank
Clearance angle (C)	7°
Cutting direction (R)	Right-hand
Shank height (25)	25 mm
Shank width (25)	25 mm
Tool length (M)	150 mm
Cutting edge length (12)	12 mm
Manufacture specific information	None

Boring bars for lathe (Figs 9 & 10)

There is a similar ISO designation system for the internal tool hodlders also (which are called boring bars). See Fig 9 and Fig 10 for details. Refer to Table 1 also for shapes not shown in these figures. In fact, tool length, clamping method, compatible insert shape, body style, clearance angle and cutting edge length have the **same representation** for tool holders as well as boring bars. For a boring bar, the cutting. If cutting is possible with clockwise rotation (MO3) of the spindle. The example, considered in the figure is S32U SSKCR12 type boring bar. Manufacturer-specific information may be appended in the end, after a gap. Separating dash (1) is also used in place of gaps. Referring to Table 1 and Figs 8&9, the description of S32U SKKCR12 is as follows:

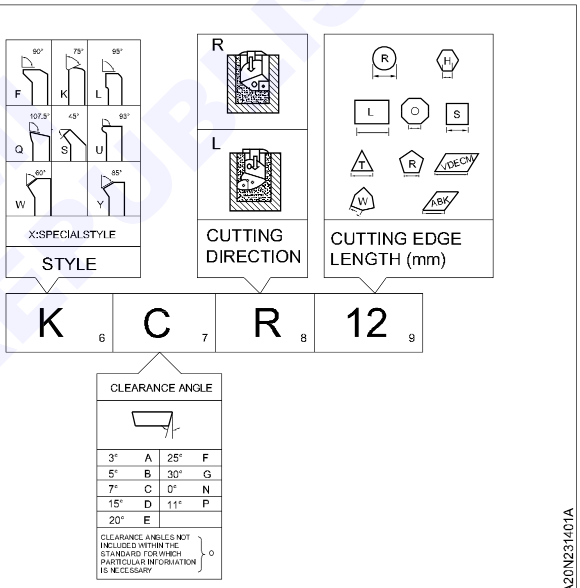
Shank type (S)	Steel shank
Shank diameter (32)	ϕ 32 mm
Tool length (U)	350 mm
Insert clamping method	Retained via central screw
Compatible insert shape (S)	Square
Style of the boring bar body (R)	-15° end cutting edge angle. offset shank.
Clearance angle (C)	7°
Cutting direction (R)	Right-rotated (CW spindle)
Cutting edge length (12)	12 mm
Manufacturer information	- specific None

Fig 9



ISO DESIGNATION FOR LATHE BORING BARS

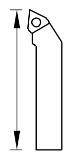
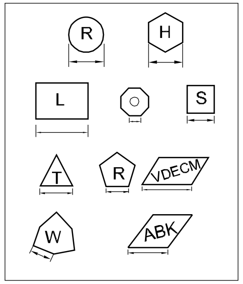
Fig 10



ISO DESIGNATION FOR LATHE BORING BAR

TABLE 1

ISO designation for lathe tool holders (Contd.)

1st char	2nd char	3rd char	4th char	5th char	6th char	7th char	8th char	9th char
Insert holding method	**Insert shape**	**Tool holder style**	**Insert relief (clearance)**	**Hand**				
M=top clamp and lock pin via the bore	A-85° parallelogram	A = 0° side cutting straight shank	N = 0°	R=right - hand	Shank height	Shang width	Tool holder length	Insert cutting edge length
P = lock pin via the bore only	B = 82° parallelogram	B = 15° side cutting cutting straight shank	A = 3°	L = left- hand				
C=top clamp only	C=80° diamond	C = 0° end cutting shank	B = 5° straight	N - neutral				
S=centre screw lock only	D=55° diamond	D = 45° side cutting, 45° end cutting, straight shank	C = 7°					
X = other methods	E = 75° diamond	E = 30° side cutting, straight shank	P = 11°					
	H = hexagon	F = 0° end cutting, offset shank	D = 15°					
	K = 55° parallelogram	G = 0° side cutting, offset shank	E = 20°					
	L = rectangle	H = -17.5° side cutting, offset shank	F = 25°					
	M=86° diamond	J = -3° side cutting,offset shank	G = 30°					
	O = octagon	K = 15° end cutting, offset shank						
	P = pentagon	L = 5° side cutting, 5°end cutting, offset shank						
	R = round	M = 40° side cutting, 50° end cutting, straight shank						
	S = square	N = 27° side cutting, straight shank						

TOOL LENGTH (mm)			
32	A	160	N
40	B	170	P
50	C	180	Q
60	D	200	R
70	E	250	S
80	F	300	T
90	G	350	U
100	H	400	V
110	J	450	W
125	K	500	Y
140	L	SPECIAL LENGTH	X
150	M		

TABLE 1

ISO designation for lathe tool holders (Contd.)

1st char	2nd char	3rd char	4th char	5th char	6th char	7th char	8th char	9th char
Insert holding method	Insert shape	Tool holder style	Insert relief (clearance)	Hand				
	T = triangle V = 35° W = 80° trigon	P = -27.5° side cutting, offset shank R = 15° side diamond cutting, offset shank S = 45° side cutting, offset shank T = 30° side cutting, offset shank U = 3° end cutting, offset shank V = 17.5° side cutting, straight shank W = 30° end cutting, offset, shank Y = -5° end cutting, offset shank						

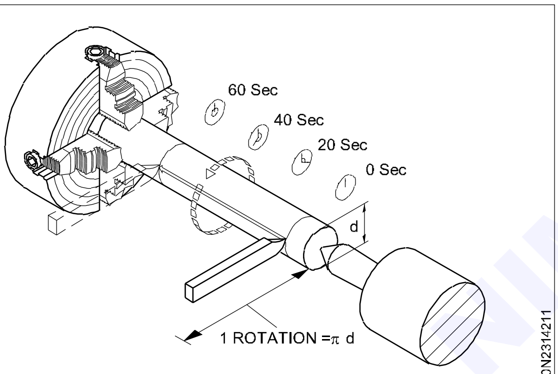
Cutting speed and feed

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

Cutting speed (Fig 1)

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

Fig 1



CUTTING SPEED FOR TURNING OPERATION

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

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

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

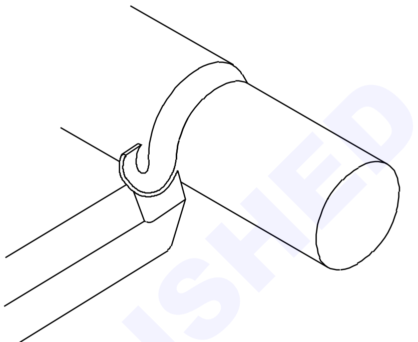
Example

Find out the rpm of a spindle of a 50mm bar to cut at 25m

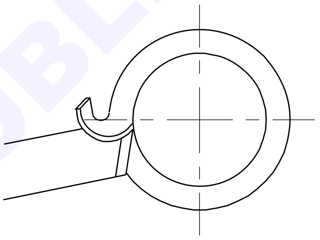
$$\text{min. } V = \frac{\pi \times DN}{1000} \text{ ; } n = \frac{1000 \text{ V}}{\pi \times D}$$

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

Fig 2



ROUGHING CUT OF 3mm ON A CYLINDER



DIAMETER IS REDUCED BY TWICE THE DEPTH OF CUT

DEPTH OF CUT

Factors governing the cutting speed

Finish required

depth of cut

tool geometry

properties and rigidity of the cutting tool and its mounting

properties of the workpiece material

rigidity of the workpiece

type of cutting fluid used

rigidity of the machine tool

Feed (Fig 3)

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

Factors governing feed

Tool geometry

surface finish required on the work

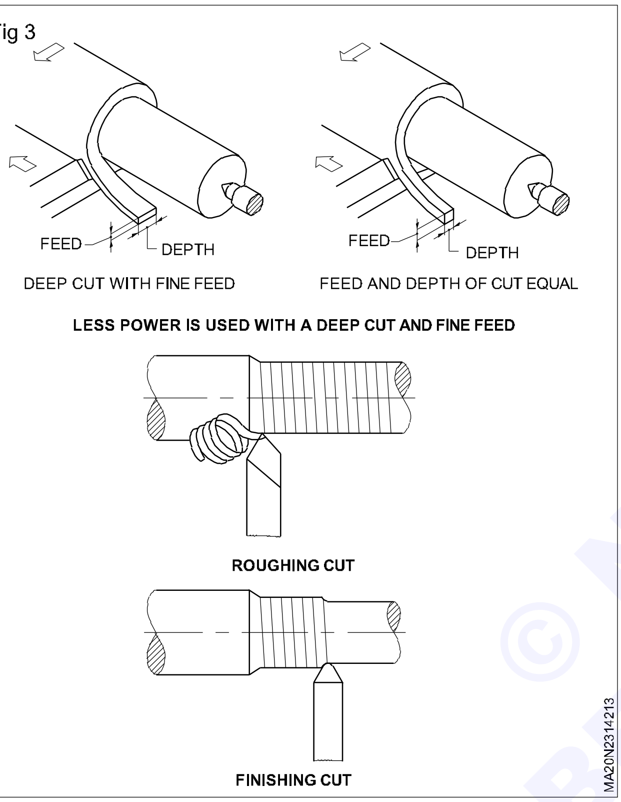
rigidity of the tool

coolant used

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

Cutting speed 120m/min	length of metal passing cutting tool in 1 revolution	Calcualted r.p.m. of spindle
	____78.56mm	1528
	____157.12mm	756
	____235.68mm	509.3

Fig 3



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

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

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

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

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

Depth of cut

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

If D1 = initial diameter

D2 = Final diameter

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

TABLE 1

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

The relationship between limiting spindle speed and constant cutting speed

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

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

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

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

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

E.g., if we want to cut at a constant cutting speed of 250 m/min and limit the RPM to 3000, for Fanuc we would write this

G96 S250

G92 S3000

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

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

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

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

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

Cutting speed-wear life

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

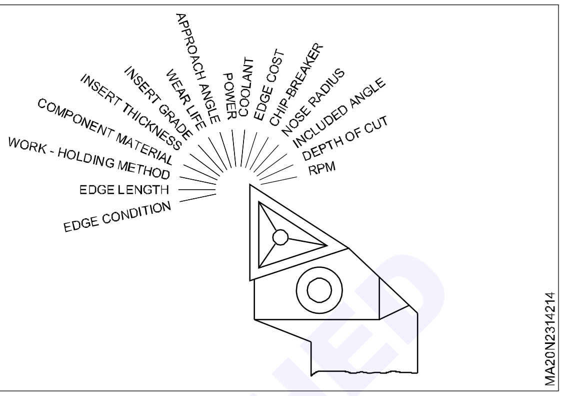
To achieve longer wear life, the cutting speed must be reduced. Multiply the recommended cutting speeds by the following factors.

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

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

Edge condition factors (Fig 4)

Fig 4



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

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

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

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

Selection turning tools from tool catalogues

Objective: At the end of this lesson you shall be able to
- **select turning tool depending on the machining parameters.**

First step

Define material and type of operation

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

Second step

Define application and machining conditions

Locate first choise of insert geometry and grade by application.

F - Finishing

M - Medium

R - Rough

Conditions

O - Good

S - Normal

S - Difficult

Third step

Chose insert with recommended cutting data

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

Fourth step

Choose tool holder

Select the tool holder using the insert shape and size

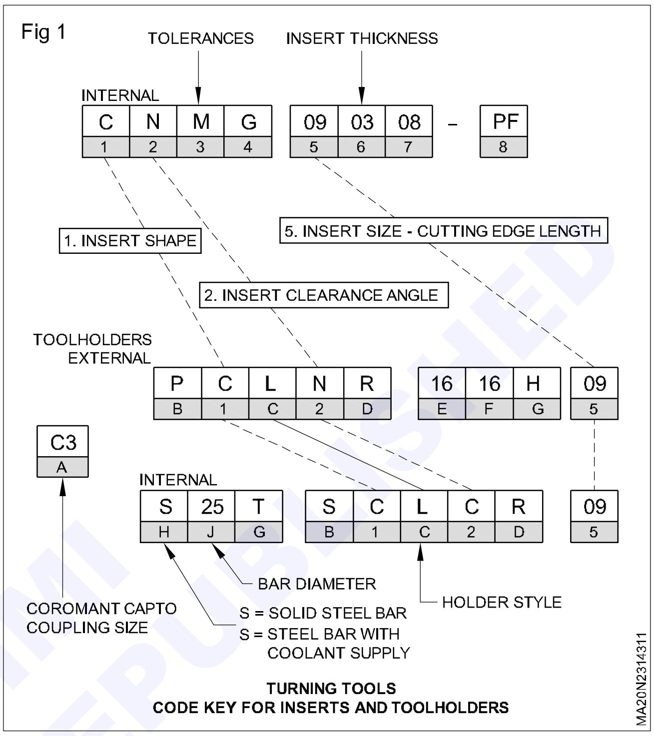
External machining (Fig 2)

CoroTurn® RC

External machining, from roughing to finishing

CoroTurn® TR

Fig 1



TURNING TOOLS
CODE KEY FOR INSERTS AND TOOLHOLDERS

First choice for external profiling

CoroTurn® 107

External machining of small, long and slender components

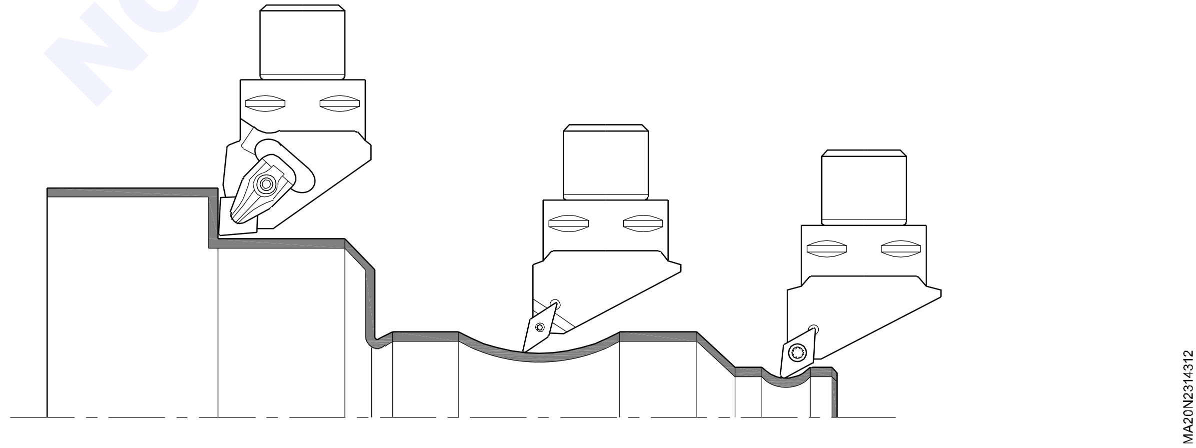
General points to consider

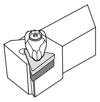
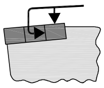
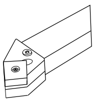
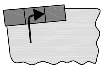
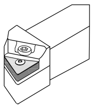
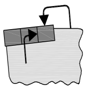
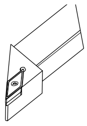
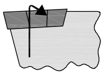
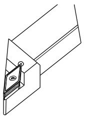
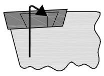
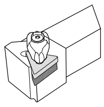
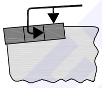
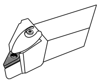
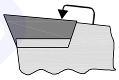
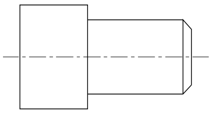
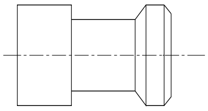
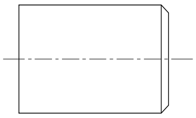
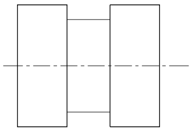
1 Use an entering angle less than 90° (lead angle larger than 0°), if possible, to reduce the impact and the forces.

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

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

Fig 2



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

Internal machining (Fig 3)

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

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

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

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

CoroTurn® 107

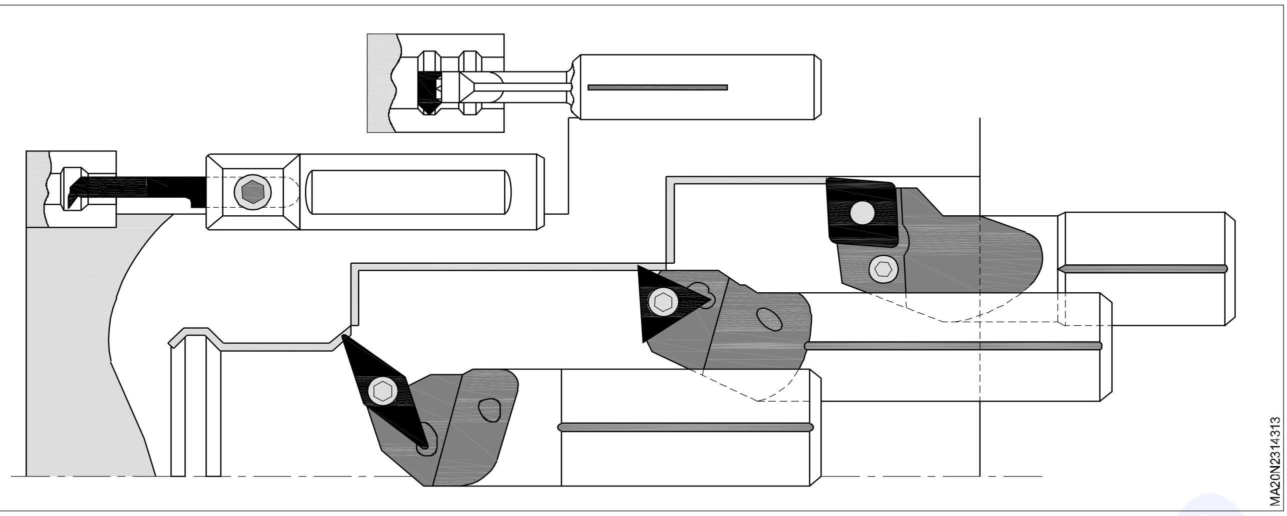
First choice for internal machining of small and medium holes from 6 mm (.236 inch) diameter.

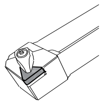
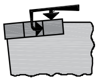
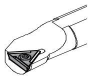
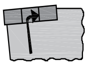
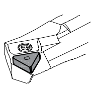
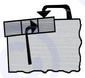
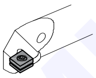
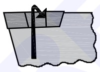
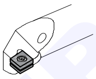
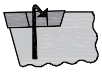
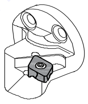
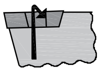
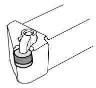
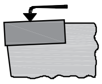
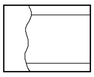
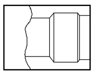
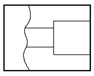
General points to consider

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

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

Fig 3



	Negative basic-shape inserts			Positive basic-shape inserts		Ceramic and CBN inserts	
Tooling system	CoroTurn® RC	T-Max P		CoroTurn® 107	CoroTurn TR 111	CoroTurn® TR	T-Max®
Coromant Capto®	A261	A263	A266	A280	-	-	-
Shank holder	A269	A273	A275	A286	A309	-	A319
cutting units	I21	I16	-	I27	I32	I18	-
	 Rigid clamp design 	 Lever design 	 Wedge clamp design 	 Screw clamp design 	 Screw clamp design 	 Screw clamp design 	 Top clamp design 
Longitudinal turning/ facing 							
Profiling 							
Facing 							

Search for tools in an online databse

Machining cloud is an independent provider of CNC cutting tool product data. A single source of access to the most current product data from a variety of suppliers, in digital format, available from your desktop. Machining cloud provides the most up-to-date information, directly obtained from the manufactureres. The data is formatted very closely to each manufacturere's catalogue system so it familiar to users.

- Download the machining cloud app
- Install the machining cloud app

Search for cutting tools in the machinign cloud online databases

The machining cloud has partnered with several cutting tool manufacturers. This lesson will show you how to select a tool manufacturer and search for tools in their onlune database. You are not restircted to one manufacturer in the machining cloud. You will see how to select cutting tools from different tool manufactureres and add them to your personal tool list.

Search for an O.D. roughing tool

Select the holder with the 1" x 1" x 6" long shank and click add to tool assembly.

Writing part program as per drawing and verifying programme using simulator

Objectives: At the end of this lesson you shall be able to
- **list the steps in planning and part programming for CNC turning**
- **write the manual CNC part program as per drawing dimension**
- **verify the program in CNC simulator.**

Manual CNC programming is a traditional and the most tedious approach which requires the programmer to be aware of the machine's responses by anticipating the program's outcome.

However, if programmer follow the steps given below it will be easy to write the CNC part program

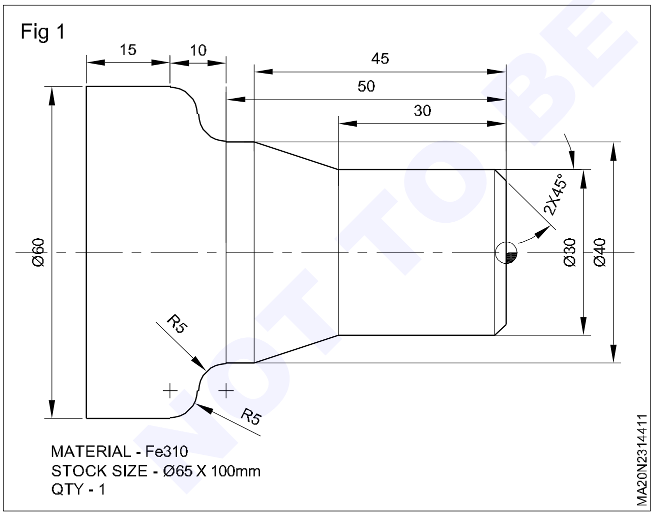
Steps in planning and programming

- Read the drawing and determining the machining process and its control
- List out he operations required
- Selecting the appropriate tool for machining
- Determining the cutting parameter
- Deciding the work holding method
- Fixing the programming zero on the part
- Writing the part program using G codes and M codes
- Verify the program in a cnc simulator

Example

Part drawing (Fig 1)

Fig 1



Planning

Select the turning centre with Fanuc control

Operation required rough profile turning with canned cycle

Work holding method: - 3 jaw self centring chuck

Tool required external rough turning carbide tool turning

Cutting parameter: -

Programming zero fixed at the left side face centre

Write the part program using G71 outer diameter roughing cycle

Verify the program in cnc simulator

Programming

```
O1234; (PROFILE TURNING PROGRAMMING)
N1 G90 G21 G80 G40;
N2 TO3O3;
N3 S750 M04;
N4 G00 X100.0 Z20.0;
N5 G71 P6 Q14 U4.0 W2.0 D4.0 F0.3 S550;
N6 G01 X26.0 Z0.0 F0.15;
N7 G01 X30.0 Z-2.0;
N8 G01 X30.0 Z30.0;
N9 G01 X40.0 Z-45.0;
N10 G01 X40.0 Z-50.0;
N11G02 X50 Z-55.0 I5.0 K0.0;
N12 G03 X60.0 Z-60.0 I0.0 K-5.0;
N13 G01 X60.0 Z-75.0;
N14 G01 x 65.0 Z -75
N15 G00 X100.0 Z100.0 M05;
N16 G28 U0.0 W0.0 T0300;
N17 T0606 M04 S800;
N18 G00 X100.0 Z5.0;
N19 G70 P13 Q14 F0.1;
N20 G00 X100.0 Z100.0 M05;
N21 G28 U0.0 W0.0 T0600;
N22 M30;
```

- Enter the program in CNC simulator
- Verify it for its correctness.

Collisions due to improper machine setup and operation

Objectives : At the end of this lesson you shall be able to
- **identify the collision reason**
- **identify to axis of collision**
- **recover from collision.**

Collisions due to program error, tool brackage

Full collision avoidance can be provided in the machine design phase, the full cycle simulation is combined with real - time monitoring of all moving and stationery components on the machine.

STL files to model all the machine assemblies and structures relevant to determine full collision avoidance plus this can be done in all operating modes, including JOG/MDI and AUTO.

In the shop using the operating program on the CNC can further identify potential collisions when altering a program changing a rotary table dimension, resetting fixture or simply monitoring the tooling changes Basic machining areas and limits can be defined by modeling those protected areas. Head stock, rotary table trunnions, or a simple clamp position can be monitored in this real time collision.

All the stationary and moving elements, as well as the cutting edges, are effectively monitored for potential collision. The affected area or machine components are quickly Identified on screen during a cycle simulation or in real time.

Human error can also be factored in so that the resulting collision that would have been caused by - a change in a program or a set-up error can be anticipated, triggering an alert that is high light the affected area on screen.

Faulty CNC Machine Programming

CNC machining is not exactly a foolproof process. In some instances, either a work piece or the machine itself could be directed in a harmful way. When this happens, a crash might occur, whereby tools or machine parts end up broken. The tools that could get damaged by a crash may include the vices or clamps that hold the work piece in place. When damage occurs within the machine, it could range from minor screw breakage to serious structural deformity.

The fact is, CNC equipment lacks the sentience to know exactly which distances are too far. Therefore, the tools must be exactly programmed in order to work without fault. If a program code is miscalculated, a CNC machine could be driven outside its physical bounds and cause an internal collision. Even though most of today's CNC machines are manufactured with parameter boundaries, these inputs can be manipulated by operators.

Likewise, CNC tools are oblivious to a given environment. While certain CNC machines are equipped with spindle load sensing, others lack this feature. In the latter case, the software must be coded properly to ensure nothing goes off parameter, otherwise a crash would be the likely outcome. Even if a CNC machine is equipped with load sensors, a crash could still occur. When a tool function goes astray, it's up to the operator to rectify the situation.

Crash Prevention on Different Types of CNC Machines

With the installation of encoder-disk position sensors, the possibility of a crash can be detected in advance and thwarted. Alternately, torque sensors can help determine whether a CNC machine is moving as intended and also detect unwanted cutting.

In garage CNC systems, tools are reliant on the rotational precision of stepper motors for the correct number of degrees. To monitor the tool position, the pulses that go to the stepper must be counted, because in most cases, there's no form of alternate monitoring.

On industrial CNC machines, closed-loop controls are employed, whereby the control always knows the axis position. If properly controlled, the potential for crashes is significantly lowered, though it's still the responsibility of programmers to see that codes are inputted accurately for utmost safety.

Over the last two decades, CNC software has advanced to where a vast range of machine tools-axes, clamps, fixtures, spindles, turrets-can be based precisely on 3D solid models. With those specs programmed into the code, it's easier to determine whether a crash will occur with a particular cycle.

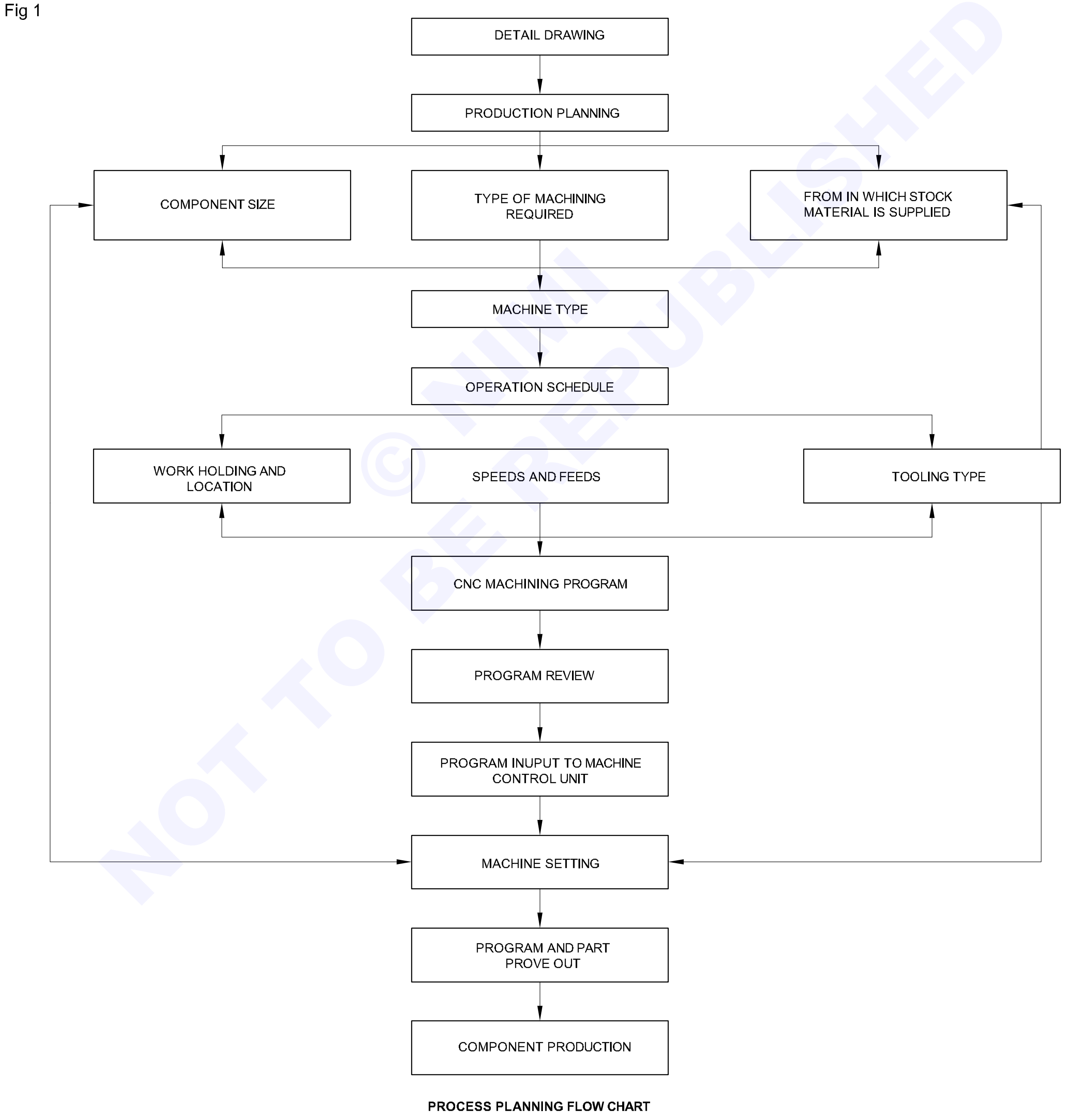
Process planning, sequencing tool layout & selection and cutting parameter

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

Process planning (Fig 1)

Process planning flow chart

Fig 1



PROCESS PLANNING FLOW CHART

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

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

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

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

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

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

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

Types of process planning

The Types are

1 Manual process planning

2 Automated process planning

3 Generative process planning.

Machining sequence

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

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

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

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

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

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

Tool layout

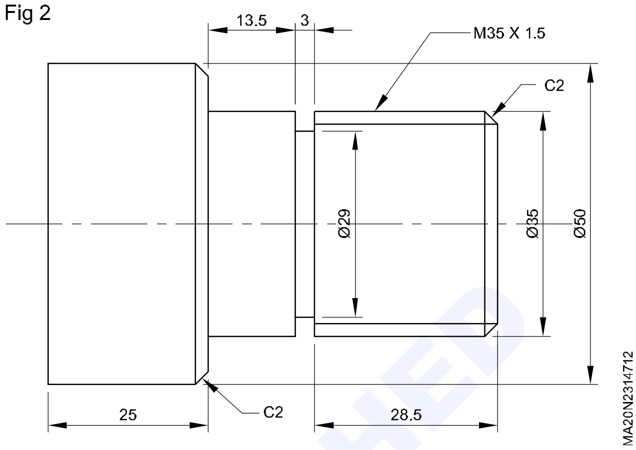
The tool layout for a job constitutes the predetermined plan for machining operation of a particular component the layout is dependent upon the number of pieces to be manufactured in lot size.

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

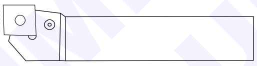
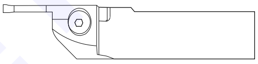
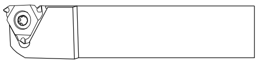
For preparation of the tool layout, it is necessary to have the finished drawing of the part to be machined and if is a forging or casting will determine how much machining has to be done

Example of tool layout (Fig 2)

Fig 2



Tool layout

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

Cutting parameters

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

Cutting speed

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

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

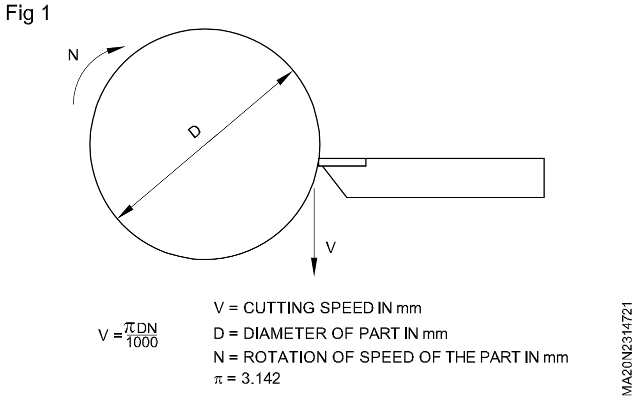
V = Cutting speed in m/min

D = Diameter of part in mm.

N = rotation speed of the part in rpm

π = 3.142

Fig 1



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

Fig 2

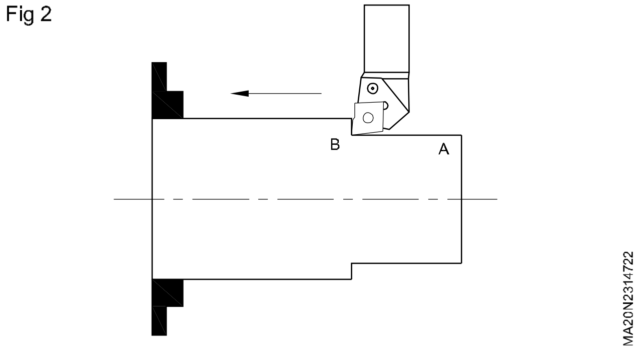


Fig 3

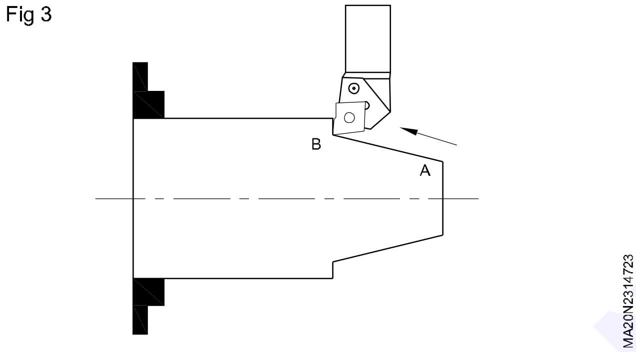
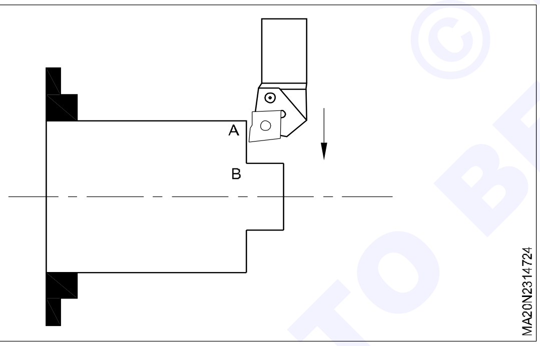


Fig 4



Spindle speed

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

Feed Rate

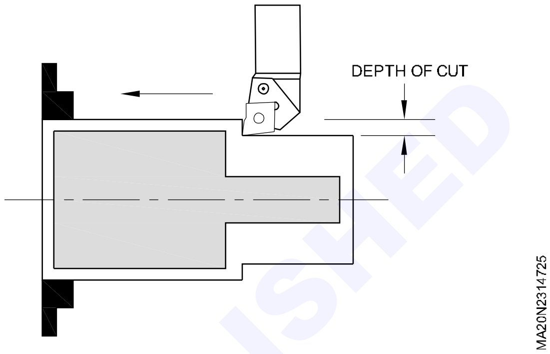
Feed rate is the rate of linear travel of the tool. It is programmed as the distance travelled per minute or per revolution. On CNC lathes it is usually programmed as the mm/rev. The relationship between feed rate per minute and feed rate per revolution is:

Feed per minute= Spindle speed X Feed per revolution

Depth of cut

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

Fig 5



Tool life

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

Surface speed

Feed rate

Depth of cut

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

Cutting parameters for CNC turning applications

Speed / feed /depth of cut for general turning

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

Speed / feed / width of cut for grooving

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

Threading

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

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

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

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

Work and tool offsets

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

- **state about work and tool offset**
- **state wear offset.**

Work offset

Program coordinates are specified with reference to a workpiece zero position, which is usually a point on the workpiece. The machine, however, internally only understands coordinates with reference to a fixed machine zero point. The work offset is the distance from the work zero to the machine zero. It is fed into the machine's memory, and enables the machine to translate the program coordinates into machine coordinates.

Work offset values can be input into the work co ordinate page of the CNC lathe given below (G54 To G59)

Work coordinates (G 54)

No		Data	No		Data
000	X	0.000	002	X	0.000
EXT	Z	0.000	G55	Z	0.000
No		**Data**	**No**		**Data**
001	X	0.000	003	X	0.000
G54	Z	0.000	G56	Z	0.000

Tool offset

Tool offset is used to compensate for the difference when the tool actually used differs from the imagined tool used in programming usually, standard tool.

Tool offset valves can be input into the offset/ geometry page of the CNC lathe given below in table - 1

Geometry **Table 1**

Tool	X	Z	R	T
No	X offset value	Z offset value	Tool nose radius	Tool type no
G01				
G02				
G03				
G04				
G05				
G06				
G07				
G08				
G09				
G10				
G11				
G12				
G13				
G14				
G15				

Wear offset

Wear values in X(U) and Z(W) are incremental displacement of tool from the actual offset value and can be input into the offset /wear page of the CNC lathe given below in tabe 2

Tabe 2

Tool	U	W	R	T
No	Tool wear X	Tool wear in Z	Nose radius wear	Tool type No.
W01				
W02				
W03				
W04				
W05				
W06				
W07				
W08				
W09				
W10				
W11				
W12				
W13				
W14				
W15				
W 16				

Turning in multiple setups hard and soft jaws, soft jaw boring

Objectives : At the end of this lesson you shall be able to
- **explain the steps in boring the soft jaws to required size**
- **state the uses of follower and steady rest.**

How to bore out soft jaws

Boring and truing soft jaws

When you're ready to bore or turn a diameter on your soft jaws to clamp your workpiece the following simple guidelines may be useful.

Step 1

Prepare a disc for clamping the chuck making sure the wall thickness and material strength is strong enough to take the chuck pressure. You will need different sizes to cover different work piece and chuck diameters. Make sure the soft top jaws are correctly fitted in position on the base jaws.

Step 2

Actuate the chuck so the base jaws are open and set the diameter to be gripped around the middle of the maximum jaw stroke.

Step 3

Place the disc/stopper on the chuck face making sure when you clamp up that the stopper is totally flat against the face. Clamp the stopper.

Step 4

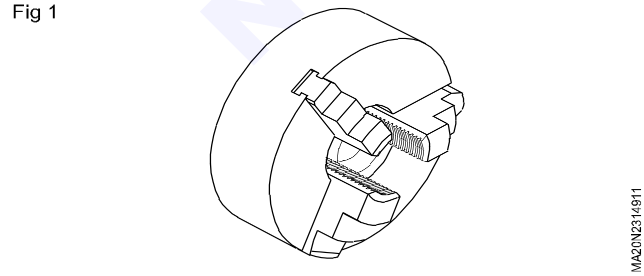
Now while gripping the stopper, bore out the soft jaws to the desired diameter and finish required to grip the work piece. Ideally the gripping pressure should be set to the same pressure, where the work piece will be clamped at.

Step 5

After the jaws have been bored out, do a test to make sure the workpiece is clamped securely and the accuracy is corrected before continuing to machine the workpiece.

Turning soft jaws is very similar except you will need ring instead of a stopper to clamp the jaws for turning. The chuck jaws must have a diameter turned of the ring usually at the back of the jaws. Once this is machined, the ring an be turned clamped and the required diameter of workpiece.

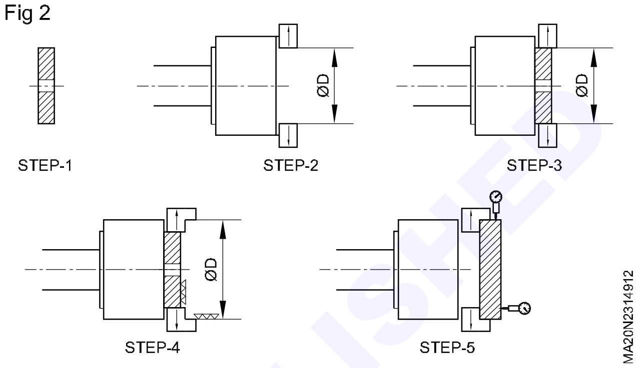
Fig 1



Purpose of using soft jaws

Soft jaws are used to hold parts that are thin compared their diameter. (e.g., 25 mm diameter disc with 3 mm thickness) If the thickness is less than the job it would tend collapse. To avoid this chuck fitted with soft jaws are used.

Fig 2



Steady Rest

The steady rest or center rest, as it is also called, is used to support long workpieces or shafts being machined between centers or for boring operations. It is also used for internal threading operations where the workpiece projects a considerable distance from the chuck or faceplate. The steady rest is clamped to the lathe bed at the desired location and supports the workpiece within three adjustable jaws. The rest prevents the workpiece from springing under cut, or sagging as a result of the otherwise unsupported weight.

The workpiece must be machined with a concentric bearing surface at the point where the steady rest is to be applied. The jaws must be carefully adjusted for proper alignment and locked in position. The area of contact must be lubricated frequently. The top section of the steady rest swings away from the bottom section to permit removal of the workpiece without disturbing the jaw setting.

Follower Rest

The follower rest is used to back up a workpiece of small diameter to keep it from springing under the stress of the cutting operation. The follower rest gets its name because it follows the cutting tool along the workpiece. The follower rest has one or two jaws that bear directly on the finished diameter of the workpiece opposite and above the cutting tool. The rest is bolted to the lathe carriage so that it will follow the cutter bit and bear upon that portion of the workpiece that has just been turned. The cut must be started and continued for a short longitudinal distance before the follower rest is applied. The rest is generally used only for straight turning or threading long, thin workpieces.

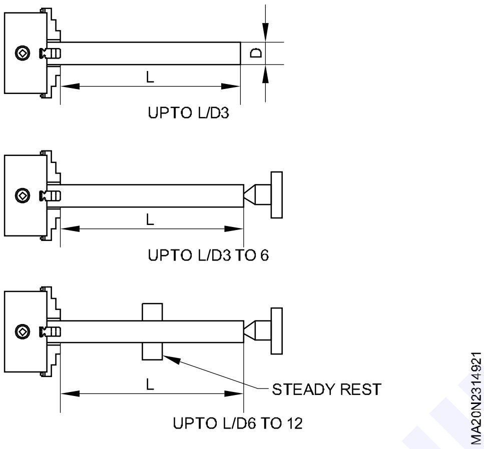
Length to diameter (L/D) ratio and deciding work holding parameter

Objectives : At the end of this lesson you shall be able to
- **decide to hold the job**
- **length to diameter ratio base deciding.**

Length to Diameter (L/D) ratio and deciding work holding based on it.

The Job held in the chuck is cantilever and the radial cutting force of the tool tends to bend the part. You use can a tail stock or steady rest to prevent the bending. Here's a thumb rule that tells you when you can hold in a chuck, when to use a tail stock, and when to use a steady rest. Based on the L/D ratio of the part.

Fig 1



Chuck only

Use it L/D is Less than 3 you can go up to L/D 5 with reduced cuttting parameter which reduce the cutting force.

Tail stock

Use it L/D if between 3 and 6 you can go upto L/D 10 with reduced cutting parameter.

Steady rest

Use if L/D is between 6 & 12 you can go upto L/D 20 with reduced cutting parameter.

L/D ratio	Chuck	Tail Stock	Steady rest
Upto 3	Normal cutting parameters recommended by tool manufacturing.		
3 to 5	Reduce depth of cutting feed rate. Use fine nose radius.		
Upto 6		Normal cutting paramter recommended by tool manufacture.	
Upto 6 to 10		Reduce depth of cut dry feed rate use smaller nose radius.	
Upto 12			Normal cutting parameter recommended by tool manufacturing.
12 to 20			Reduce depth of cut and feed rate use smaller nose radius.

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

Objectives : At the end of this lesson you shall be able to
- **list the different modes in CNC lathe**
- **explain the steps in each mode of operation.**

Operational modes (Fig 1)

Jog mode

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

Incremental jog mode

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

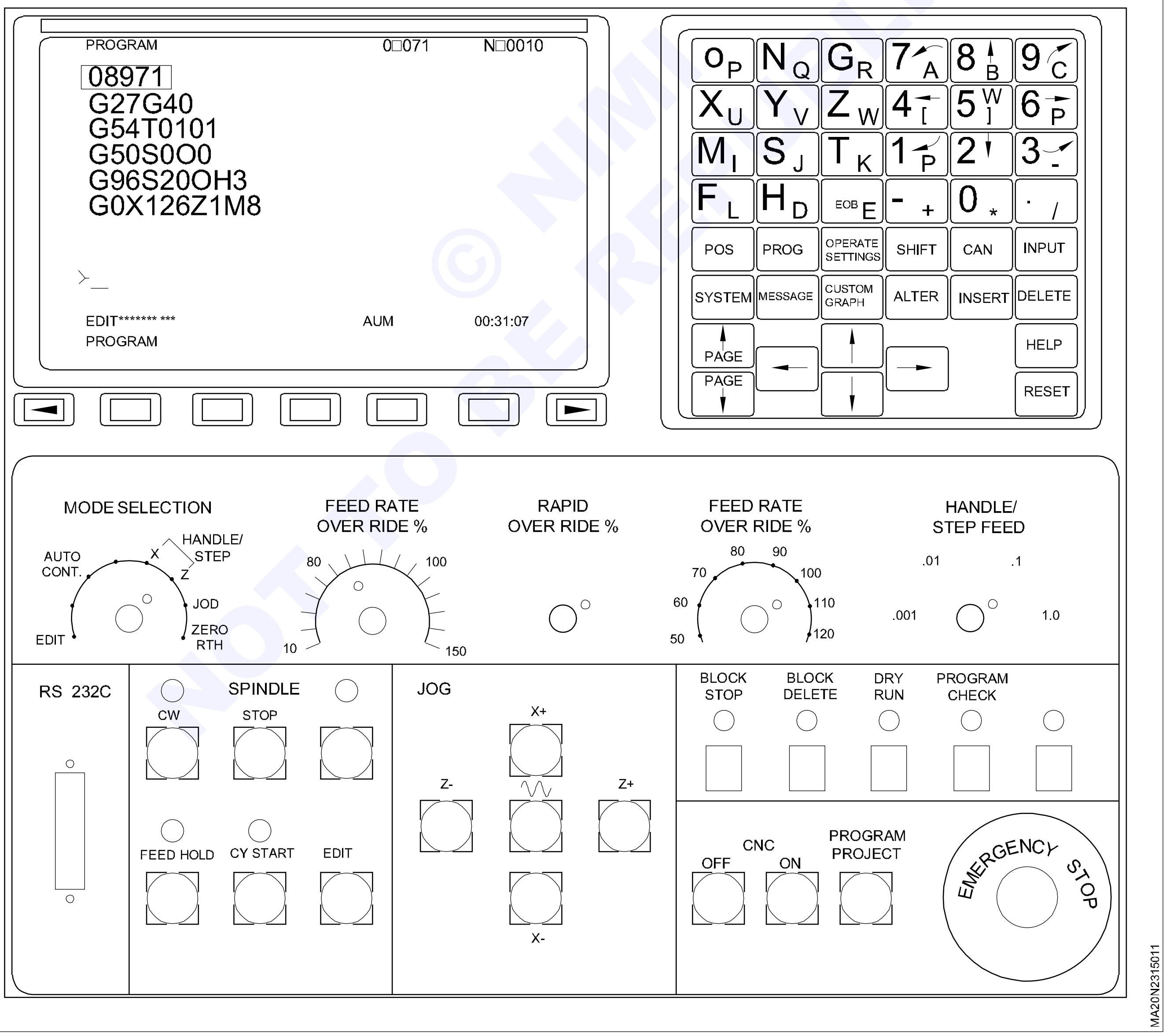
Edit mode

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

Manual mode: MDI mode

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

Fig 1



```
PROGRAM                    O□071      N□0010
08971
G27G40
G54T0101
G50S0O0
G96S20OH3
G0X126Z1M8

>_

EDIT******* ***            AUM        00:31:07
PROGRAM
```

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

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

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

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

Coolant ON/OFF Controls the coolant.

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

Chip conveyor forward backward Moves the chip conveyor.

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

Machine lock and auxiliary function lock

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

Manual pulse generator

A Manual Pulse Generator (MPG) is a device for generating electrical pulses in electronic system under the control of human operation.

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

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

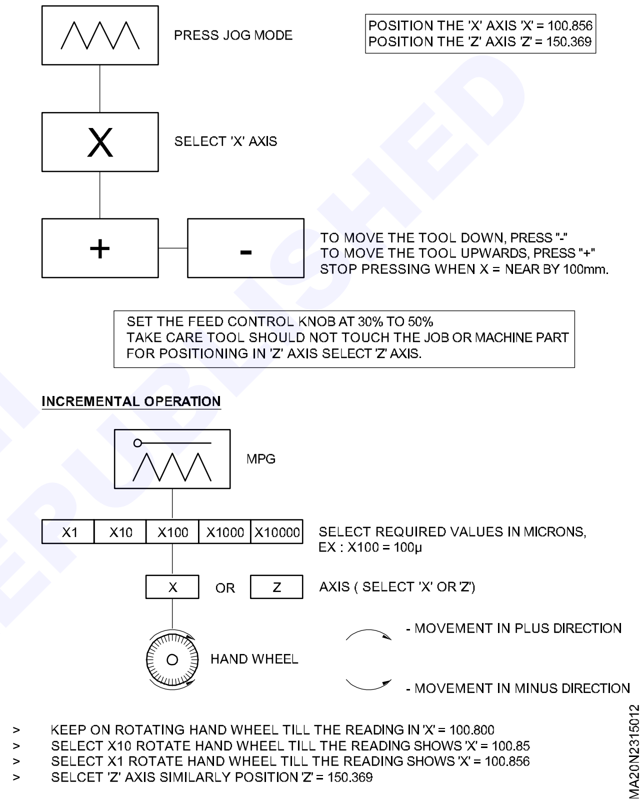
Procedure for JOG feed

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

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

Procedure for INCREMENTAL feed (Fig 2)

Fig 2



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

Procedure for MANUAL HANDLE feed

- Press the HANDLE switch.
- Select the axis.
- Select the incremental value.
- Move the tool along selected axis by rotating the handle 360 degrees moves the tool the distance equivalent to 100 graduation.

The instructor will demonstrate the various axis and models.

Entering and editing programs

Objective : At the end of this lesson you shall be able to
- **explain the steps in entering, inserting, altering and deleting program.**

Creating program

Steps

Enter the **EDIT** mode.

Press the **PROG** key.

Press address key O and enter the program number

Press the **INSERT** key.

Comments cane be written in a program the control in/out codes

Example O0001 (FANUC ERIES 16);

M08 (COOLANT ON);

- When the **INSERT** key is pressed after the control out "(", comments, and control in code ")" have been typed, the typed comments are resgistered.

Inserting a word

A word is an address followed by a number.

- Key in an address to be inserted.
- Key in data
- Pess the **INSERT** key.

Altering a word

- Position the cursor on the word, which is to be altered
- Key in the word
- Press the **ALTER** key

Example

Altering T15 to M04

Program

O0054

N1 G90 G40 G80;

N2 T0101;

N3 G00 X100 Z100 **T15**

N4 G01 X0 20 F01;

M5 M02;

Key in **M 0 4**

Press **ALTER** key

Program

O0054

N1 G90 G40 G80;

N2 T0101;

N3 G01 X0 Z0 **M 0 4**;

Deleting a word

Position the cursor, on the word is to be deleted

Press the **DELETE** key

Deleting a block

Position the cursor, on the block number that is block to be delted

A line in the program is called block

- Key in **EOB**
- Press the **DELETE** key
- The entire block will be deleted

Deleting one program

- Select **EDIT** mode
- Press **PROG** to display the program screen
- Key in address O
- Key in a desired program number
- Press the **DELETE** key

The program with entered program number is deleted.

Entering offset data in offset page

Objectives : At the end of this lesson you shall be able to
- **describe the method of entering the work off set value in offset page**
- **explain the method for entering tool offset and tool wear off set in tool offset page**
- **brief the steps involved in enabling and disabling the setting data.**

Displaying and Setting the Work piece Origin Offset Value

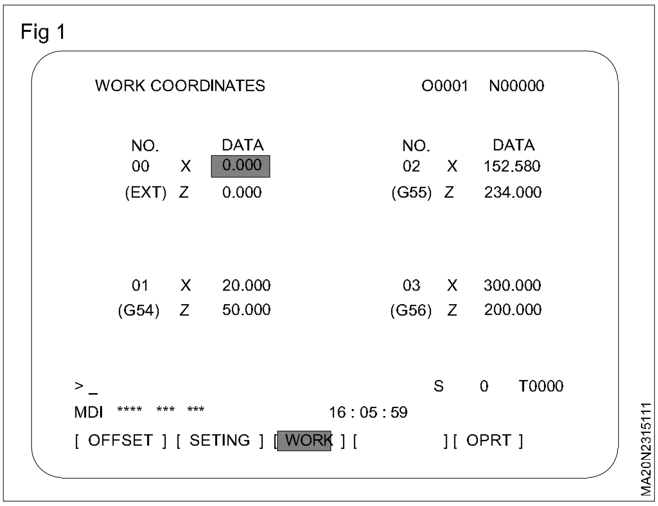
Displays the work piece origin offset for each work piece coordinate system (G54 to G59) and external work piece origin offset. The work piece origin offset and external work piece origin offset can be set on this screen.

Press function key **OFFSET SETTING**

Press chapter selection soft key **[WORK]**

The workpiece coordinate system setting screen is displayed (Fig 1)

Fig 1



```
WORK COORDINATES                        O0001    N00000

     NO.        DATA                 NO.        DATA
     00    X    0.000                02    X    152.580
    (EXT)  Z    0.000               (G55)  Z    234.000


     01    X    20.000               03    X    300.000
    (G54)  Z    50.000              (G56)  Z    200.000


> _                                       S    0    T0000
MDI   ****   ***   ***          16 : 05 : 59
[ OFFSET ] [ SETING ] [ WORK ] [            ] [ OPRT ]
```

The screen for displaying the workpiece origin offset values consists of two or more pages. Display a desired page in either of the following two ways

Press the page up ↑ or page down ↓ key.

Enter the workpiece coordinate system number (0: external workpiece origin offset, 1 to 6: workpiee coordinate systems G54 to G59 and press operation selection softkey [NO.SRH].

Turn off the data protection key to enable writing

Move the cursor to the workpiece origin offset to be changed.

Enter a desired value by pressing numeric keys, then press softkey [INPUT]. The entered value is specified in the workpiece origin offset value. Or, by entering a desired value with numeric keys and pressing softkey [+INPUT], the entered value can be added to the previous offset value.

Repeat to change other offset values.

Turn on the data protection key to disable writing.

Setting and displaying the tool offset value

Precedure for setting displaying the tool offset value and the tool nose radius compensation value

Press function key

Press chapter selection soft key [OFFSET] or press serveral times until the tool compensation screen is displayed.

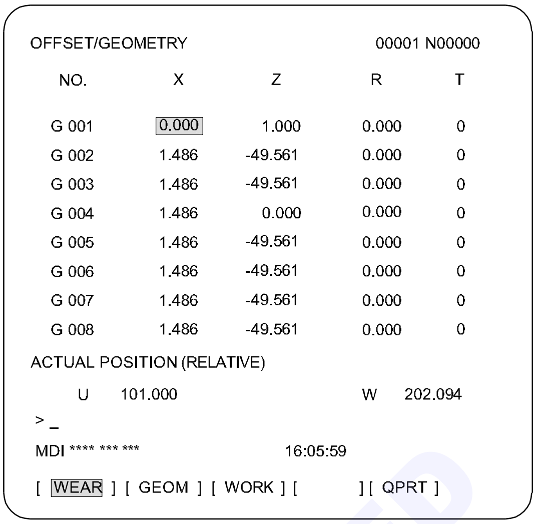
Pressing soft key [GEOM] display tool geometry compensation values. (Fig 2)

Pressing soft key [WEAR] displays tool wear compensation values. (Fig 3)

Move the cursor to the compensation value to be set or changed using page keys and cursor keys, or enter the compensation number for the compensation value to be set or changed and press soft key [NO.SRH].

To set a compensation value, enter a value and press softkey [INPUT]. To change the compensation value, enter a value to add to the current value (a negative value to reduce the current value) and press softkey [+INPUT]. Or, enter a new value and press softkey [INPUT]. TIP is the number of the virtual tool tip. TIP may be specified on the geometry compensation scren or on the wear compensation screen.

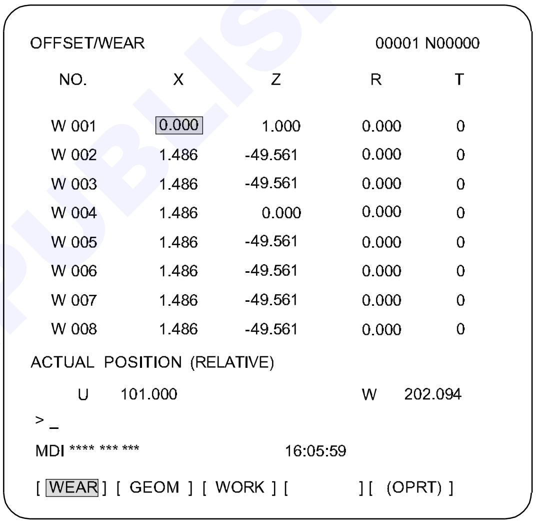
Fig 2



```
OFFSET/GEOMETRY                              00001 N00000
    NO.          X           Z           R         T

   G 001       0.000       1.000       0.000       0
   G 002       1.486     -49.561       0.000       0
   G 003       1.486     -49.561       0.000       0
   G 004       1.486       0.000       0.000       0
   G 005       1.486     -49.561       0.000       0
   G 006       1.486     -49.561       0.000       0
   G 007       1.486     -49.561       0.000       0
   G 008       1.486     -49.561       0.000       0
ACTUAL POSITION (RELATIVE)
      U      101.000                    W     202.094
> _
MDI **** *** ***                  16:05:59
[ WEAR ] [ GEOM ] [ WORK ] [          ] [ OPRT ]
```

WITH TOOL WEAR OFFSET

Fig 3



```
OFFSET/WEAR                                  00001 N00000
    NO.          X           Z           R         T

   W 001       0.000       1.000       0.000       0
   W 002       1.486     -49.561       0.000       0
   W 003       1.486     -49.561       0.000       0
   W 004       1.486       0.000       0.000       0
   W 005       1.486     -49.561       0.000       0
   W 006       1.486     -49.561       0.000       0
   W 007       1.486     -49.561       0.000       0
   W 008       1.486     -49.561       0.000       0
ACTUAL  POSITION  (RELATIVE)
      U      101.000                    W     202.094
> _
MDI **** *** ***                  16:05:59
[ WEAR ] [ GEOM ] [ WORK ] [          ] [ (OPRT) ]
```

WITH TOOL WEAR OFFSET

Displaying and Entering Setting Data

Data such as the TV check flag and punch code is set on the setting data screen. On this screen, the operator can also enable/disable parameter writing, enable/disable the automatic insertion of sequence numbers in program editing, and perform settings for the sequence number comparison and stop function

Procedure for setting the setting data

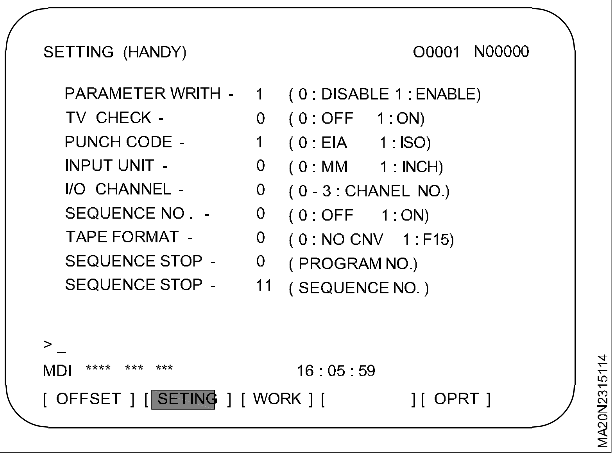
Select the MDI mode.

Press function key

Press soft key [SETING] to display the setting data screen. This screen consists of several pages.

Press page key or until the desired screen is displayed. An example of the setting data screen is shown in Fig 4&5.

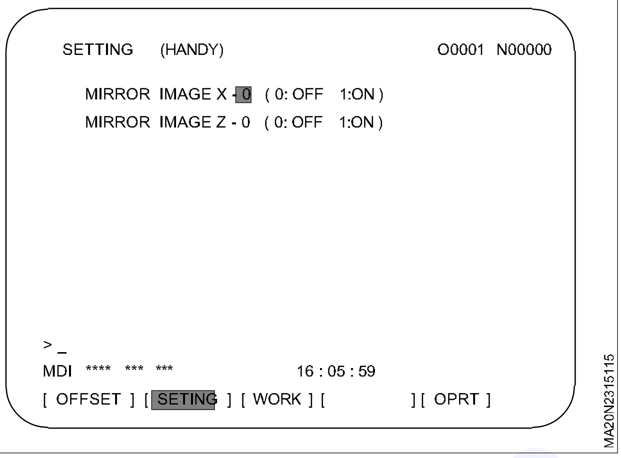
Fig 4



```
SETTING  (HANDY)                              O0001   N00000

   PARAMETER WRITH -   1    ( 0 : DISABLE 1 : ENABLE)
   TV  CHECK -         0    ( 0 : OFF     1 : ON)
   PUNCH CODE -        1    ( 0 : EIA      1 : ISO)
   INPUT UNIT -        0    ( 0 : MM       1 : INCH)
   I/O  CHANNEL -      0    ( 0 - 3 : CHANEL  NO.)
   SEQUENCE NO . -     0    ( 0 : OFF      1 : ON)
   TAPE FORMAT -       0    ( 0 : NO CNV    1 : F15)
   SEQUENCE STOP -     0    ( PROGRAM NO.)
   SEQUENCE STOP -     11   ( SEQUENCE NO. )


> _
MDI   ****  ***  ***             16 : 05 : 59
[ OFFSET ] [ SETING ] [ WORK ] [             ] [ OPRT ]
```

MA20N2315114

Fig 5



```
SETTING     (HANDY)                           O0001  N00000

   MIRROR  IMAGE X - 0   ( 0: OFF   1:ON )
   MIRROR  IMAGE Z - 0   ( 0: OFF   1:ON )


> _
MDI   ****  ***  ***             16 : 05 : 59
[ OFFSET ] [ SETING ] [ WORK ] [             ] [ OPRT ]
```

MA20N2315115

Move the cursor to the item to be changed by pressing cursor keys. ↑ ↓ ← →

Enter a new value and press soft key [INPUT]

Contents of settings

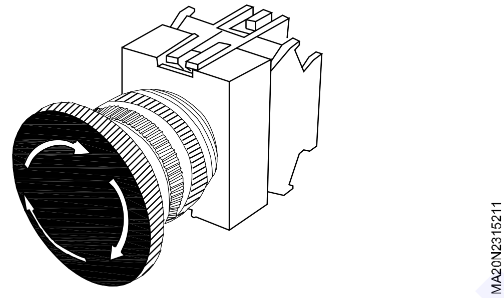
Parameter write	Setting whether parameter writing is enabled or disabled 0: Disabled 1: Enabled
TV check	Setting to perform TV check 0: No TV check 1: Perform TV check
Punch code	Setting code when data is output through reader puncher interface 0: ELA code output 1: ISO code output
Input unit	Setting a program input unit, inch or metric system 0: Metric 1: Inch
I/O channel	Using channel of reader/ puncher interface 0: Channel 0 1: Channel 1 2: Channel 2
Sequence No.	Setting of whether to perform automatic insertion of the sequence number or not at program edit in the EDIT mode. 0: Does not perform automatic sequence number insertion 1: Perform automatic sequence number insertion
Tape format	Setting the F10/11 tape format conversion 0: Tape format is not converted 1: Tape format is converted See programming for the F10/11 tape format
Sequence stop	Setting the sequence number with which the operation stops for the sequence number comparison and stop function and the number of the program to which the sequence number belongs
Mirror image	Setting of mirror image ON/OFF for each axes 0: Mirror image off 1: Mirror image on
Others	Page key ↑ or ↓ can also be pressed to display the SETTING (TIMER) screen.

Functions of important keys / knobs in CNC machines

Objectives : At the end of this lesson you shall be able to
- **state the uses of emergency switch in CNC machine**
- **list the function of reset key in CNC machine**
- **brief the uses of feed rate and speed rate override knobs**
- **state the uses s of machine lock and program lock function in CNC machine.**

Emergency switch (Fig 1): The purpose of an emergency push button is to stop the machinery quickly when there is a risk of injury or the workflow requires stopping. It is intended to avert harm or to reduce existing hazards to people, machinery, or working, so people also says safety push button switch or push button safety switches

Fig 1

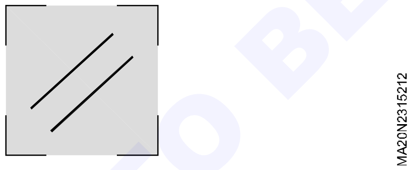


Reset (Fig 2)

When the Reset key is pushed, part program execution is stopped and program execution returns to the top of the program.

The program will not restart without the NC Start key being pressed.

Fig 2



Feed rate override (Fig 3): Normally you can control speed override and feed, override from 0% to 120%. At 0% speed override the spindle will stop rotating, and at 0% feed override the tool will stop working (the tool will be stationary).

No doubt 120% feed override and speed override is just safe. But some CNC machines give even more flexibility 0% to 200%. Normal CNC machine has just 0% to 120% feed override and speed override.

Speed Override (Fig 4): Whenever handling the speed override and feed override always think about safety, safety of yourself, tool, machine, component and your surroundings. Never try to use the speed override and feed override if you don't need it. Because when CNC programs are made the speed and feed is properly set for the machined component. The increase in feed or speed might break insert or even tool.

Fig 3

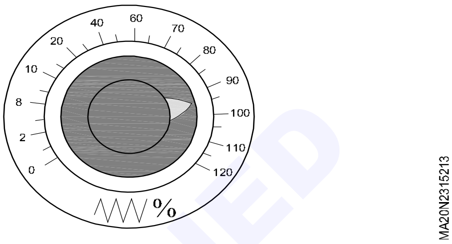
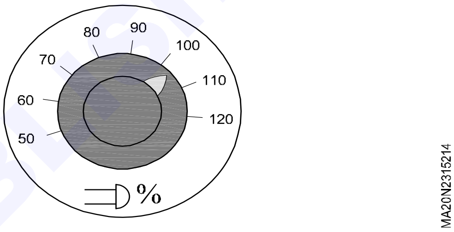


Fig 4



CNC Machine Lock

CNC machine lock is very handy function for testing a CNC program before actually making a part.

In machine lock axis stay stationary on their positions, but their positions change with program on the display, this way CNC machinists can figure out any problems before making the part.

There are two types of machine lock:

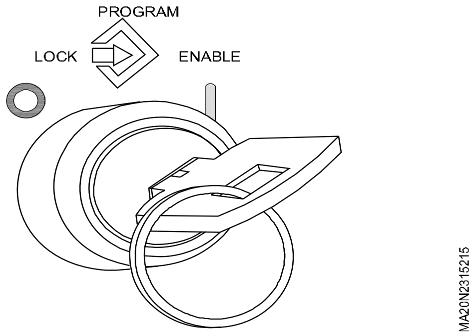
all-axis machine lock, which stops the movement along all axes.

specified-axis machine lock, which stops the movement along specified axes only.

To activate machine lock, press the machine lock switch on the operator's panel. The tool does not move but the position along each axis changes on the display as if the tool were moving.

Program lock (Fig 5): The program lock function in CNC machine is to enable and disable the editing of pat program.

Fig 5



Proving the CNC program

Objectives: At the end of this lesson you shall be able to
- **state the steps in proving the CNC part program**
- **state the tool path simulation and its steps**
- **state the dry run and its steps**
- **execute the program in single block mode.**

Running the first part

CNC machine operator usually starts a new job by studying the documentation included with the Program mainly drawing, setup sheet and tooling sheet.

The following steps describe a standard setup procedures, that will remain the same for most jobs.

Set the cutting Tool

Use a tooling sheet or tooling information from the part program

Set the cutting tools into their holders and respective tool station. Register all tool numbering into the control memory. Make sure the tools are sharp and mounted properly in the holders cutting tool check list.

- Are the tools are properly mounted in holder
- Are the proper inserts are used
- Are all the tools are right size
- Are the tools are placed in proper magazine station
- Are the offsets set correctly
- Is there an interference between individual tools
- Are all the tools sharp

Set the fixture/work holding device

A fixture that holds or support the part mounted on the machine,aligned and adjusted if necessary, but the part is not mounted at this stage.

Set the part: Locate the part into the fixture and make sure it is safely mounted. Check for possible interferences and obstacles in the setup.

Part checkup check list

- Is the part mounted safely
- Is the part properly oriented on the chuck/fixture
- Is the part aligned squarely
- Are the clearance sufficient
- Are all the clamps away from the cutting path
- Does the tool change takes place in clear area

Set the tool offsets: Depending on the types of machine, get the tool geometry and wear offsets, tool length offsets and cutter radius offset. This setup includes the work offset setting.

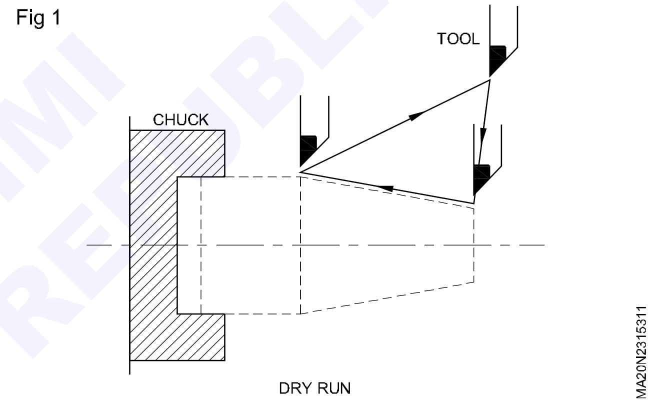
Checking the part program

This step is the first evaluation of the part program. The part may be removed from the fixture temperarily. Since all the offsets are already set in the control. Program is checked accurately with all considerations.

Dry run

The tool is moved at the feed rate specified by a parameter regardless of the feed rate specified in the program. This function is used for checking the movement of the tool under the state that the work piece is removed. (Fig 1)

Fig 1



Step for dry run

Press the dry run switch on the machine operators's panel during automatic operation. The tool moves at the feed rate specified in a parameter. The rapid traverse switch can also be used for changing the feed rate.

Refer to the appropriate manual provided by the machine tool builder for dry run.

Single block operation

Pressing the single block switch starts the single block mode. When the cycle start buttons is pressed in the single block mode, The tool steps after a single block in the program executed. Check the program in the single block mode by executing program block by block

Steps for single operation

Press the single block switch on the machine operator's panel. The execution of the program is stopped after the current block is executed.

Press the cycle start button to execute the next block. The movement stops after the block is executed.

While executing the program in single block the feed override knob should be set to zero.

The block contain the movement the slide will not make but the distance to go is appeared on the screen.

Read the values of the distance in X and Z axes compare the distance to go and the actual work movement. Judge manually their will not be any collision then release the feed hold or increase in feed rate knob.

Similarly execute the program block by block

Make a Trail cut

Trail cut may be required in order to establish whether the programmed speeds and feeds are reasonable or not and if the various offsets are set properly.

At this point, any necessary adjustments are finalized before production begins.

The ideal way to run a new program is to run it through the control graphic display. It is fast and accurate. Test can be done with a variety of mode in effect, for example, dry run single block

Any unproved program is a potential source of problem. In manual CNC programming, mistakes are common than in CAD/CAM programs

A good way to look at a new program is through the machine operator's eyes. The most important thing is consistency in programming approach. For example.

- Al the tool approach clearances the same way as always
- Basic programming format maintained from one program to another program
- Scans the written program twice once on paper copy.
- Common mistakes such as a missing minus sign or an address, a misplaced decimal points
- Double check the program

Check the program through control graphic display that is tool path simulation. It is fast and accurate, and offers of confidence before actual machining

The tool path simulation shows the outline of finished part and the tool motions . The tool motions are identified by a dashed line(rapid motion) and a solid line(cutting motion).

Offset adjustment on first part and in production run

Objectives: At the end of this lesson you shall be able to
- **explain the step to adjust the first part dimension in x and z axes**
- **brief the steps in wear adjustment in x and z axes.**

When writing a program, the operator fix zero on the part drawing that is work zero/program zero and develop the CNC part program according to the part dimension either in absolute mode or incremental mode using G codes and verify it.

The tool nose radius centre, the tool direction of the turning tool, and the tool length are not taken into account. However, when machining a work piece, the tool path is affected by the tool geometry.

In CNC program one should remember that, the machine zero point, work zero point, reference point and tool tip point that is the commanding point.

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

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

The reference point is a fixed point inside the CNC machine whose values are already stored in the CNC machine. So when we take the CNC machine to the reference point those values automatically get active and the CNC machine comes to know its current position.

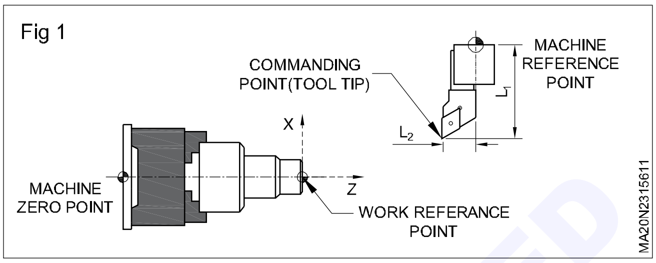
If you want or know more about the reference point then, there are limit switches for every axis in CNC machine to tell the reference point position. So when we take the CNC machine to the reference point those limit switches got active and tell the CNC machine control that the specified axis is reaching to its reference point position, so the CNC control update the current position for that specific axis because CNC control already knows the position at that point, and this is repeated with all the CNC machine axis

Tool Length Compensation There are two kind of ways to specify the value of tool length compensation.

- Absolute value of tool length compensation (the distance between tool tip and machine reference point)
- Incremental value of tool length compensation (the distance between tool tip and the standard tool)

As it is shown in Fig 1, L1 is the tool length on X axis. L2 is the tool length on Z axis. It should be noted that the tool wear values on X axis or Z axis are also contained in the tool length compensation.

Fig 1



Programmer will make arrangement to shift machine zero to work zero point and reference point to tool tip by appropriate G codes and tool compensation function, now tool tip will move considering the work piece zero and tool tip as a commanding point,

The measurement of work offset and tool offset is correct the part dimension will be equal to the programed values, that is no variation in part dimension and the size of the work piece is within the tolerance of the machine

If there is a variation in dimension of first part the variation should be corrected.

Many current-model turning centres have two sets of offsets: Geometry offsets are used to assign program zero during setup, and wear offsets are used to make sizing adjustments during the production run.

When users view these offsets on the display screen, they look very similar. Each will have the same number of offsets and four registers (X, Z, R and T).

Turret index and offset specification are done with a four-digit T word (T0101, for instance). The first two digits of the T word specify the turret station number and the geometry offset number. The second two digits specify the wear offset number.

When an offset is invoked, say T0101, the control will add the value in the geometry offset to the value in the wear offset and use the result as the total offset. If a value of -10.027 is in the X register of the geometry offset and a value of (+) 0.001 is in the X register of the wear offset, the total offset will be -10.026.

So, it really doesn't matter into which kind of offset you enter a given value. If you want to make a sizing adjustment, you could enter it into the geometry offset; if you want to enter a program-zero assignment value, you could enter it into a wear offset (assuming the wear offsets do not have a maximum entry value on your machine).

Though these things are possible, it is recommended that you to separate the use of offsets. Use geometry offsets solely for program-zero assignment during setup. Use wear offsets solely for the purpose of work piece sizing during the production run.

While these may seem to be standard practices there are times when offsets are used somewhat inappropriately. Here are two examples:

Initial sizing on the first work piece. The setup workers have just finished making the setup, and they're running the first work piece hoping that it will pass inspection. They might be using trial machining techniques to ensure that new tools just placed in the turret will machine the work piece to size. Tool number two, the finish-turning tool, has just completed its machining operation and they find that it has machined a ∅ 50.04 mm instead of ∅ 50.00What should they do? Before answering, let me ask two more questions. What caused the 0.04mm deviation? Did it have anything to do with tool wear?

Though this initial deviation has more to do with program-zero assignment (possibly an inaccurate touch off) than tool wear, most setup people will modify the wear offset (reducing it by 0.04 mm). But do remember, they can just as easily reduce the geometry offset by 0.04 mm and the machine will behave in exactly the same manner.

What is the advantage of making the initial adjustments in the geometry offsets? For very small lots there may not be any. But with larger lots, finishing tools will eventually wear out and be replaced. During the tool's life, it's likely that several sizing adjustments have been made to accommodate tool wear. When the cutting tool is replaced, the operator must also remember to reset its wear offset. To what value will it be reset? If the initial adjustment is done in the wear offset, the operator must remember its initial setting (-0.04 mm in the example above). But if the initial adjustment is done in the geometry offset, they can simply reset the wear offset to zero. (You may be questioning if the operators can precisely change or index an insert in such a manner that it is in exactly the same position as the previous insert. But even if they cannot, they must still know the initial wear-offset setting, regardless of whether trial machining will be done when the tool is replaced.)

So again, it is recommended that setup people make initial adjustments in geometry offsets so that the values of wear offsets will be zero when the production run begins.

Tool nose radius compensation offset entries. The R and T registers are related to tool nose radius compensation. R specifies the radius of the cutting tool and T is a code number that specifies the tool type (T2 specifies a boring bar, and T3 specifies a turning tool). Again, there are R and T registers in both the wear and geometry offset pages.

First of all, be sure your setup people are not entering duplicate values in both wear and geometry offsets. If, for example, they enter a value of 0.04 in both R registers, most controls will add them together and use the total (0.08 in our case). Worse, if they enter the T value in both registers-like T3 for a turning tool-most controls will interpret the T word as T6 (not a turning tool). Note that there are some parameter settings that deal with these issues, so some controls may behave differently than others in this regard.

While the R and T registers have nothing to do with program-zero assignment, It is recommend entering tool nose radius compensation values into geometry offsets (leaving the R and T registers of the wear offset at zero). There are applications when as a cutting tool dulls, its radius gets smaller such as a button tool that machines a ball shape on the work piece. Trying to deal with this problem with the X and Z registers will never render the desired results. Entering the amount of tool wear in the R register of the wear offset will correct the problem.

Examples

Correction of work piece diameter in CNC lathe

1 If the work diameter measure more than the required value

Reduce the tool's X offset the difference between the measured value and the required value

or

Enter the difference between the measure value and the required value gainst X wear with a minus sign

Example

Required value	35.20 mm
Measured value	35.37 mm
X tool offset	47.32 mm

Solution

Tool offset correction = 35.37 - 35.20

= 0.17

New tool offset = 47.32 - 0.17 = 47.49

2 If the work diameter measured less than the required value

Increase the tool's X offset the difference between the measured value and the required value.

or

Enter the difference between the measured value and the required value against X wear with a plus sign.

Example

Required value	40.37 mm
Measured value	40.15 mm
X tool offset	57.732 mm

Solution

Tool offset correction = 40.37 - 40.15 = 0.22

= 0.22

New tool offset = 57.732 + 0.22 = 57.952

3 If the work piece length is measured less than the required value

Reuce the tool's Z offset by the difference between the measured value and the programmed values

Example

Solution

Tool offset in Z = 40.00 mm

Programmed length = 50.00 mm

Measured length = 49.50 mm

Difference = 0.50 mm

New tool offset in Z 40.00 - 0.50 = 39.50

4 If the work piece length is measured more than the required value

Add the tool Z offset by the difference between the measured value and programmed value

Example

Solution

Tool offset in Z = 40.00 mm

Programmed length = 50.00 mm

Measured length = 50.50 mm

Difference = 0.50 mm

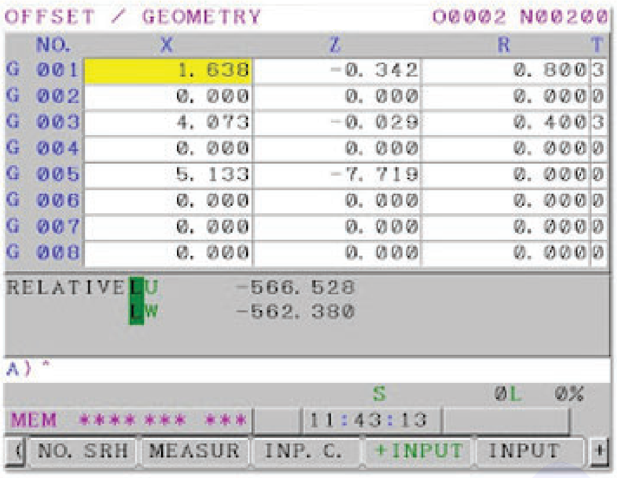
New tool offset in Z 40.00 - 0.50 = 40.50

Similarly, variation in part dimension in production run, calculate the variation as in tool offset and enter in wear offset page

Entering the tool offset date in geometry page

Press offset page /geometry and position the cursor at appropriate tool number and enter the tool data's. (Fig 2)

Fig 2



OFFSET / GEOMETRY O0002 N00200

NO.	X	Z	R	T
G 001	1. 638	-0. 342	0. 800	3
G 002	0. 000	0. 000	0. 000	0
G 003	4. 073	-0. 029	0. 400	3
G 004	0. 000	0. 000	0. 000	0
G 005	5. 133	-7. 719	0. 000	0
G 006	0. 000	0. 000	0. 000	0
G 007	0. 000	0. 000	0. 000	0
G 008	0. 000	0. 000	0. 000	0

RELATIVE U -566. 528
W -562. 380

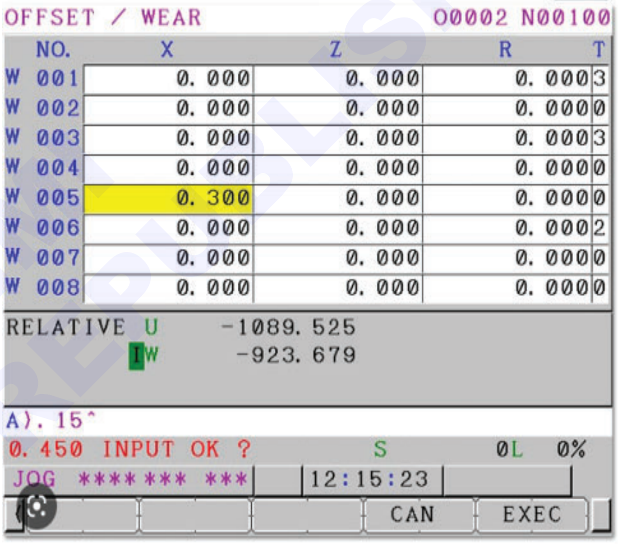
A) ^

S 0L 0%

MEM **** *** *** 11:43:13

Similar go to the wear offset page and enter the data's (Fig 3)

Fig 3



OFFSET / WEAR O0002 N00100

NO.	X	Z	R	T
W 001	0. 000	0. 000	0. 000	3
W 002	0. 000	0. 000	0. 000	0
W 003	0. 000	0. 000	0. 000	3
W 004	0. 000	0. 000	0. 000	0
W 005	0. 300	0. 000	0. 000	0
W 006	0. 000	0. 000	0. 000	2
W 007	0. 000	0. 000	0. 000	0
W 008	0. 000	0. 000	0. 000	0

RELATIVE U -1089. 525
W -923. 679

A). 15^

0. 450 INPUT OK ? S 0L 0%

JOG **** *** *** 12:15:23

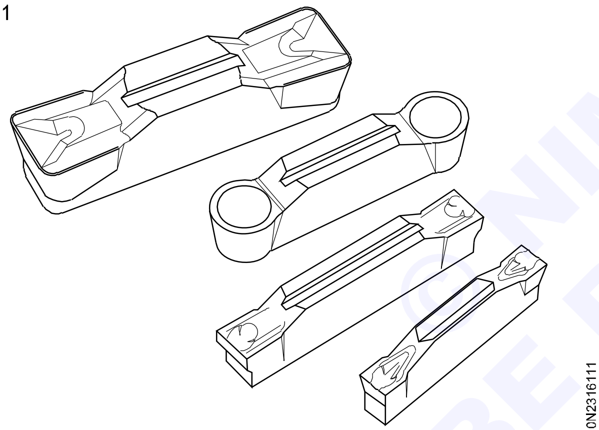
Tool selection related to grooving, drilling, boring and threading operations

Objectives: At the end of this lesson you shall be able to
- **list the points to considered for the selection of grooving tools**
- **explain "U" drill and grade of inserts**
- **select indexable boring tool**
- **state indexable threading insert and its grades.**

Grooving Inserts

Grooving inserts have either a square or round shape on the end. The shape typically corresponds to the shape of the groove being cut, although round inserts are the strongest inserts available and are preferred for roughing in demanding situations. When the depth of the groove is greater than the width, a multiple grooving operation is preferred. Multiple grooving consists of multiple plunge moves into the groove. The width of the tool is smaller than the width of the groove to allow for multiple side-by-side cutting passes. Figure 1 shows the different shapes of grooving inserts

Fig 1



GROOVING INSERTS

When the width of the groove is greater than the depth, a plunge turning operation is a better choice. Plunge turning combines plunging and turning motion, first plunging the tool to a shallow depth of cut, then moving laterally to the opposite end of the groove before plunging again to the next depth. For the tool holder, the tool overhang should be minimized to improve the stability of the tool during plunge moves. The width and depth of the groove on this part are about the same size, so the grooving operation can consist of multiple grooving. The width of the insert should be between 3 mm and 5mm to allow for multiple plunges across the 8mm wide groove.

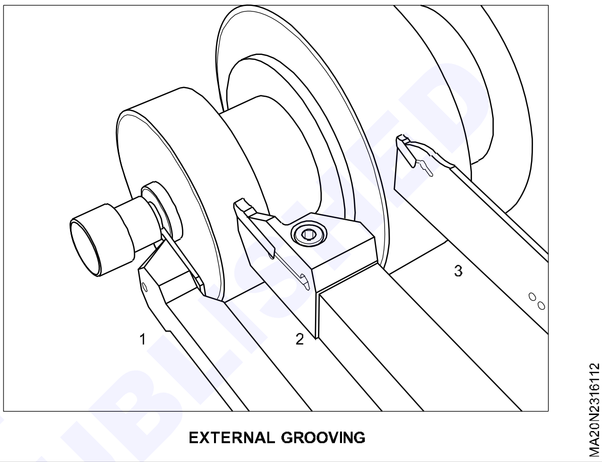
External grooving tools shown in Fig 2

Internal grooving tool with tool holders shown in Fig 3

Drilling Tools

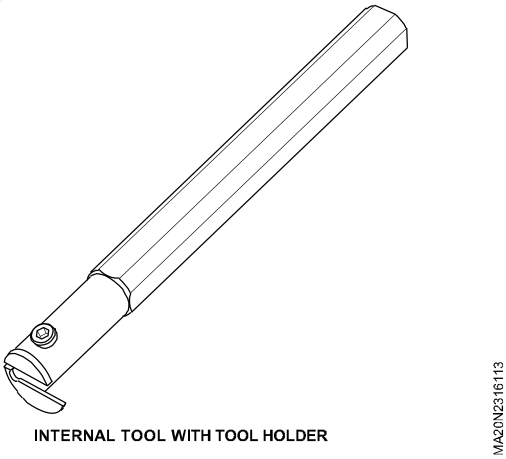
Indexable drills/Hole Making Tools Also Known as U drills. U drills are drilling tool with carbide inserts which can change easily and cost economical a normal U-drill look like this. (Fig 4)

Fig 2



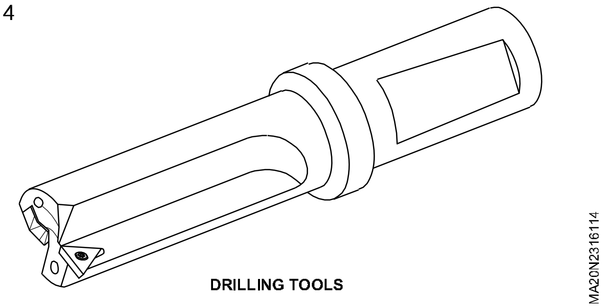
EXTERNAL GROOVING

Fig 3



INTERNAL TOOL WITH TOOL HOLDER

Fig 4



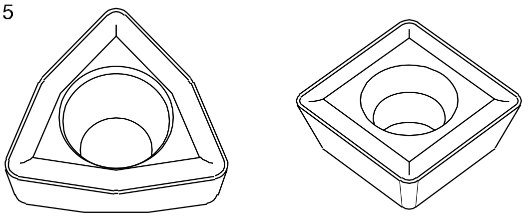
DRILLING TOOLS

U drill are those cutting tools which are used to make drill in any work piece weather its wood, aluminium, brass, steel, cast iron, titanium etc.

U drills are most commonly used because of economic cost and easy usage and can be used on both turning and milling machine.

They are SPMG and WCMX type U drills comes in Square and Trigon Shape inserts from Drilling range LXD 2 to LXD5 , 12 mm to 60 mm now 8 mm to 12 mm small drills are also becoming common if they are successful they can be very cost effective for these diameters drills. (Fig 5)

Fig 5



SQUARE AND TRIGON SHAPE INSERTS

No need of Centre Mark (Centre Hole)

As U-drills can plunge directly into work piece so you don't need centre drill mark it saves a turret space in lathe and extra time in both the machines.

Tolerance: Drill has Positive (+) and negative (-), tolerance the more LXD has more the tolerance if you choose a LXD5 U drill there are chances of .2 size of drill size the smaller the diameter the less is tolerance so its not a good choice to take a LXD5 drill when you require LXD3.

Through Coolant

U drills are through coolant as external coolant cannot reach in depth so it's always better to choose drill with coolant holes only as coolant can reach inside and save it from burning and help better chip evacuation

Boring Tools

Indexable boring bars are single-point cutting tools that use replaceable inserts to size, straighten, and finish the inside of drilled or cast holes. They mount to a lathe or turning machine and remain stationary while the machine rotates the work piece against the boring bar's cutting tip. The inserts are seated at the cutting tip and can be swapped out for new ones of the same style or a different style compatible with the boring bar without removing the boring bar from the machine. Each insert will have multiple cutting edges, so it can be rotated (indexed) to expose a fresh cutting edge when the old one dulls. Index able boring bars require fewer tool changes than solid tools in high-volume metalworking and fabrication applications with high speeds, high feeds, and difficult-to-machine materials.

Fig 6 shows the details of the boring tool

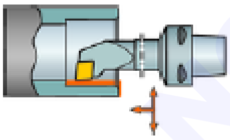
Fig 6

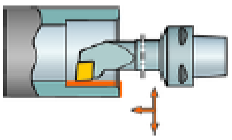
Coromant Capto® boring bars
CoroTurn® RC rigid clamp

DCLNR/L
Entering angle: K_r 95°
Lead angle: -5º

DDUNR/L
Entering angle: K_r 93°
Lead angle: -3º

Coolant Inlet: Axial through the center
Right hand style shown

Main application				Dimensions, mm, inch								Gauge inserts		
		iC	Ordering code	D_{5m}	$D_{m min}$	D_1	f_1	l_1	l_3	$\gamma^{1)}$	$\lambda_s^{2)}$	ISO	ANSI	Nm3)
	09	3/8	C4-DCLNR/L-13080-09	40	25	20	13	80	57	-6°	-14°	CNMG 09 03 08	CNMG 322	1.7
				1.575	.984	.787	.512	3.150	2.244					
			C5-DCLNR/L-13080-09	50	25	20	13	80	56	-6°	-14°	CNMG 09 03 08	CNMG 322	1.7
				1.968	.984	.787	.512	3.150	2.205					
	12	1/2	C4-DCLNR/L-17090-12	40	32	25	17	90	68	-6°	-12°	CNMG 12 04 08	CNMG 432	3.9
				1.575	1.260	.984	.669	3.543	2.677					
			C5-DCLNR/L-17090-12	50	32	25	17	90	66	-6°	-12°	CNMG 12 04 08	CNMG 432	3.9
				1.968	1.260	.984	.669	3.543	2.598					
			C6-DCLNR/L-17100-12	63	32	25	17	100	72	-6°	-12°	CNMG 12 04 08	CNMG 432	3.9
				2.480	1.260	.984	.669	3.937	2.835					
	16	5/8	C6-DCLNR/L-27140-16	63	50	40	27	140	114	-6°	-16°	CNMG 16 06 12	CNMG 543	6.4
				2.480	1.968	1.575	1.063	5.512	4.488					

Main application				Dimensions, mm, inch								Gauge inserts		
		iC	Ordering code	D_{5m}	$D_{m min}$	D_1	f_1	l_1	l_3	$\gamma^{1)}$	$\lambda_s^{2)}$	ISO	ANSI	Nm3)
	11	3/8	C4-DDUNR/L-17090-11	40	32	25	17	90	68	-6°	-12°	DNMG 11 04 08	DNMG 332	1.7
				1.575	1.260	.984	.669	3.543	2.677					
			C5-DDUNR/L-17090-11	50	32	25	17	90	66	-6°	-12°	DNMG 11 04 08	DNMG 332	1.7
				1.968	1.260	.984	.669	3.543	2.598					
	15	1/2	C4-DDUNR/L-27080-15	40	50	39.7	27	80	59	-6°	-11°	DNMG 15 06 08	DNMG 442	3.9
				1.575	1.968	1.563	1.063	3.150	2.323					
			C5-DDUNR/L-27140-15	50	50	40	27	140	118	-6°	-11°	DNMG 15 06 08	DNMG 442	3.9
				1.968	1.968	1.575	1.063	5.512	4.646					
			C6-DDUNR/L-27140-15	63	50	40	27	140	114	-6°	-11°	DNMG 15 06 08	DNMG 442	3.9
				2.480	1.968	1.575	1.063	5.512	4.488					

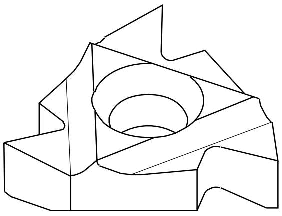
1) γ = Rake angle (valid with flat insert).
2) λ_s = Angle of inclination.
3) Insert tightening torque Nm.

R = Right hand, L = Left hand

Threading Inserts (Fig 7)

There are three types of threading inserts: Full Profile, Partial Profile, and Multi-Tooth.

Fig 7



THREADING INSERT

Full profile inserts cut the full shape of the thread groove, from bottom to top. These are useful for high productivity in threading.

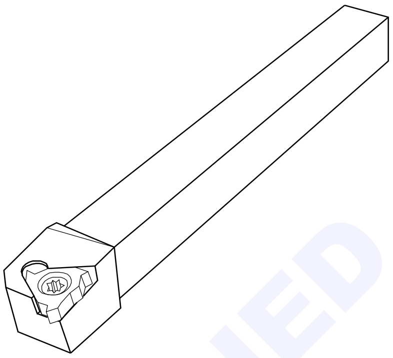
Partial profile inserts cut the bottom of the thread groove but leave clearance at the top. These are useful for cutting a range of thread sizes with a small inventory of inserts.

Multi-Tooth inserts feature multiple teeth in a series to reduce the number of passes required to cut a thread. These are useful for mass production in threading.

GC1125 is the first choice grade for ISO P, M, K and N materials in Coro Thread 266. This PVD grade combines the superior wear resistance of a coated grade with the edge sharpness and toughness of an uncoated grade. Optimized for steel threading and for medium to high speeds.

Indexable threading insert with holder shown in Fig 8

Fig 8



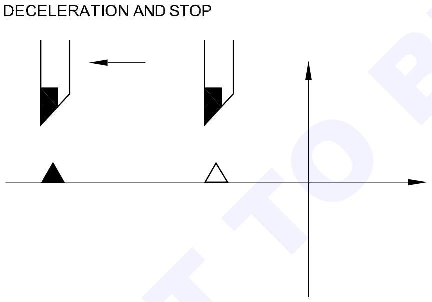
INDEX ABLE THREADING WITH HOLDER

Over travel recovering from over travel

Objectives: At the end of this lesson you shall be able to
- **state about over travel**
- **recover from over travel.**

Over travel: When the tool tries to move beyond the stroke end set by the machine tool limit switch, the tool decelerates and stops because of working the limit switch and an OVER TRAVEL is displayed (Fig 1)

Fig 1



OVER TRAVEL

Over travel during automatic operation: When the tool touches a limit switch along an axis during automatic operation, the tool is decelerated and stopped along all axes and an over travel alarm is displayed

Over travel during manual operation: In manual operation, the tool is decelerated and stopped only along the axis for which the tool has touched a limit switch. The tool still moves along the other axes.

Releasing over travel: Press the reset button to reset the alarm after moving the tool to the safety direction by manual operation. For details on operation, refer to the operator's manual of the machine tool builder

Alarm in Fanuc system

Table 1

No.	**Message**	**Description**
506	Overtravel: +n	The tool has exceeeded the hardware specified overtravel limit along the positive nth axis (n: 1 to 4)
507	Overtravel: -n	The tool has exceeded the hardware specified overtravel limit along the negative nth axis (n: 1 to 4)

Procedure to disable overtravel

This will vary control to control and machine to machine

1 Turn off the control.

2 Press the [P] key and the [CAN] key simultaneously in the keyboard.

3 Do not release those keys and turn on the control.

4 Hold the keys until the control turns on completely.

5 Release the keys.

6 Try to send the axis to home position after moving the slides away from the over travel limit switch, be very careful since this procedure disables the software limits and you just have the limit switches to stop the machine at the end of its travel.

7 If it goes to home with no problem, turn off and then on the control again to set the software limits active again.

Causes of tool collision failure in CNC machine tools

Objectives: At the end of this lesson you shall be able to
- **list the common causes of collision in CNC machines**
- **brief the causes of each collision.**

Compared with ordinary machine tools, CNC machine tools have high machining accuracy, good dimensional stability, low labour intensity, and convenient modern management. However, due to improper operation or programming errors, it is easy to cause the tool or tool holder to hit the work piece or the machine tool, the lighter will damage the tool and the processed parts, and the severe will damage the machine components, making the CNC machine tool processing. Loss of accuracy, even causing personal accidents. The following are the main reasons for tool collision failure of CNC machine tools:

1 Programming Errors

Programming errors can be difficult to detect, especially when inexperienced employees are involved. The most common errors are spelling and syntax related. Replacing an "O" with zero, or putting a comma in the incorrect place particularly in axes values and sign. Wrong value and sign in z axis will collide with the work and table, similarly x and y axes result in over travel of machine slide

Errors may be difficult to detect and so most modern CNC machines have a failsafe feature to minimise serious machine and tool damage. Employee training on the use of G-codes and M-codes is the best preventative measure as this ensures the program does not endanger equipment or work pieces. Teaching new employees to program slowly and to regularly review their programs will aid with minimising programming errors.

Minimising manual data entry will minimise the risk of human error. As long as an over-reliance on human intervention exists, CNC machining mistakes will continue to occur. Investing in CNC machine monitoring systems will contribute to efficiency improvements and eliminating human error.

2 The simulation processing machine is not locked

Since the CNC machine tool is locked by software, it is impossible to visually see whether the machine tool is locked in the simulation interface when the automatic operation button is pressed during simulation processing. There is often no tool setting during simulation. If the machine tool is not locked and running, tool collision is very likely to occur. Therefore, you should go to the running interface to confirm whether the machine tool is locked before simulating processing.

3 Forgetting to turn off the dry run switch during processing

In order to save time during program simulation, the dry run switch is often turned on for the CNC turning-milling compound machine tool. Dry run means that all motion axes of the machine tool run at the speed of G00. If the dry run switch is not turned off during processing, the machine tool ignores the given feed rate and runs at the speed of G00, causing accidents of hitting the tool and hitting the machine tool.

4 No reference point return after dry run simulation

When the program is verified, the machine tool is locked and the tool is in the simulation operation relative to the work piece processing (absolute coordinates and relative coordinates are changing). At this time, the coordinates do not match the actual position. The method of returning to the reference point must be used to ensure the machine The zero-point coordinate is consistent with the absolute and relative coordinates. If no problems are found after the program is checked, the machining operation will be carried out, which will cause tool collisions.

5 The direction of over travel release is wrong

When the machine tool is over travel, you should press the over travel release button and move it in the opposite direction manually or manually to eliminate it. However, if the direction of release is reversed, it will cause damage to the machine tool. Because when the over travel release is pressed, the over travel protection of the machine tool will not work, and the travel switch of the over travel protection is already at the end of the travel. At this time, it may cause the workbench to continue to move in the over travel direction, eventually pulling the lead screw and causing damage to the machine.

6 Inappropriate cursor position during specified runtime

When specifying the operation, it is often executed downward from the cursor position. For the lathe, it is necessary to call the tool offset value of the used tool. If the tool is not called, the tool of the running program segment may not be the desired tool, and it is very likely that the tool collision accident may occur due to different tools. Of course, the coordinate system such as G54 and the length compensation value of the tool must be called first on the machining centre and CNC milling machine. Because the length compensation value of each tool is different, it may cause tool collision if it is not called.

As a high-precision machine tool, the CNC machine tool is very necessary to avoid collisions. The operator is required to develop the habit of being careful, careful, and operating the machine in the correct way to reduce the occurrence of machine tool collision.

On every CNC shop floor you're likely to run into the same mistakes. The learning curve for operating a CNC machine skilfully is steep, and beginners can benefit from some good pointers at the outset. Knowing what to do-and, more importantly, what not to do-will save time and money.

7 Setting Offsets Improperly

Offsets are a big deal. If you set any of your machine's offsets improperly, you run the risk of crashing the machine and causing significant and expensive damage. When using an edge finder, remember to compensate for half the diameter of the edge-finder tip. A lot of positional errors stem from forgetting to account for the edge-finder, a mistake which becomes obvious when all positions are off by that specific amount.

On control display, make sure that there are no offset shifts in the work offset page. For example, during setup any value in the global work shift offset should be fixed at "zero." A global work shift would result in every cutter moving stock in the wrong position. These are small details, but they can make all the difference in your final product.

8 Selecting the Wrong Tool for the Job

When starting a new project, selecting the right tool can be a process of trial-and-error. It pays to be attentive to the result, especially in initial stages. If you're using the wrong tool, you may notice scratches, burn marks, or edge distortion. These can be signs of "chatter," or machine vibration that decreases the quality of your work, and can dramatically reduce your tool and machine life.

To mitigate these problems, choose the right cutting tool for each particular job. You can perform trial runs to dial in your tooling when working with a new material.

9 Setting Cutter Height Inconsistently

More care should be taken for setting of cutter/tool height that demands precision. Wong tool height (less than then the actual value) will collide the work piece resulting in collision and breakage of tool. Check that the tools don't have excessive runout and that enough cutter length is protruding from the holder so that the cut depth is less than the protruding length of the tool. If you neglect this crucial step it's possible that the holder will crash the work piece.

10 Not Knowing the G-Code

A familiarity with your G-code can help you avoid these damages.

Verify that the programmed work offsets correspond to the intended work offsets and that all retracts are high enough to avoid any clamps that may be in use for work-holding. Inadequate retracts can result in damage to cutters, product, and the machine itself.

11 Poor Maintenance

Maintenance may sound unimportant to the beginning CNC operator, but over time it directly affects the function of your machine. A CNC machine that is poorly maintained will accumulate debris and may block your filters, resulting in overheating and significant damage over time. Your product can also suffer from accuracy and precision problems.

Alarm codes and its meaning

Objectives: At the end of this lesson you shall be able to
- **state the classifications of CNC alarm**
- **define the alarm code**
- **state the alarm lists in fame system**

CNC alarms classified into system alarm and custom alarm.

1 System alarm are related to CNC control system

2 Custom alarm are related to machine tool manufacturer.

An alarm code may indicator a problem with the CNC machine (Mechanical or Electrical) or with the G- code program the machine has loaded.

Alarm code also called as error code or fault codes.

Alarm list - (CNC)

- Alarms on program and operation (PS alarm)
- Back ground edit alarms (BG alarm)
- Communication alarms (SR alarm)
- Parameter writing alarm (SW alarm)
- Servo alarm (SV alarm)
- Over travel alarm (OT alarm)
- Memory file alarm (IO alarm)
- Alarms requiring power to be turned OFF (PW alarm)
- Spindle alarm (SP alarm)
- Over heat alarm (OH alarm)
- Other alarms (DS alarm)
- Malfunction prevention function alarm

Alarm list (PMC) programmable machine control

- Message that may be displayed on the PMC
- PMC system alarm messages.

Alarm list (serial spindle)

Error codes (Serial spindle)

Alarm numbers are common to all these alarm types

Depending on the state, an alarm is displayed in the following examples

PS alarm number Example PS 0003

BG alarm number Example BG 0085

SR alarm number Example SR 0001

Number	Message	Description
003	Too many digit	Data entered with more digits than permitted in the NC function and the word. The number of permissible digits varies according to the function and the word.
0010	Improper G - code	An unusable G code is specified.

Alarm in CNC machine

Objectives: At the end of this lesson you shall be able to
- **state the alarm codes in CNC machine**
- **list the types of alarms**
- **brief the meaning of P/S alarm codes.**

Alarm codes (also called error codes or fault codes) are essentially the way CNC controls issue error messages.

An alarm code may indicate a problem with the CNC machine (mechanical or electrical), or with the g-code program the machine has loaded.

There are different types of error code they are

Alaram List (CNC)

1 Alarms on program and operation (PS alarm)

2 Background edit alarms (BG alarm)

3 Communication alarms (SR alarm)

4 Parameter writing alarm (SW alarm)

5 Servo alarms (SV alarm)

6 Overtravel alarm (OT alarm)

7 Memory file alarm (IO alarm)

8 Alarm requiring power to be turned off (PW alarm)

9 Spindle alarm (SP alarm)

10 Overheat alarm (OH alarm)

11 Other alarms (DS alarm)

12 Malfunction prevention function alarms (IR alarm)

Alarm list (PMC)

A.2.1 Messages That may be displayed on the PMC alarm screen

A.2.2 PMC system alarm messages

A.2.3 Operation errors

A.2.4 I/O Communication error messages

Alarm list (Serial spindle)

Error codes (Serial spindle)

Some of the common p/s alarm (Fanuc system) is listed in Table 1

Table 1

0	PLEASE TURN OFF POWER
6	ILLEGAL USE OF NEGATIVE SIGN
7	ILLEGAL USE OF DECIMAL POINT
10	IMPROPER G-CODE
11	NO FEEDRATE COMMANDED
22	NO CIRCLE RADIUS
28	ILLEGAL PLANE SELECT
46	ILLEGAL REFERENCE RETURN COMMAND
59	PROGRAM NUMBER NOT FOUND
60	SEQUENCE NUMBER NOT FOUND
70	NO PROGRAM SPACE IN MEMORY
73	PROGRAM NUMBER ALREADY IN
74	ILLEGAL PROGRAM NUMBER
77	SUB PROGRAM NESTING ERROR
85	COMMUNICATION ERROR
153	T-CODE NOT FOUND

General safety on CNC machine

Objectives: At the end of this lesson you shall be able to
- **state the points under general safety**
- **brief the safety points while working on machining centre**
- **list the safety points related to programming**
- **state the safety points while replacing the memory backup battery**
- **compare conventional NC and CNC machines**
- **list the difference between CNC & VMC.**

1 General safety

- Always be ALERT while maching on CNC machine.
- Know / Familiarize yourself with the machine before attempting to set (or) operate.
- First know all the emergency switches location before proceeding with operation.
- Do not press any switch / button / key unless you know fully well about the function of the same.
- Do not poke your head inside the machine when the auxiliary is on or in auto mode for any inspection /setting.
- Main switch should be off during cleaning the machine.
- Do not use compressed air for cleaning.
- Do not remove / adjust any sefety / limit / proximity switches.
- Do not use nonstandard tools/holder.

Safety check on the machine - before start

- Check the voltage and current.
- Check the lubricant tank oil level.
- Check the hydraulic tank oil level.
- Check the clamping devices.
- Check the cleanliness of the machines.
- Check the clamping stock.

Safety check on the machine

- Check the main pressure of the hydraulic/pneumatic system.
- Check the clamping pressure of the hydraulic system for chuck / fixture
- Check the chuck function.
- Lubricate the clamping device if any manual.
- Move the slides to Ref. points in return mode.
- Jog the slides to and fro and check the sliding movement.
- Check the centralized lubricant system manually by pressing appropriate Switches.
- Check the coolant supply by pressing appropriate switches.
- Check the selectable zero offset entries.
- Check the tool offset entries.
- Check the program and the appropriate. Zero and tool offset.
- Check the Feed overide position.
- Check the speed overide position.
- Check for the alarm if any and set it right.
- Check the tool fits.
- Check the tools positon with reference to program.
- Check the tool indexing.

Safety while machining

- Check the program control levels.
- (Single block, Dry run, skip optional stop etc.)
- Be careful Dry run should not be activated unless it is warranted.
- While in auto if - speed, feed is missing in put manually.
- If coolant is not activated through program, manually activated the coolant switches.
- Try to cut the job in single block when your are first time proving the program.
- Open the door at the end of machining preferably in Jog mode.
- Clean with brush, the tools and the job. Avoid compressed air for cleaning.

2 Safety - while working on machining centre

- First check the operations of the machine then try to machine a workpiece, preferably a trail with out job / tools is advisable to avoid unexpected accident.
- Check first thoroghly the work zero point , tool offsets etc. These two checks are very important.
- Check speed and feed in the program otherwise it may cause vibrations, loosening of the fixture etc. and end up in an accident.
- Avoid changing the factory set parameters; setting wrong and unknown parameter may damage the machine.

- Immediately after switching on the power do not touch any key on the MDI panel until the position display appears on the screen. Some of the keys are dedicated to maintenance (or) other special operations.
- Unexpected pressing of these keys may end up abnormal condition.
- Operations and programming manuals are general which describes several machines function. Therefore some functions may not be available to your machines. Hence check the specifications.
- Some functions may be implemented at the request of the consignees. These functions may be referred with the machine manual.

3 Safety - related to programming

- **Coordinate system setting**

  Incorrect coordinate system setting may damage the machine tool, workpiece etc, establish the coordinate system correctly in the program with reference to the machine setting.

- **Position by nonlinear interpolation**

  When performing positioning by non linear, Interpolation between the start and end points. The tool path should be carefully confirmed before performing the program.

- **Function involving 4th axis**

  When programming the 4th axis pay careful attention to the speed of the rotation axis . Incorrect programming of the rotational axis may cause the chuck to loose the grip and cause damage the machine tool, tool etc.

- **Inch/ metric conversion**

  Switching between inch and metric inputs does not convert the measurement units of data such as the workpiece origin, offset, current position etc. Hence before starting the machining determine which measurement units are being used and check the values accordingly.

- **Stroke check**

  Perform the manual reference position return as required. Stroke check is not possible before manual reference position return. When the stroke check is disabled alarm is not issued even if the stroke limit is exceeded.

- **Tool offset check**

  If the called tool is not considering the tool offset value -may cause heavy damage to the tool- hence check in the program for the tool offset, CRC values etc.

- **Plane selection**

  The machine behaves unexpectedly in wrong movement if the plane selection is wrongly selected in the program

- **Programmable mirror image**

  Note that programmed operation vary considerably when a programmable image is established.

- **Compensation function**

  Compensation is cancelled during reference position return command (or) a command based on the machine coordinate system. Before issuing such commands always cancel compensation function mode.

- Every time when the workpiece zero is set kindly avoid programming with G10 use the G 10 command when standard fixture is used for machining.

4 Safety - while replacing the memory backup battery

When the battery alarm is on the CRT, only maintenance people who know this work may be allowed.

Follow the given instructions

- First power on the CNC.
- Apply the emergency on.
- Do not touch any high Voltage part on the machine cabinet
- Replace only the recommended type batteries after checking the voltage & current.
- Secure the batteries correctly otherwise loss of data may end with a heavy loss when the alarm is displayed on the CRT within a week's time.

Note:

- **Change the battery within a week's time, which will prevent the contents of the memory,in programs, offsets, and parameters.**
- **Refer the maintenance manual for further details.**

	Comparison of conventional system with NC and CNC		
Sl. No.	**Conventional system**	**NC system**	**CNC system**
1	More manual work	Less manual work	Less manual work
2	Skilled labour is needed	Less skill is enough	Less skill is enough
3	Less accuracy	More accuracy	More accuracy
4	Less flexible	Medium flexible	More flexible

5	No part programming	Part programming is used	Re-programming is easy
Sl. No.	**Conventional system**	**NC system**	**CNC system**
6	Machining is done every time by the operator	Programming and punched in tape and storing is possible	Only one time the program is read storing is possible
7	Simulation cannot be done	Simulation is possible	Simulation is possible

8 Less production rate
Medium production rate
Medium production rate
(comparatively)
(comparatively)

Difference between CNC and VMC

- CNC having two axis x and z whereas VMC having three axis x, y and z.
- We have to change the tool for different operations in CNC while an automatic change of tool is done in VMC.

The tool changing process is done by a turret index in CNC whereas by ATC arm in the case of VMC.

- CNC most often used for cutting metal whereas VMC is very expensive and at the same time are very precise.
- CNC mainly used for turning inner and outer diameter whereas VMC mainly used for milling and end milling operations.
- Geometry offset in CNC is z and x axis and in VMC XYZ axes.
- The tool is rotated in VMC whereas not in the case of CNC.
- In CNC, a turret is worked as a tool holding and supporting device while Magazine is used for tool holding tools suppporting device in VMC.

Principle of position control

Objectives: At the end of this lesson you shall be able to
- **explain the principle of position control in a CNC machine**
- **explain the terms pulse and pulse frequency**
- **explain the concept of positioning accuracy and repeatablity.**

The slides of CNC machine are driven by servo motors.

The feed rate and position to be reached by the slide (tool) are indicated to the control by means of a programme line such as the one given below.

G91 G71 G01 X 60 F30

The meaning of the above instruction is that the tool should travel a distance of 60 mm along the x axis at a feed rate of 30 mm/min from its existing positon. In other words the tool should travel a distance of 60 mm in 2 minutes.

Principle of position control

The above instruction requires that the feed motor should rotate at a predetermined r.p.m for a period of 2 minutes: and to achieve this, the feed motor should receive the required quantity of electrical energy. This calls for regulation of the input current to the motor and basically what the computer in the CNC machine does for position control is nothing but regulating the current input to the feed motor.

The control computer after receiving a travel instruction perform certain calculations and arrive at the number of pulsed to be sent out in the given time in order to achieve the required travel. (It may be noted that the basic output from any computer is an intermittent supply of low strength electric current called pulse or signal) Usually one pulse sent out by the control causes a tool travel of 0.001 and Hence, in the above example to achieve a travel of 60mm in 2 minutes, the control after reading the above program line would send 60,000 pulses(60 ± 0.001) in 2 minutes.

Pulse frequency

This term refers to number of pulses sent per second in the above case the pulse frequency is 500/S (60,000/ 120). The pulse frequency , it may be noted, is proportionate to the feed rate.

The pulse output by the computer being of low strength and intermitted in nature, is not directly send to the drive motor, Instead the intermitted current is converted to an equivalent continuous form, and is also strengthened suitably.

The feed motor thus receives a regulated supply of continuous current for a specified time interval, and hence, ensures a tool travel as instructed by the programm.

Concept of positioning accuracy and repeatability

Two important specifications of CNC machines are Positional accuracy and repeatability.

Positional accuracy: The difference in the positional coordinates to which a machine actually moves and the theoretical coordinates. Eg, you have commanded the machine to move to position X=10.58mm. When the machine moves, it actually reaches 10.57mm. So the positional accuracy in this case is difference between actual and theoretical values, i.e. 10.58–10.57=0.01mm. Generally, positional accuracy is not calculated by a single measurement. Multiple readings are taken when a machine moves to multiple programmed point and all deviations recorded. The statistical average value all these deviations is the positional accuracy of the machine. It is usually 0.020mm for normal CNC machines and can go upto 0.003mm for Jig Boring machines.

Repeatability: It is the reproducibility of the values when the machine is asked to move to a targeted point multiple times and the difference in various values is the repeatability. Eg you ask machine to move to target point 10.58mm, 5 times. the actual values to which the machine moves are 10.57, 10.568, 10.571, 10.567 and 10.572mm. Repeatability in this case is difference between to extreme positions, i.e. 10.572–10.567=0.005. In reality, repeatability is calculated using multiple readings over multiple points.

- Repeatability is more important then positional accuracy.

Positional accuracy and repeatability is measured separately for each individual axis of the machine (X, Y, Z etc.). This measurement is done by an instrument called Laser Interferometer.

CNC, VMC machine elements and their functions

Objectives: At the end of this lesson you shall be able to
- **state what is CNC and VMC**
- **list the use of VMC**
- **explain the function of various external and internal parts of VMC.**

Introduction

The term CNC stands for "Computer numerical control" and the CNC is a metal removing manufacturing process that typically employ's computerized controls and machine tool to remove material from workpiece and produce a custom designed part.

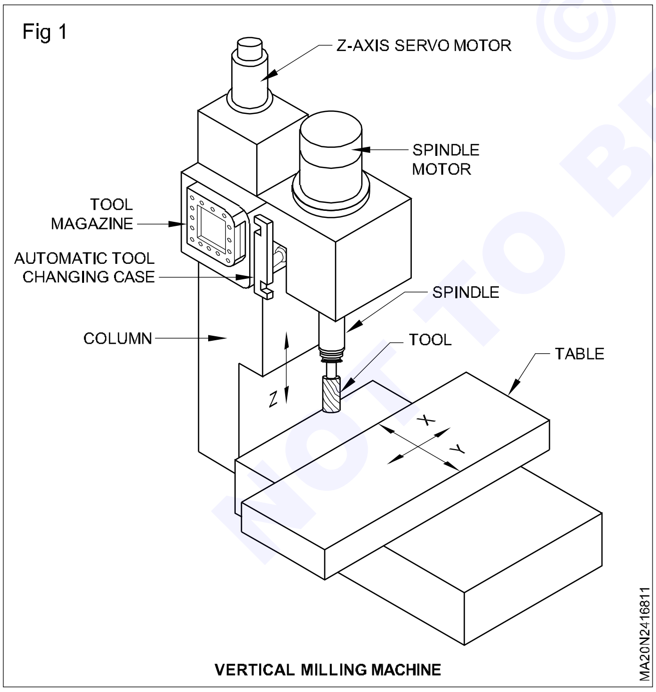
This process is suitable for a wide range of materials including metals, plastic, wood, glass, foam and etc.,The CNC machine are used in a variety of industries large scale to small scale, aerospace and telecommunication, which require higher tolerance and complicated shapes.

VMC

The term VMC stand for 'Vertical machining centre'. VMC machining refers to machining operations that utilize vertical machining centers, which as the name suggests, have vertically oriented machine tools. This machines are primarily utliized to machine components which require higher accuracy and complex shapes.

Construction of VMC (Fig 1)

Fig 1

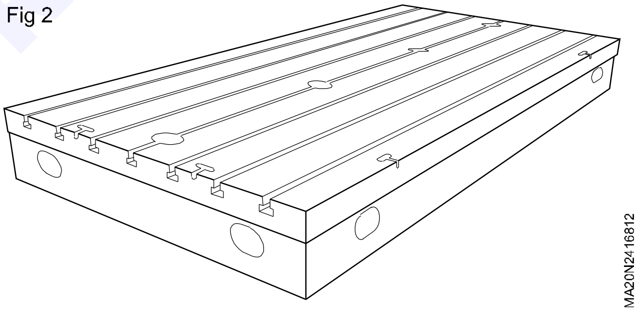


VERTICAL MILLING MACHINE

- Table/machine bed
- Automatic tool changer (ATC)
- Spindle
- Column
- CNC controller
- Ball screw
- Coolant system
- Hydraulic system
- Chip conveyor
- Rotary table
- Pallet changer
- CNC console
- Spindle motor
- Axes motor
- Encoders
- Drive system
- Centralise lubrication system

Table/Machiine bed (Fig 2)

Fig 2



The table is a platform on which the workpieces are mounted either directly or though a variety of fixture. The table has a motion of left and right which we call the 'X' axis and front to back, which is called the 'Y' axis.

Automatic tool change ATC (Fig 3)

The tool changer greatly increases a VMC productivity by allowing for the automatic, computer controlled selection of tools for a variety of tasks from rough cutting to fine boring.

CNC controller/Console (Fig 4)

The CNC controller is the brain of the machine. It contains the electronics that drive the axis motor to move axes.

Fig 3

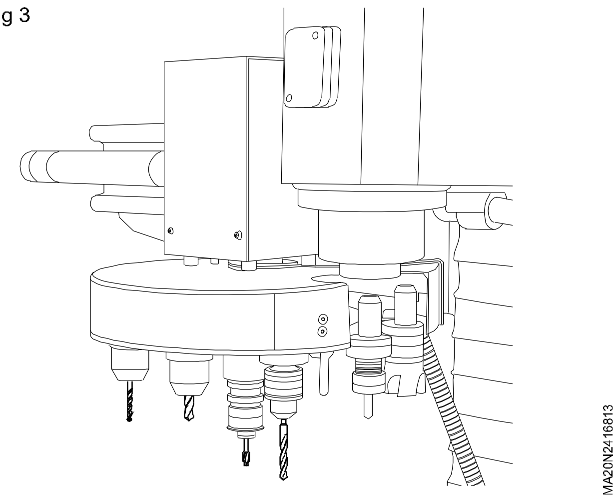
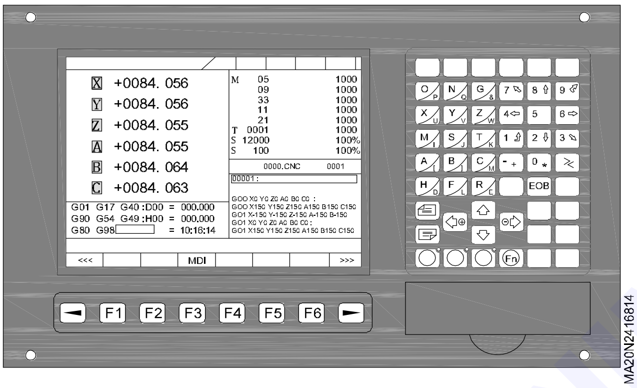


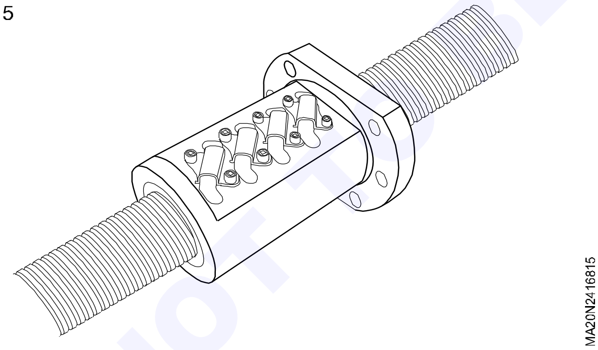
Fig 4



CNC controller are responsible for accepting G-code and manual inputs from CNC control panel and converting that into proper signals to control the axis stepper or servo motors.

Ball screw (Fig 5)

Fig 5



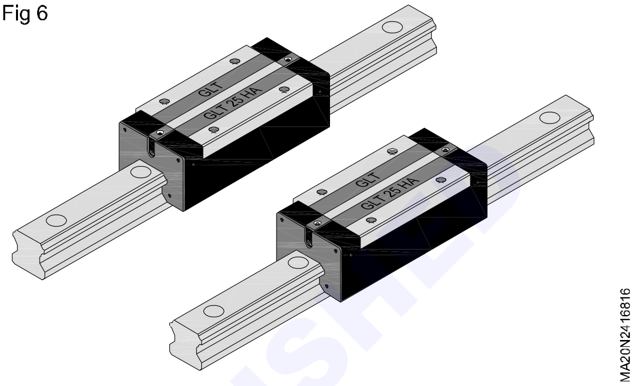
A ball screw (or recirculating ball screw) is a mechanical linear actuator that translates rotational motion to linear motion with little friction. A threaded shaft provides a helical race way for ball bearings which act as a precision screw. As well as being able to apply or withstand high thrust loads, they can do so with minimum internal friction. They are made to close tolerances and are therefore suitable for use in situations in which high precision is necessary. The ball assembly acts as the nut while the threaded shaft is the screw. In contrast conventional leadscrews, ball screws, ball screws tend to be rather bulky, due to the need to have a mechanism to recirculate the balls.

Applications

Ball screws are used in aircraft and missiles to move control surfaces, especially for electric fly by wire and in automobile power steering to translate rotary motion from an electric motor to axial motion of the steering rack. They are also used in machine tools, robots and precision assembly equipment. High precision ball screws are used in steppers for semi conductor manufacturing.

Linear motion guide ways (LM Guide ways) (Fig 6)

Fig 6



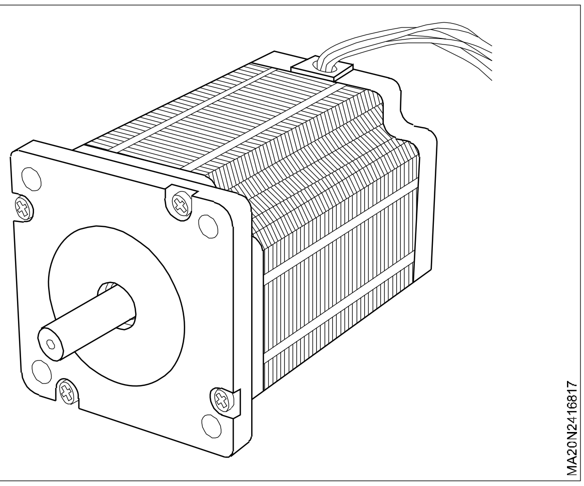
LM Guide ways provides low friction and high rigidity. Due to this low friction, the system able to maintain higher accuracy through out its life.

Spindle

Spindle is the main part of the machine tool. It receives power from the driven unit and delivers into the tool.

Spindle drive motors (Fig 7)

Fig 7



Normally, DC spindle drive motors are used in machine tool. But due to the availability of micro processor based technology. AC drives are prepared.

Axes motors (Fig 8)

Axes motors are used to drive the axes. Each axis will have separate axis motor to drive it.

Coolant (Fig 9)

The CNC machining centers have a coolant system to supply coolant to the cutting surface or the spindle tool during machining operation.

Fig 8

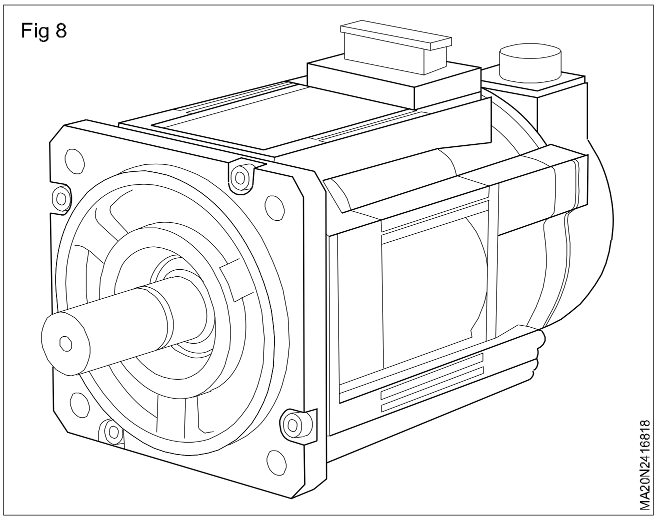
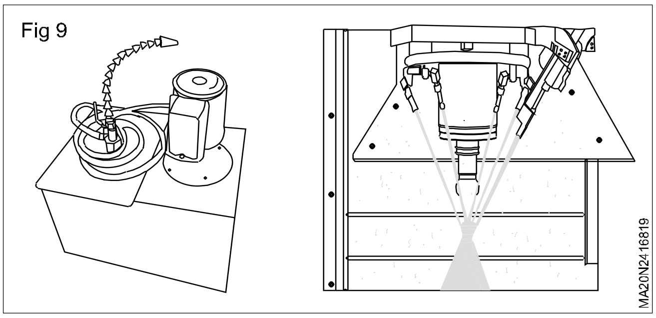
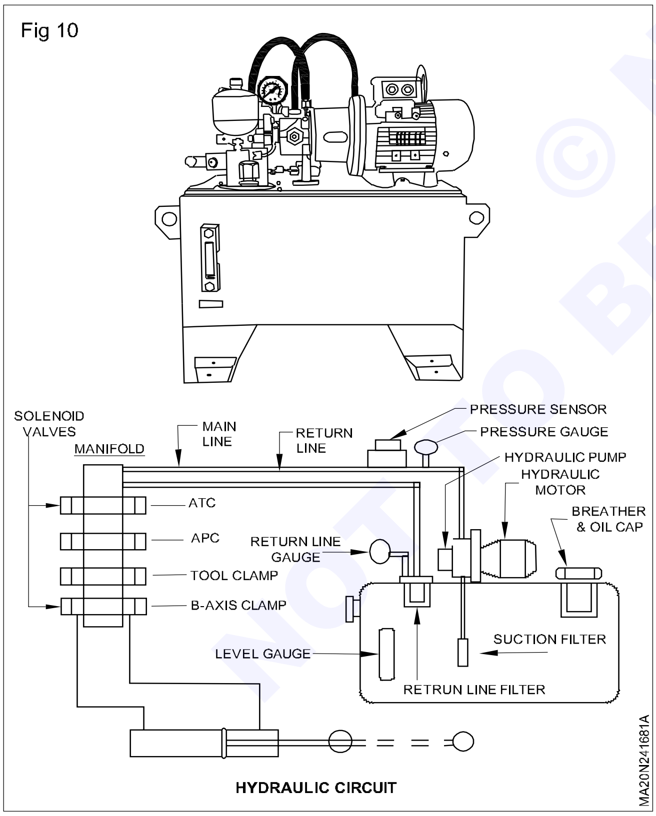


Fig 9



Hydraulic system (Fig 10)

Fig 10



Hydraulic system in CNC : Hydraulic system is nothing but a system operated by hydraulic oil. When there is a need of much pressure in low volumetric area. Hydraulic system give the solution. CNC machine is such type of machine where several jobs are executed in compact area and in less time. In such scenario, hydraulic system play a big role to execute several functions.

Hydraulic pressure in CNC machine is developed by hydraulic power pack which is a combination of several hydraulic circuit is a close circuit.

A hydraulic pump is incorporated to raise the pressure up to desired system pressure and that system pressure is distributed to various work place through solenoid valves and regulated through hydraulic pressure regulators.

Use of hydraulic system in CNC machine

Tool clam/de clamp operation: Spindle hold the cutting tools in both milling and turning center. Tools get fixed in the spindle through the draw bar or draw tube and spring catcher or ball catcher. Actually clamp operation is generally disc-spring operated and during clamp operation tool is pulled through draw bar or draw tube with the disc springs. So the hydraulic system get involved in the time of de clamp, when the total unit is pushed against the disc springs through a hydraulic cylinder. In some cases both the clamp and de-clamp operations are operated by hydraulic system.

Pallet clamp/de clamp operation: Job is held firmly through the fixture which is positioned on the work pallet. So during the cutting operation the pallet should be fixed firmly. This fixation of pallet is confirmed by hydraulic clamping. Hydraulically operated ball catcher hold firmly the cones of pallet and during pallet changing the ball catcher releases.

Hydro-motor for arm rotation or turret rotation: For most of the cases serve motor or induction motor is used in CNC machines for rotational operation. But for some cases where positional feed back and breaking is not so necessary, hydraulically operated hydro motor is used for rotation. Such as turret rotation in turning center where hydro motor is incorporated with Geneva wheel and gear train.

Rotational axis clamp/de-clamp: Rotational axis of CNC machine (A axis, B axis or C axis) are needed to be de clamped before rotation.

Those axes have to be clamped firmly in the definite degree in order to perform perfect cutting. These clamp and de-clamp operation is performed hydraulically. A hydraulic cylinder is incorporated for clamping and de clamping the axis.

So those are the use of hydraulics system in CNC machine. Apart from those, many other important operations are executed through hydraulic circuit.

Chip conveyor (Fig 11): Chip conveyor is used to evacute the chips from work zone. A variety of chip conveyors is employed to increase productivity and reduce down time from manually shoveling out waste chips.

Centralised lubrication system (Fig 12)

An automatic lubrication system (ALS) some time referred to as a centralized lubrication system (CLS), is a system that delivers controlled amounts of lubricant to multiple location on a machine while the machine is operating.

Fig 11

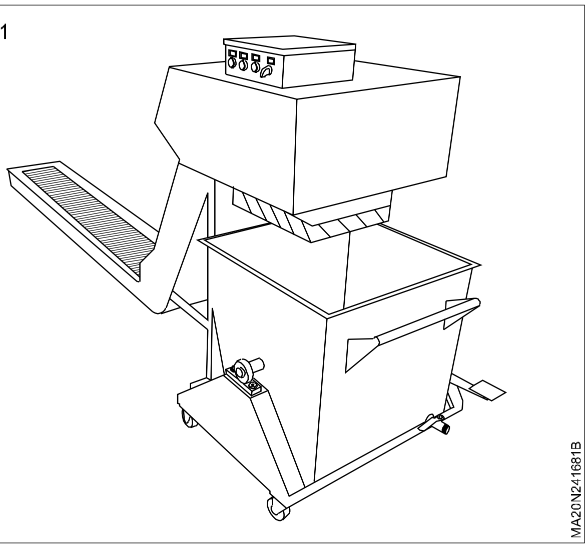
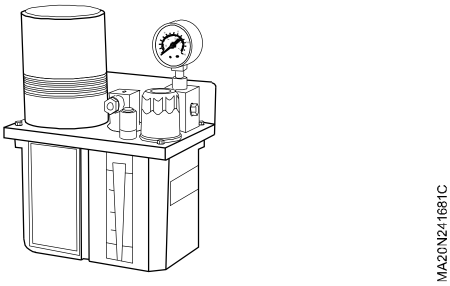


Fig 12

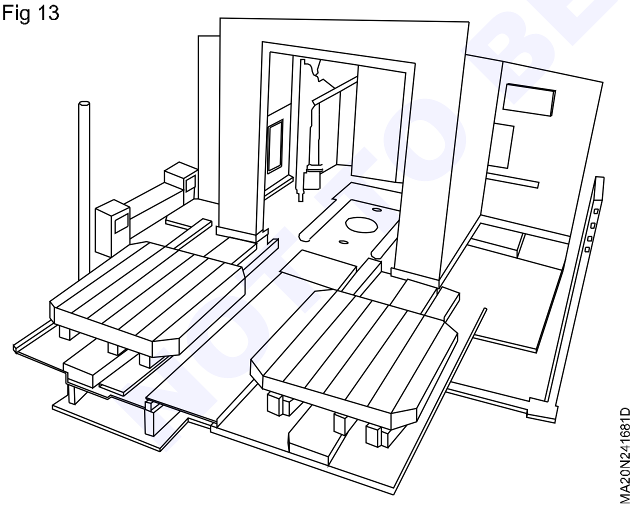


Pallet changer (Fig 13)

A pallet iin CNC is a movable part of a machine tool which helps to transport raw or finished parts from the machine in order to reduce downtime for part loading/un loading.

Pallet changers allow CNC machine operators to load and unload work piece in one area of the tool while the VMC is busy making chips and parts in another.

Fig 13

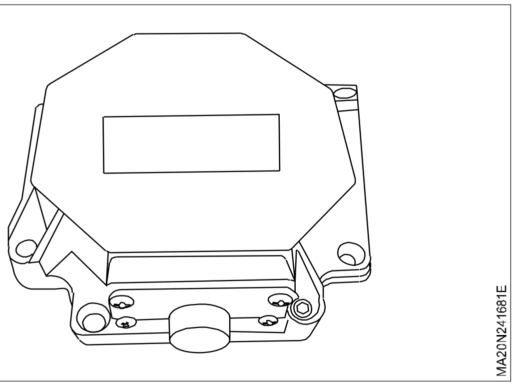


Encoders (Fig 14)

An encoder is a sensing device that provide feed back. Encoder convert motion to an electrical signal that can be read by some type of control device in motion control system.

The encoder sends feed back signal that can be used to determine posiion, count, speed or direction.

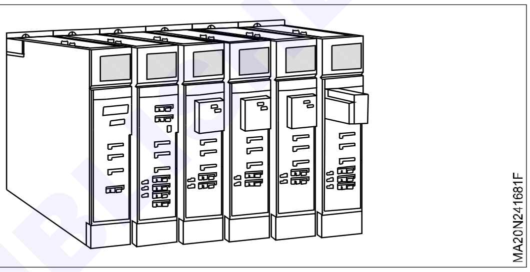
Fig 14



Drive system (Fig 15)

Drives are used to provide controlled motion to CNC elements. The MCU feeds the control signals to the amplifier circuits. The control signals are augmented to actuate drive motors which in turn rotate the ball lead screw to position the machine table.

Fig 15



Control switch

In electrical engineering, a manual switch with one or more positions for closing and opening electrical control circuits, they are mounted on control boards and panels.

Control switches are classified according to their functions.

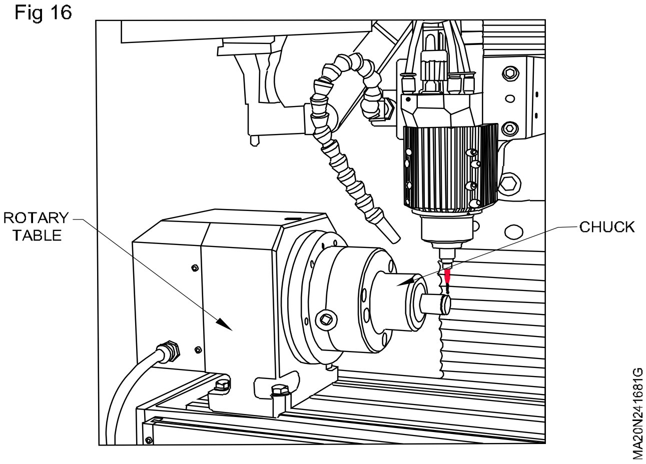
Switch that form command or signal control circuit.

Switch that acknowledge or correct incoming signals.

In analog computers a control switch is an electronic switching unit that controls the operation of the integrating amplifies.

Rotary table (Fig 16): It is an additional axes to the machine and it can greatly increase its productvity by turning a simple three axis machine into four or even six axis system capable of machining complex components with varying surfaces.

Fig 16



Feedback and control system

Objectives: At the end of this lesson you shall be able to
- **state the feed back in CNC**
- **brief the open loop control system in CNC**
- **explain the closed loop control system in CNC machine.**

Feedback system

A feedback system is one that compares its output to a desired input and takes corrective action to force the output to follow the input.

A feedback system is also referred to as a measuring system.

It uses position and speed transducers to continuously monitor the position of the cutting tool at any particular time.

The MCU uses the difference between reference signals and feedback signals for correcting position and speed errors.

Encoder is a device that used to convert linear or rotational position information in to electrical output signal.

Control system

A control system is defined as a system of devices that manages, commands, directs or regulates the behavior of other devices or systems to achieve a desired result. A control system achieves this through control loops, which are a process designed to maintian a process variable at a desired set point.

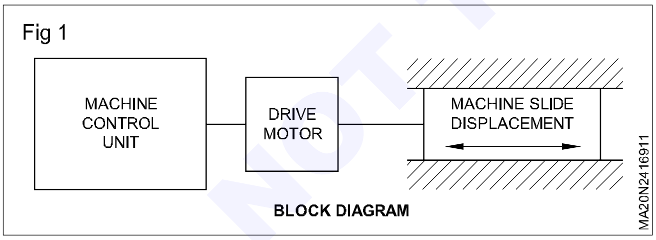
There are two main types of control systems. They are as follow

1 Open-loop control systems

2 Closed-loop control systems

Open loop control system (Fig 1 & 2)

Fig 1

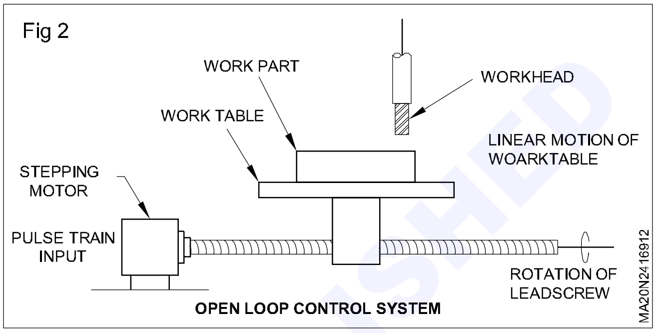


In an open loop control system (Fig 1 & 2) in which there is no arrangement for detecting or comparing the actual position of the cutting tool on the job with the commanded value.

Therefore, this system is not providing any check to see that the commanded position has actually been achieved. There is no feedback of information to the control also.

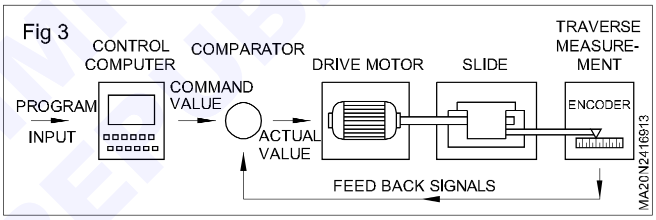
These system are not good where extremely accurate positioning is required.

Fig 2



Closed loop control (Fig 3)

Fig 3



Closed loop control (Fig 3) is a term which is used very often when we talk about CNC machines. This term signifies, that the control system has provisions to ensure that the tool reaches the desired position, at the correct feed rate, even if some errors creep in due to unforeseen reasons.

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

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

CNC interpolation

Objectives: At the end of this lesson you shall be able to
- **state the interpolation**
- **list the purpose of interpolator**
- **list the 5 type of interpolations**
- **brief the each type of interpolation.**

CNC interpolation

Interpolations are done to execute contouring tool paths. It produces a series of intermediate data points between given coordinate positions and computes the axial velocity of an individual axis along the contour path. Now a day's all CNC controls are equipped with linear and circular interpolations, some provides helical interpolation, and a very few provides parabolic and cubic interpolations.

Types of interpolation

1 Linear interpolation

2 Circular interpolation

3 Helical interpolation

4 Cubic interpolation

5 Parabolic interpolation

Linear interpolation

This moves tool from start point to the target point along a straight line. It can be implemented in a 2-D plane or 3-D space. the programming command should indicate X, Y, Z coordinates of target point, and feed rate.

Circular interpolation

It is programmed to cut circular arcs in three principal planes; namely XY, YZ, ZX. Direction, target position, arc radius, cutting plane, and feed rate must be specified in the program.

Circular interpolation commands

G02	CW Circular interpolation motion
X	End point X-axis motion command
Y	End point Y-axis motion command
Z	End point Z-axis motion command
A	End point A-axis motion command
I	Distance from start point to arc center in X-axis (optional)
J	Distance from start point to arc center in Y-axis (optional)
K	Distance from start point to arc center in Z-axis (optional)
R	Radius of the arc to be machined (If I, J, K are not used)
F	Feed rate in inches (or mm) per minute

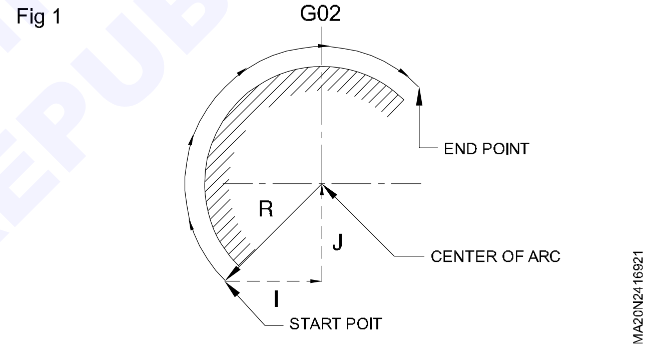
G03 CCW circular interpolation motion

G03 will generate a counter clockwise circular motion, but is otherwise defined the same way as G02.

These G codes are used to specify a clockwise or counter clockwise motion of two, of the linear axes. Circular motion is possible in two of the three axes in either the X,Y and/or Z axes as selected by G17, G18 and G19. The X, Y and Z in a circular command (G02 or G03) is used to define the end point of that motion in either absolute (G90) or incremental (G91) motion. If any of the axes, X, Y or Z for the selected plane is not specified, the endpoint location of the arc will then be recognized the same as the starting point of the arc, for that axis.There are two basic command formats for defining circular interpolation, depending on whether the IJK method or the R method is used to define the arc center.

Circular interpolation commands are used to move a tool along a circular arc to the command end position. Five pieces of information are required for executing a circular iinterpolation command. (Fig 1)

Fig 1



1 Plane selection

2 Arc start position co-ordinates

3 Rotation direction

4 Arc end position co-ordinates

5 Arc center co-ordinates or arc radius

Helical interpolation

Helical interpolation combines the two-axis circular interpolation with a linear interpolation in third axis.

Parabolic interpolation

It uses three non-collinear points to approximate curves that are of free forms. It reduces the number of programmed points by as much as 50 times the number required by the linear interpolation mode. It is mainly used in mold and die making.

Cubic interpolation

Cubic interpolation approximates the surfaces defined by third-order geometry. It involves the motion of three axes to machine complex shapes such as automobile sheet metal dies.

Machining operation and its tool path

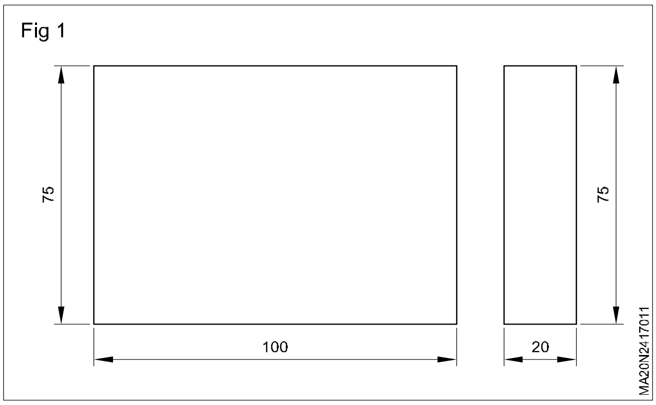
Objectives: At the end of this lesson you shall be able to

- **describe the tool path for face milling single and multi pass**
- **brief the tool path for side milling**
- **explain the tool path for pocket milling.**

Face milling (Single path)

Face milling tool path for Fig 1.

Fig 1



From the drawing, the face mill will take place along the X axis horizontal direction wil be selected. Before the deceding the tool path there are two decision to be made.

1 Face mill cutter diamer

2 Start and end point of the cut

Decided to mill in single path. Hence, the cutter diameter should be large than the face width of the work piece say 100 mm face milling selected.

Thumb rule for selecting the cutter diameter. The diameter of the cutter should be 1.3 to 1.6 times larger than the work piece.

Generally, lower left corner is considered as work zero. (X0 Y0)

To establish the start point the cutter centre and work centre in Y axis should align the Y = 37.5 mm.

In X axis the work piece zero and the cutter edge, we have to provide some clearance that is say 5 mm.

So the start point in X axis is cutter radius + the clearance

that is 50 + 5 = 55 mm

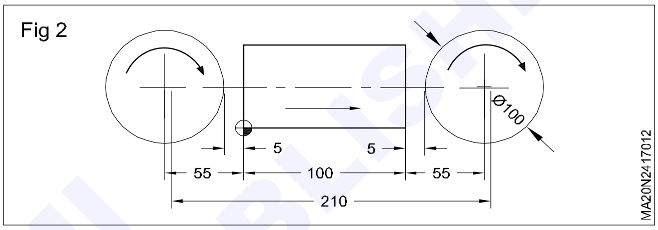
The start point is X - 55 and Y + 37.5 since it is in 2nd quardrant. (Fig 2)

The fig 2 shows the start point in X 55, Y - 37.5 and the end point in X axis is 155 and Y 37.5 in Z axis the required depth of cut may be given to obtain the required size.

Example program for single path

```
N1    G21;
N2    G17 G40 G80;
N3    G90 G54 G00 X - 55 Y - 37.5 S300 M03;
M4    G43 Z - 1.00 H01;
M5    G01 X155 Y - 37-5 F100;
M6    G00 Z100 M05;
M7    G28 G91 G00 Y0 Y0Z0;
M8    M30;
```

Fig 2



Multiple face mill cuts

General principles applying to a single cut do apply equally to multiple face cuts. Since a face mill diameter often too small to remove material in a single pass on a large material area. Several passes must be programmed at the same depth.

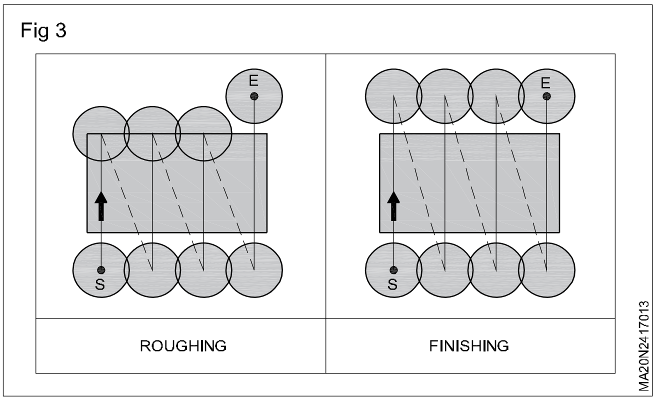
There are several cutting methods for a large area to be face milled and each may produce good machining conditions under certain circumstances.

The most typical methods are multiple/Unidirectional cutting and multiple bidirectional cutting (called zig-zag) at the same depth.

Unidirectional approach to a multiple face cut for rough and finish face milling shown in Fig 3.

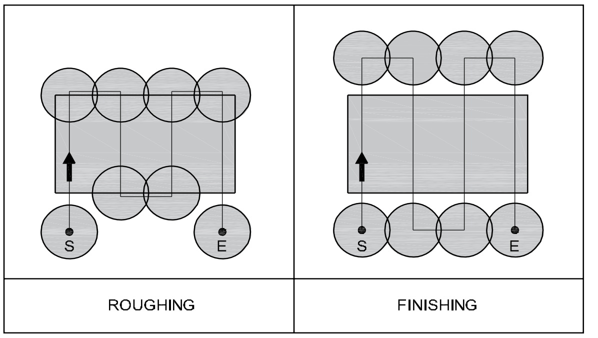
Bidirectional approach to a multiple face cut for rough and finish face milling shown in Fig 3.

Fig 3



Bidirectional approach to a multiple face cut for rough and finish face mill is show in fig 4.

Fig 4

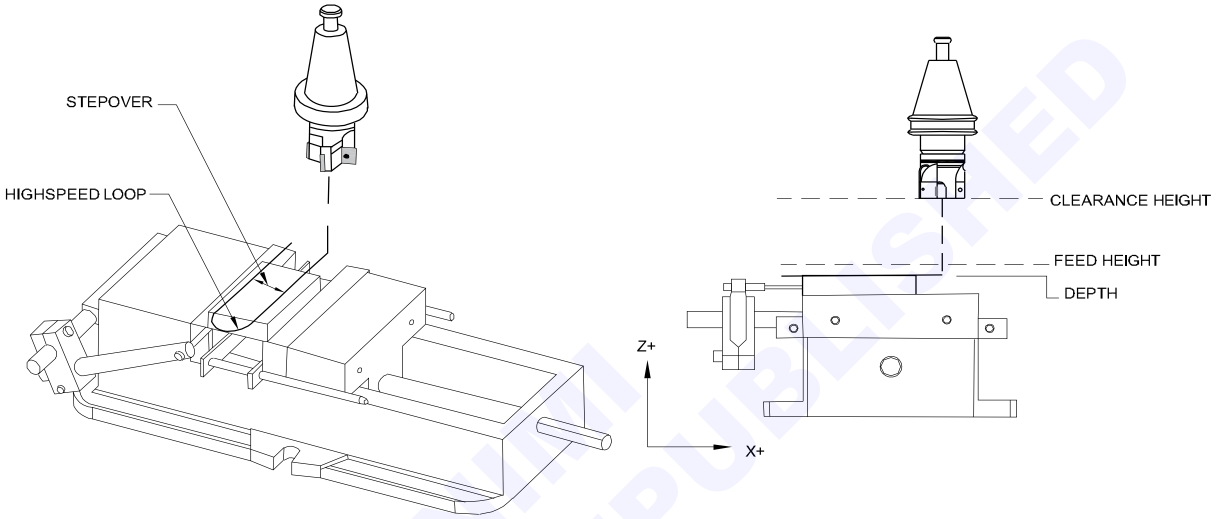


Note:

- **The step over is to make each cut approximately the same width. With only about 2/3 of the diameter cutting at any time.**
- **Do not use cutter compensation for face mill.**

Fig 5 shows the tool path for face milling

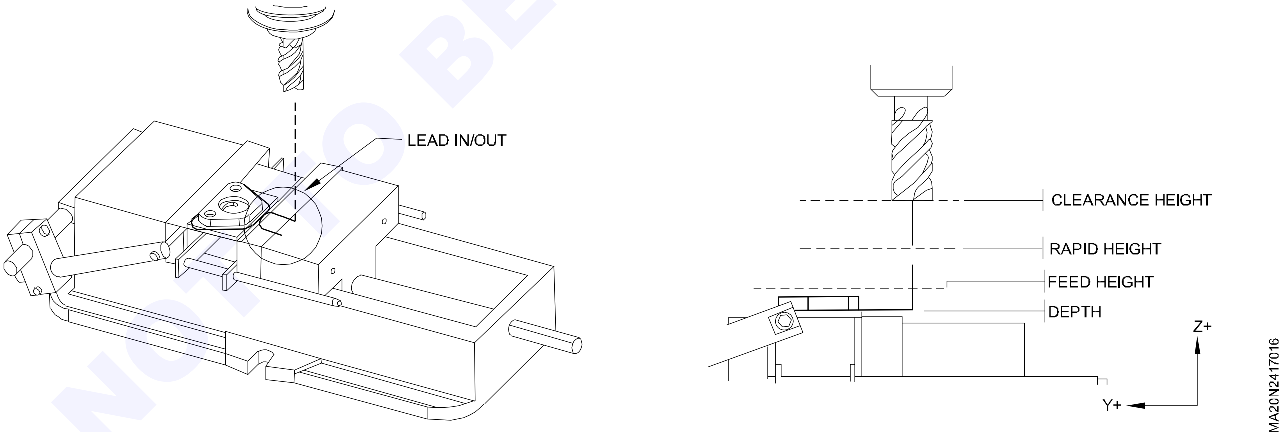
Fig 5



Side milling

Operations are used to rough and finish outside part wall as shown in Fig 6. Use cutter radius compensation (CRC) on high tolerance features so the tool path can be adjusted at the machine. If needed to account for tool wear and deflection.

Fig 6



Rules for contouring

- Only use CRC when needed.
- Start the tool path off the part to allow CRC to be fully in effect for the entire operation. The combined line - arc lead-in/out moves for most contours. The line is for-activating/deactivating compensation and the arcs blend the path into the part wall smoothly.
- Set a rapid height value to clear all clamps or other obstacles between cuts.
- Rough the wall and leave a constant thickness of material for the finsih operation. This ensures even cutting pressuure on the finish pass and thus a more accurate part.
- Extend the cut depth of full walls slightly below the bottom of the wall, but be careful not to cut into the machine table or vise hard jaws: This way, when the part if flipped over to face the other side, no flashing will be left on the bottom of the walls.

– Mill tools cut well in the XY direction, but not as well when plunging in Z. When possible, plunge the tool away from the part to avoid Z-moves into the stock material.

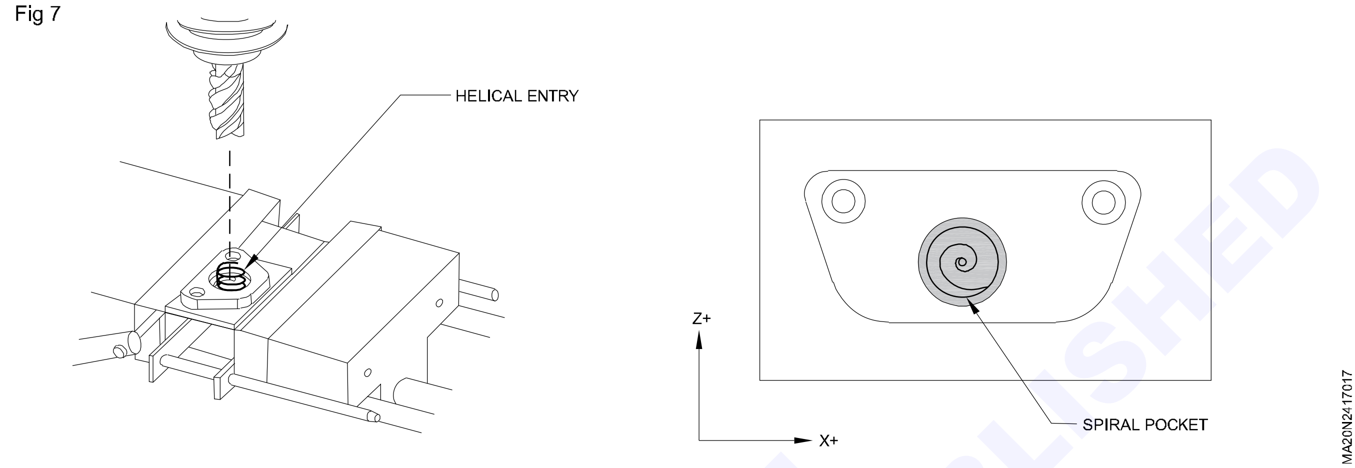
– When taking multiple depths of cut, make the last pass at full depth to remove any marks left by previous depth cuts.

– For tall walls,consider taking one additional finish pass. This so-called “spring pass” follows the same path twice to ensure the walls are perfectly straight and non slightly tapered due to cutting pressure which causes the tool to bend.

Pocketing

Pocket tool paths are used to remove excess material. An example of a spiral pocket with helical entry is shown in Fig 7. CRC is not active during the roughing cuts but may be used for finish passes on walls.

Fig 7



Rules for pocketing

- Rough passes should leave a constant thickness of material on the walls and the floor of the pocket to be removed by the finish passes.
- Consider using a roughing end mill to remove most of the material. These serrated mills can remove material at a far faster rate than finish end mills. They do leave a poor finish on the floors and walls that must be finished with a separate finish tool and operation.
- Helical moves are a good method for entering a pocket. If space does not allow a helical entry, use a center-cutting end mill or plunge the tool through an existing hole or a pilot hole created for this purpose. The pilot hole must be atleast 50% of the tool diameter.
- Spiral pocketing paths that start near the center of the pocket and move outward in a counter-clockwise direction are best because they cause the tool to continually climb cut.
- Use CRC only on finish passes.

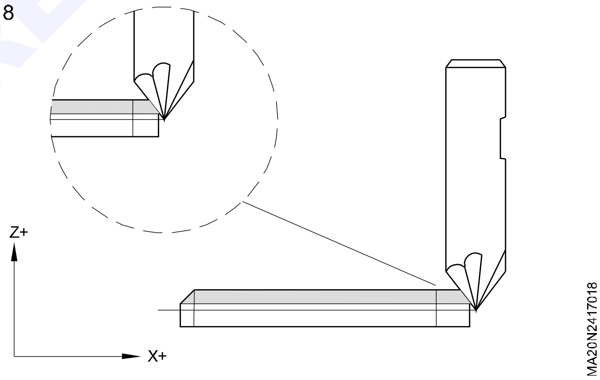
Chamfer milling

Chamfer is a type of 2D contour milling. Chamfer mills are of various tip angles are in high speed steel, carbide or as insert type tools.

Rules for chamfer milling

- Because the tip of a chamfer mill is not a sharp point the width of the chamfer may be wider than expected if you set the tool like an end mill. To prevent cutting too deep, consider raising the Tool Length Offset (TLO) about 0.3 mm after setting it. Then machine the chamfer, check its size and adjust the TLO down as needed to produce the correct width chamfer.
- Offset the chamfer mill as shown in the magnified view shown in Fig 8 to move the tool tip is away from the bottom of the chamfer. This ensures a clean bottom edge and because tool rotational velocity increases with tool diameter, is a more efficient to use the tool.

Fig 8



- Chamfer with a spot drill to precision de-burr sharp corners.

Tool path for drilling operations

Positioning the drill in XY co-ordinates at the hole location (Figs 9 & 10) 5 mm above the Z axis with rapid movement.

Fig 9

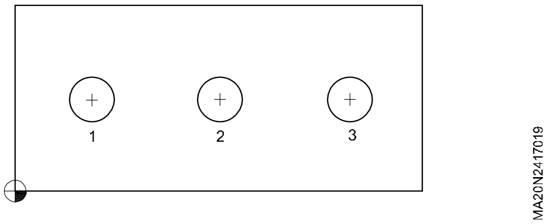
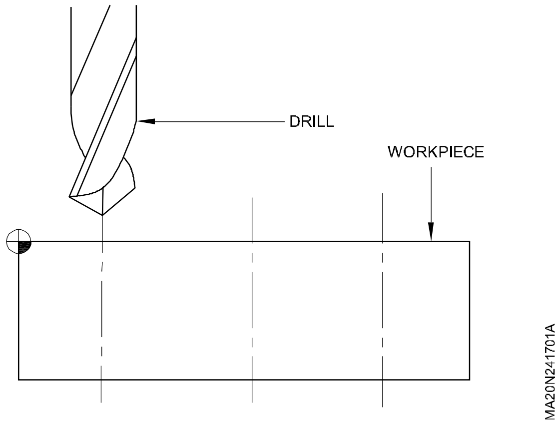


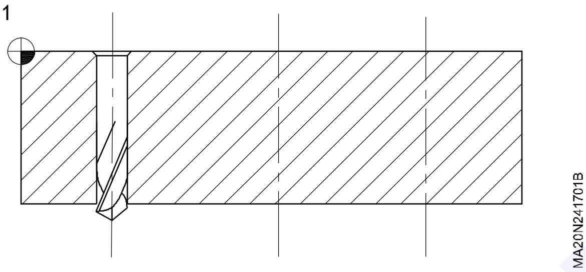
Fig 10



Spindle on to the required RPM.

Moving the tool in feed movement required depth (G01) (Fig 11) and with drawing the tool rapidly above the work. Similarly, repeat the steps for other drill holes.

Fig 11

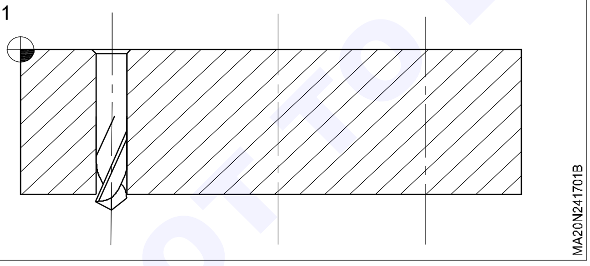


Note: For countersink, counter bore, spot face. After, reaching the required depth. Spindle should rotate without Z axis movement that it provide dwell time for few second.

For reaming, after reaching the required depth. Stop the spindle and move the tool in Z axis above the zero level.

Tapping tool path (Fig 12)

Fig 11



Tapping tool path is similar to the drilling operations only depth in Z direction is controlled by the pitch of the tap that is Z feed rate is equal to pitch and the spindle speed is 1/3rd of the original cutting speed.

Stop the spindle rotation and the Z movement after reaching the required depth.

Reverse the spindle direction and with draw the tap with same amount of feed rate.

Boring tool path

Boring Cycle is used to bore the hole(s).

The tool travels to the bottom of the hole with feed and then retracts back out of the hole at rapid feedrate.

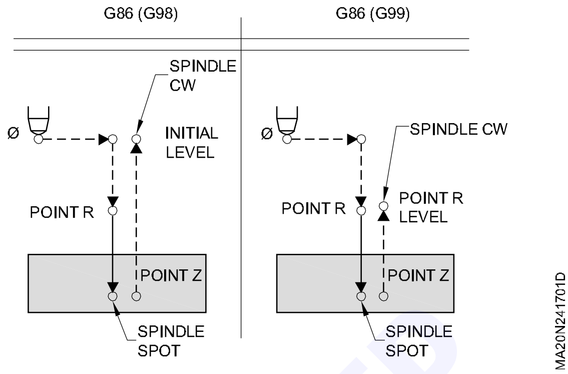
G86 Boring Cycle Format (Fanuc)

G86 X Y Z R F K

Parameters

X Y - Hole position data

Fig 13



Z - Boring depth (Absolute)

R - Tool starting position above the hole

F - Cutting feed rate

K - Number of repets (if required)

G86 Boring Cycle Operation

1 After positioning along the X- and Y-axes, rapid traverse is performed to point R.

2 Boring is performed from point R to point Z.

3 When the spindle is stopped at the bottom of the hole, the tool is retracted in rapid traverse.

Tool Return Position

Return plane is dependent on G98, G99 G-codes.

If G98 is specified with G86 boring cycle the tool returns to Initial-level.

If G99 is specified then tool will return to R level.

G86 Boring Cycle Program Example

M3 S2000

G90 G99 G86 X300. Y250. Z-150. R-100. F120.

Y550

G98 Y750

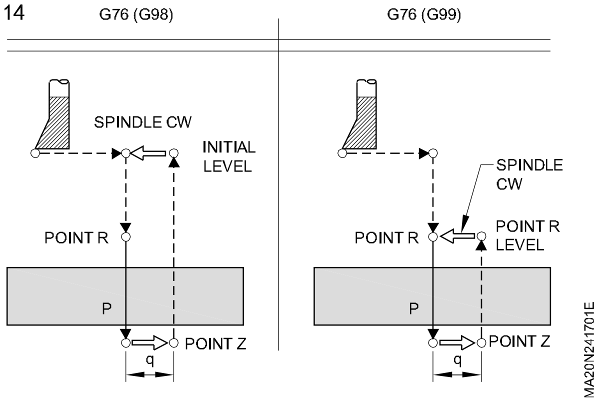
G80 G28 G91 X0 Y0 Z0

M5

Fine boring tool path

G76X-Y-Z-R-Q-P-F-K-;

Fig 14



Concept of co-ordinate geometry and polar co-ordinates

Objectives: At the end of this lesson you shall be able to
- **state about co-ordinate geometry**
- **describe the cartesian co-ordinates and its plane/quadrants**
- **explain the polar co-ordinate systems and its points.**

Co-ordinate geometry

Co-ordinate geometry is a branch of geometry. With the help of which the position of the points on the plane is defined with the help of an ordered pair of numbers also known as co-ordinates.

The use of co-ordinate geometry is found in engineering and physics.

Often the cartesian co-ordinate system is proved to help manipulate equations for planes, straight lines and squares in 2D and 3D.

The concept of co-ordinates

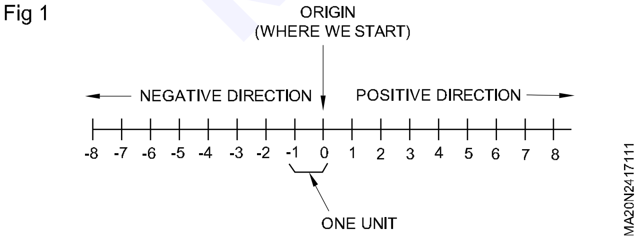
- The point where the X and Y axis intersect is known as the origin. At this point, both X as well as Y are 0.
- The values on the right-hand side of the X axis are positive while the values on the left-hand side of the X axis are negative.
- Similarly, on the Y axis, the values located above the origin are positive while the values located below the origin are negative.
- While locating a point on the plane, it is to be determined by a set of two numbers. First, we have to write about its location on the X axis followed by its location on the Y axis.

Cartesian fundamentals

Nearly all CNC machine uses a cartesian co-ordinate system based on an X, Y and Z axis. This system allows a machine to move in a specific direction along a specific plane.

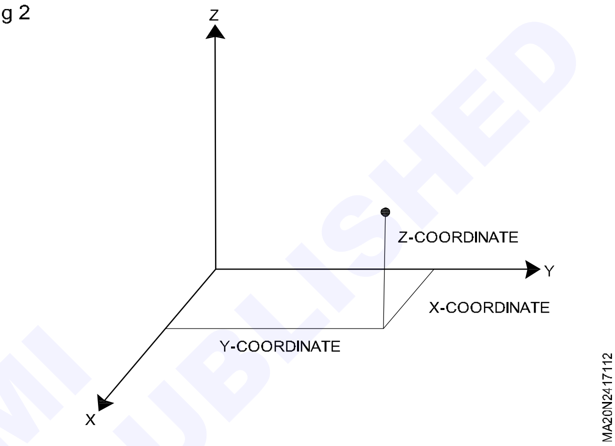
Reduce the cartesian system to its basics and you have a familiar numner line. One point on the line gets designated as the origin. Any numbers to the left of the origin are negative, while numbers to the right are poitive. (Fig1)

Fig 1



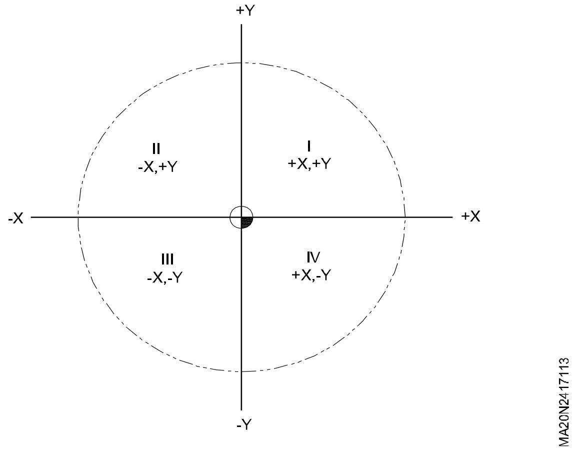
Combine the X, Y and Z axes at 90° angles and you create a three dimensional space for your CNC machine to move around in. Each axis meets at the origin. (Fig 2)

Fig 2



When two axes connect, they form a plane. For example, when the X and Y axes meet, you get an XY plane, where most of the work happens on 2.5D parts. These planes are divided into four quadrants, numbered 1-4, with their own positive and negative values. (Fig 3)

Fig 3



The polar co-ordinates system

The polar co-ordinate system is a two-dimensional co-ordinate system in which each point on a plane is determined by a distance from a reference point and an angle from a reference direction. The reference point (analogous to the origin of a cartesian co-ordinate system) is called the pole and the ray from the pole in the reference direction is the polar axis.

The distance from the pole is called the radial co-ordinate, radial distance or simply radius and the angle is called the angular co-ordinate, polar angle.

Angle is polar notation are generally expressed in either degrees or radians (2π rad being equal to 360°)

The polar co-ordiante system is extended to three dimensions in two ways: the cylindrical and spherical co-ordinate systems.

A polar grid with several angles, increasing in counter clockwise orientation and labelled in degrees. (Fig 4)

Polar co-ordinates are most appropriate in any context where the phenomenon being considered is inherently tied to direction and length from a center point in a plane, such as spirals. Planar physical systems with bodies moving around a central point or phenomena originating from a central point are often simpler and more intuitive to model using polar co-ordinates. (Fig 5)

Points in the polar co-ordinate system with pole O and polar axis L.

The point with radial coordinate 3 and angular coordinate 60 degrees or (3, 60°). The point (4, 210°).

Fig 4

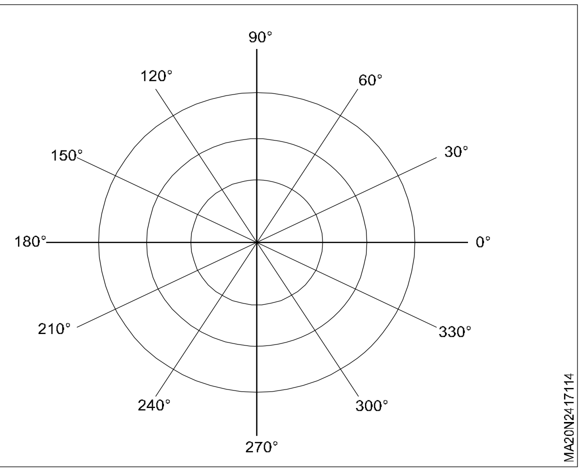
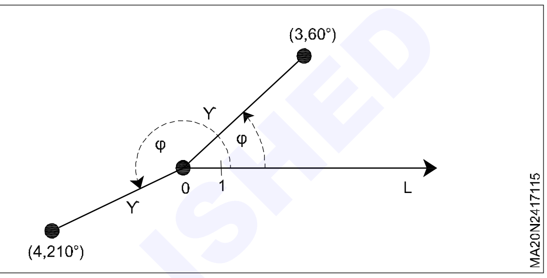


Fig 5

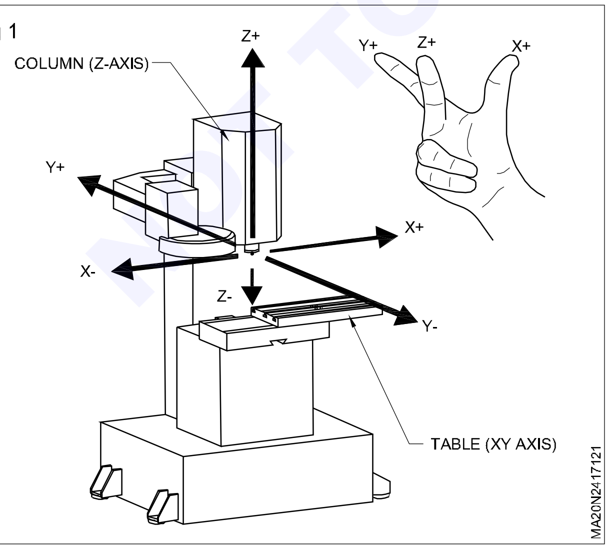


Concept of machine axes, machine zero and work zero

Objectives: At the end of this lesson you shall be able to
- **brief the method of designation of CNC machine axes**
- **explain the axes movement direction symbol**
- **state about machine origin, machine zero and where it is located**
- **describe work zero and its purpose.**

An easy way to understand the Cartesian coordinate system in relation to your CNC machine is using the Right Hand Rule. Hold your hand out palm up with your thumb and index finger pointed outwards, and your middle finger pointed upwards. Place your hand in front of your CNC machine, aligned with the machine's spindle, and you'll see the axes line up perfectly. (Fig 1)

Fig 1



- Middle finger is the Z-axis.
- Index finger is the Y-axis.
- Thumb is the X-axis.

CNC Machine Uses Coordinates: Using the Cartesian coordinate system, we control CNC machines along each axis to transform a block of material into a finished part. Although it's difficult to describe axes using relative terms, based on each axis, you typically get the following movements from the perspective of an operator facing the machine. (Fig 2)

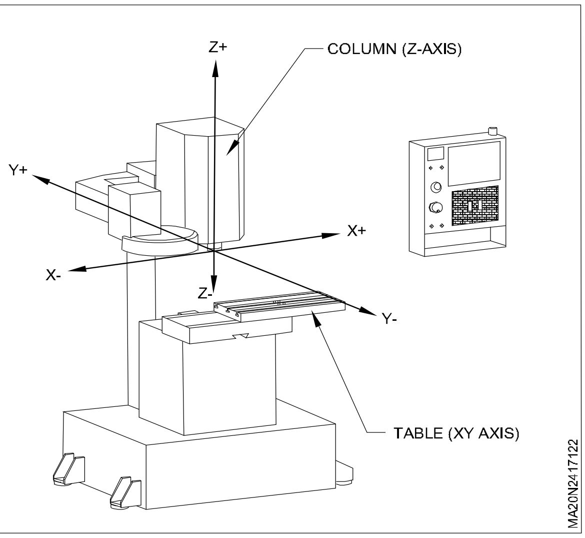
- X -axis allows movement "left" and "right"
- Y -axis allows movement "forward" and "backward"
- Z -axis allows movement "up" and "down"

Put all of this together, and you have a CNC machine that can cut various sides of a workpiece in the XY plane and at various depths along the Z axis. (Fig 2)

The movement of your CNC along the coordinate system is always based on how your tool moves, not the table. For example, increasing the X coordinate value moves the table left, but looking from the perspective of the tool, it's moving right along a workpiece.

Increasing the Z axis coordinate would move the spindle up, decreasing it would move it down into a workpiece. You are cutting into the piece which corresponds to a negative Z axis coordinate.

Fig 2



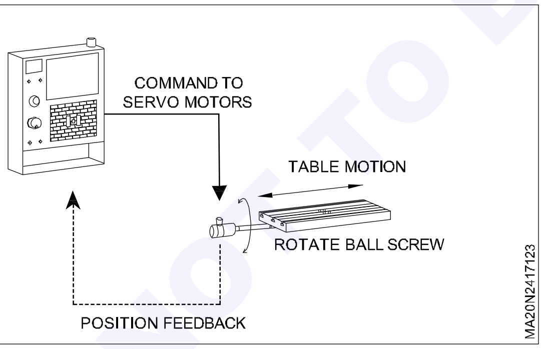
CNC Machine's Origin

Every CNC machine has its own internal origin point called Machine Home. When your CNC first boots up, it has no idea where it is in physical space and requires a calibration to get its bearings.

When this process occurs, all three axes of your CNC move towards their maximum mechanical limit. Once a limit is reached, a signal is sent to a controller which records the home position for that particular axis. When this occurs for all three axes, the machine is now "homed."

The process can vary from machine to machine. For some machines there is a physical limit switch that signals the controller that a machine has reached an axis limit. On some machines there's an entire servo system in place that makes this entire process incredibly precise. A machine controller sends a signal through a circuit board to a servo motor, which connects to each machine axis. The servo motor rotates a ball screw that's attached to the table on your CNC machine, making it move. (Fig 3)

Fig 3



Work Coordinate system (WCS)

To writing, CNC programs easier, we use a different coordinate system designed for human manipulation called a Work Coordinate System or WCS. The WCS defines a particular origin point on a block of material, usually.

we can define any point on a block of material as the origin point for a WCS. Once an origin point is established, you will need to locate it inside your CNC machine using an edge finder, dial indicator, probe, or other locating method.

Choosing an origin point for your WCS requires some careful planning. Keep these points in mind when going through the process:

- The origin will need to be found by mechanical means with an edge finder or probe
- Repeatable origins help save time when swapping out parts
- The origin needs to account for the required tolerances of downstream operations

There could be another full blog on selecting the most optimal origin point, especially for each subsequent setup as the tolerance stack-up begins to grow. Make sure you keep in mind the tolerances of previously machined features, your locating mechanism, and your machine to ensure your final part is to spec.

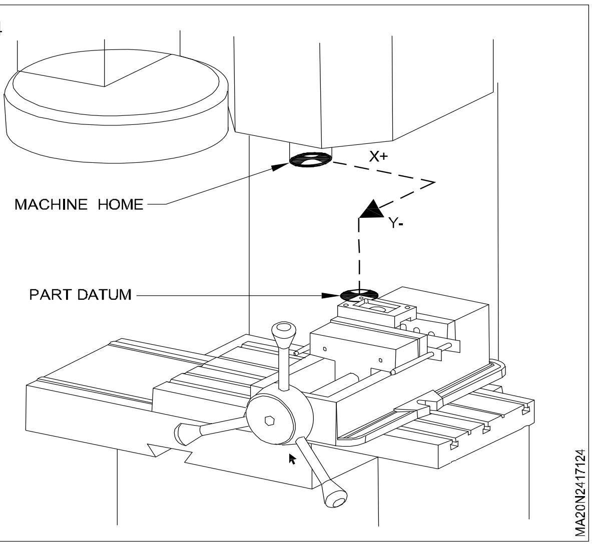
As we mentioned above, human operators will use a WCS, which provides an easy set of coordinates for writing a CNC program. However, these coordinates are always different from a machine's coordinates, so how does your CNC machine line the two up? With offsets.

A CNC machine will use a work offset to determine the difference in distance between your WCS and its own home position. These offsets are stored in the machine's controller and can typically be accessed in an offset table.

CNC and Human Coordinates Interact (WCS)

A CNC machine will use a work offset to determine the difference in distance between your WCS and its own home position. These offsets are stored in the machine's controller and can typically be accessed in an offset table. (Fig 4) several offsets are programmed; G54, G55, and G59. The benefit of having multiple offsets Is, if you are machining multiple parts in one job, each part can be assigned its own offset. This allows the CNC machine to accurately relate its coordinate system to multiple parts in different places and complete multiple setups at once.

Fig 4



Converts part dimensions into G90 and G91

Objectives: At the end of this lesson you shall be able to
- **describe methods of converting part drawing into CNC co-ordinate system**
- **explain the method of indicating the tool positions**
- **explain the difference between absolute coordinates and incremental coordinates.**

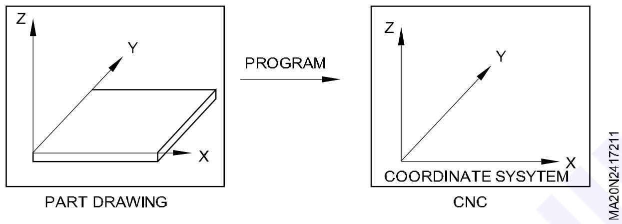
Co-ordinate system on part drawing and co-ordinate system specified by CNC co-ordinate system

Explanations

- **Co-ordinate system**

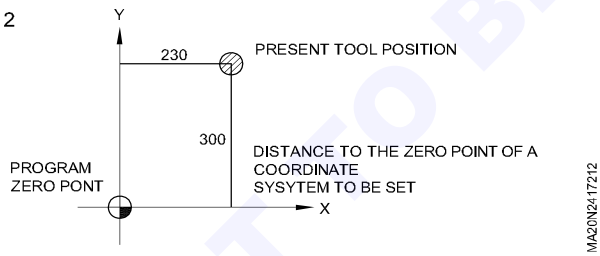
1 Co-ordinate system on part drawing used for the program data. The origin of this co-ordinate system is established on a convenient point on the work piece. Hence from now onward this is called work co-ordinate system. (Fig 1)

Fig 1



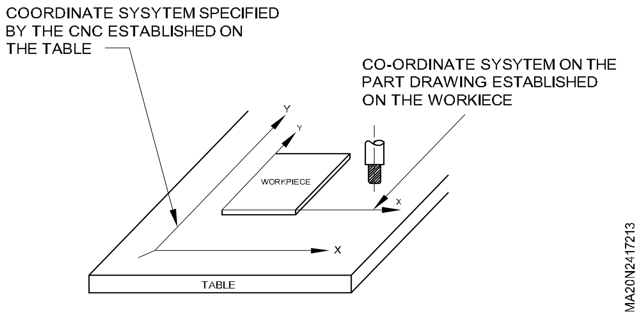
2 Co-ordinate system specified by the CNC. The co-ordinate system is prepared on the actual machine tool table. This can be achieved by programming the distance from the current position of the tool to the zero point of the co-ordinate system to be set. (Fig 2)

Fig 2



The positional relation between these two co-ordinate systems is determined when a work piece is set on the table. (Fig 3)

Fig 3



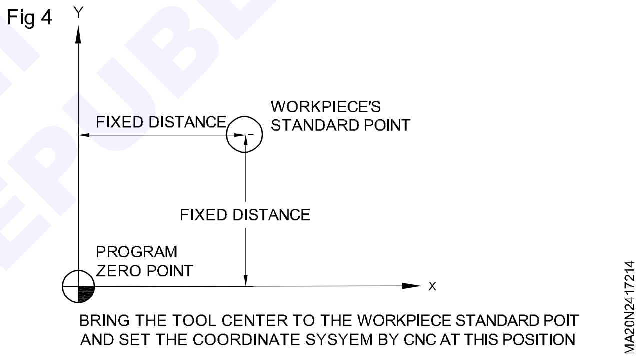
The tool moves on the co-ordinate system specified by the CNC in accordance with the command program generated with respect to the co-ordinate system on the part drawing. Therefore, in order to correctly cut the work piece as specified on the drawing, the two co-ordinate systems must be set at the same position.

- **Methods of setting the two co-ordinate systems in the same postion**

  To set the two co-ordinate systems at the same position, simple methods shall be used according to work piece shape, the number of machining.

1 Using a standard plane and point of the work piece.

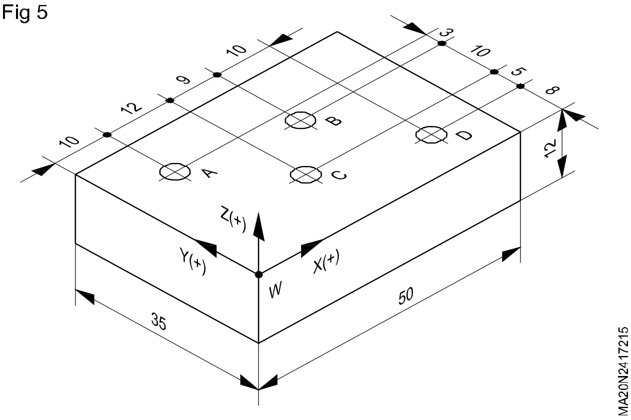
Fig 4



Absolute coordinates (G90)

Having chosen the programme zero point in the work piece, the position of the tool has to reach, is indicated in programme by means of its distance from the programme/ work zero point. These distances are called absolute coordinates and depending on the position to be reached its value may be positive or negative, In figure 5 'W' is the work zero point and the positive directions of the axes are indicated by arrows.

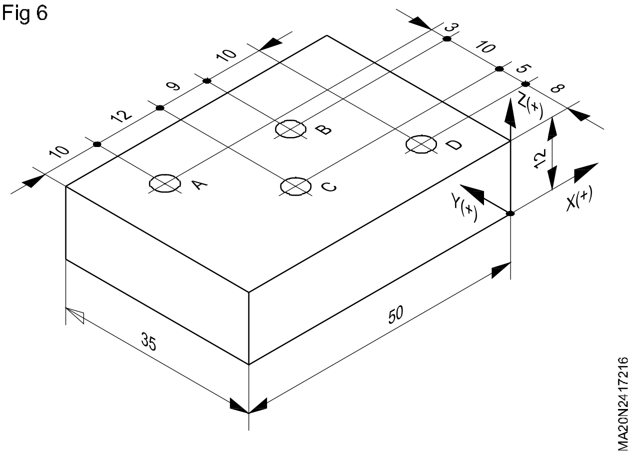
Fig 5



Hole position	X	Y	Z
A	10	26	0
B	31	23	0
C	22	13	0
D	41	8	0

Also shown in figure are the absolute coordinates of the positions of holes A, B, C, & D. Fig 6 shows the same component with work zero chosen at a different point in the workpiece.

Fig 6



Notice that the absolute coordinates of the holes ABCD are now changed both in its value and sign. The preparatory code G90, when written along or before a motion command, is an indication to the control that includes of X,Y, and Z in the block are the absolute coordinates of the position to be reached.

Hole position	X	Y	Z
A	-40	+26	+12
B	-19	+23	+12
C	-28	+13	+12
D	-9	8	+12

Incremental coordinates (G91)

This is an alternative method of indicating the tool position. In this method, the values of X Y and Z in motion blocks will be the distance to be travelled by the tool in different distance from its existing position.

These distances are called incremental co-ordinals and they may be positive or negative. In a programme, preparatory code G91 tells the control that XYZ values are incremental coordinates. It should be remembers the current position of the tool and when it sees an incremental coordinate in the programme the respective axis will simply move by that much distance in the positive or negative direction as per the sign of the coordinate

Example

The incremental coordinate for the tool to reach B from A both in Figs 5 and 6 when the tool was at A would be X21, coordinals will be X-21, Y3.

It should be emphasised that the indication of tool position by incremental coordinates is possible only if we know the programme initial position of the tool.

When the initial position of the tool is not known to the programmer, the programme should be written only in absolute coordinates. For example, in Fig 5, to make the tool reach the position B, when we do not know the existing position of the tool, we should use only absolute coordinates the values of which will depend on the position of the programme zero/work zero only. In this example, programme zero is at W, and hence, the absolute coordinate of B is X31 Y23.

Choice of G90/G91

The choice of the method G90/G91 for positioning is largely dependent on the method of dimensioning adopted in the part drawing. The only consideration here is that the coordinates should be obtained from the drawing with minimum calculation.

Format

N_G90

N_G01 X_Y_Z_F

Programming

Objectives: At the end of this lesson you shall be able to
- **state the CNC program**
- **brief the various programming codes in CNC**
- **explain a program block and the layout**
- **state about the simulations.**

A CNC program is a step-by-step instruction that tell the machine as to what action it has to take for machining a part.

Programme are initially written on paper using codes

consisting of alphabets and numbers.

Before we can machine the part, the completed programme has to be fed into the memory of the machine computer and this can be done in different ways.

But the most easily available method is to enter the codes direct into the machine using the keys/buttons provided on the control panel, and through electronic media. (Figs 1& 2)

Data entering

Data entering can be done in 2 ways:

By key board typing on the CNC milling machine (Fig 1)

Fig 1

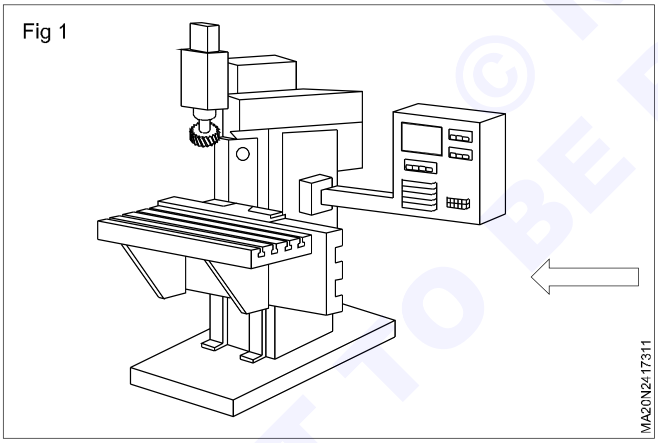
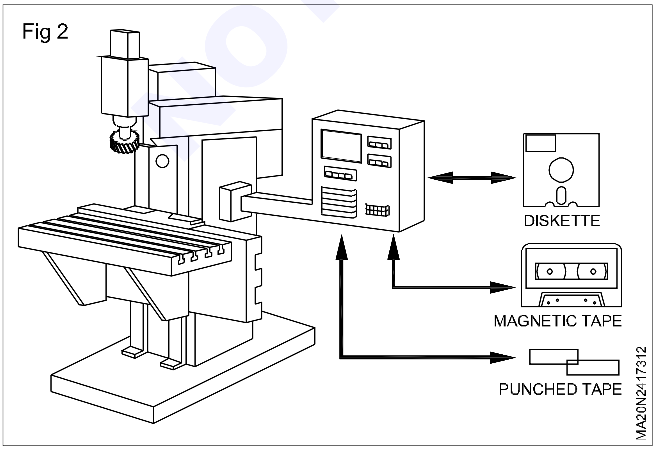


Fig 2



Programme codes

The alpha numeric codes used for programme writing are. in a way, machine-dependent because the codes may vary from machine to machine. The codes pertaining to a particular make of machine are available in the machine manual and it should be thoroughly studied before writing/ reading a programme.

However, ISO standards are available for CNC programming unless otherwise stated, all programming codes discussed in this book will follow ISO codes/ Sinumeric system / Fanuc system.

Few codes are,

G00

G01

M1

M2

T2

F90 etc.

Programme block and programme word

A line in the programme is called a block and the individual codes in the blocks are called programme words. The alphapet in these words are called address letters.

Example

N1 G00 X100 Y10 Z20

Block structure

The term block structure/format refers to the rules for writing a programme block. In otherwords it refers to the essential programme words/codes that should be written along with certain commands.

The first word of any block is the block identification number and they are formed by the letter N.

Thus N1, N2, N10, N20 etc are block numbers and they do not carry any specific instructions to the machine.

Block numbers can be written serially (N1, N2, N3, N4, etc) or in suitable increments like N10, N20, N30, etc)

Layout of a CNC-Program

A CNC-Program, also known as part program, consists of a logical sequence of commands, which are executed step-by-step the control unit after the program has been started.

Each program is compiled and stored under a program name in the control unit. The name can contain letters as well as numbers.

A block starts with a block number followed by the commands.

Each command consists of command words, which in turn consist of an address letter (A-Z) and an associated numerical value. Both upper or lower case characters are permissible)

Program layout

The block number is a program-technical assignment which is not evaluated by the control unit as a command. It is usually programmed to go up in steps of 10 and serves only the user for better oversight. It has no effect on the program execution.

The geometrical data include all instructions that clearly define mathematically the motion of the tool or the axes.

The technological data are used for instance to activate the required tool and to re-select the necessary cutting parameters feed rate and spindle speed. Miscellaneous functions can control for example such things as direction of rotation and auxiliary appliances. (Table 1)

Table 1

Departure information (Geometrical data switching information (Technological data)

Block Nr...	Auxillary command	Co-ordinate axes			Interpolation parameter			Feed	Speed	Tool	Misc. function
N	G	X	Y	Z	I	J	K	F	S	T	M

Programming example

....

N80 T1; Roughing tool

N90 M6

N100 G54 F0.2 S180 M4

N110 G00 X20 Y0Z2 D1

N120 ....

In order to improve the oversight within a program commentaries can be optionally added at the end of a block. These must be preceded by a semicolon(;). Any characters that follow thereafter will not be taken account of by the control unit..

Simulation

Mechanical parts with free form surfaces used in various industries (moulds, automotive, aerospace, etc...) are machined on multi-axis CNC milling machines because of their highly complex geometric shapes. Toolpaths for obtaining these parts are generated by taking into account several parameters (cutting conditions, tools shapes, surface models, etc...)

The final shape of the part is obtained in three operations:roughing, semi-finishing and finishing. Before real machining. it is essential to simulate virtually the machining to verify the toolpaths geometry of the finished part and to predict physical factors that are necessary to optimize the cutting parameters.

During simulation it is necessary to lock all the machine axis and auxiliary functions.

Address codes

Objectives: At the end of this lesson you shall be able to
- **describe the list of address characters applied in CNC machine tools**
- **explain the features of the address characters.**

The lists of address characters as per DIN 66025 are as follows

Word address	Description
A	Rotation about X axis
B	Rotation about Y axis
C	Rotation about Z axis

D & E	Rotation about additional axes
F	Feed rate
G	Preparatory function
H	Tool height offset
I	Circular interpolation parameter or thread pitch parallel to X axis
J	Circular interpolation parameter or thread pitch parallel to Y axis
K	Circular interpolation parameter or thread pitch parallel to Z axis
L	Number of repetitions
M	Miscellaneous function
N	Block number
O	Program number
P, Q & R	Third movement parallel to X,Y and Z axes respectively
S	Spindle speed
T	Tool function
U, V & W	Second movement parallel to X, Y and Z axes
X	Movement in X axis
Y	Movement in Y axis
Z	Movement in Z axis

The word address format is based on a combination of one letter or more digits.

Character,

A smallest unit in CNC programming. It can be a letter, digit or symbol. Example : X,Y,Z, 1-9,+,-, etc.

X	-	3	.	0

Word,

Word is a combination of character. It consists of a capital letter, followed by a number, and sometime symbol, depending on the code. Example : N5, G01, F100, G91

N	5	G	0	1	Z	-	3	2	9	.	5

Block,

Block is a multiple of words seperated by end of block, otherwise each line in the programme is a block.

N	1	0	G	0	2	X	3	9	.	7	Z	-	8	7	R	2	F	.	5

G-codes and M-codes

Objective: At the end of this lesson you shall be able to
- **explain the 'G' codes and 'M' codes used in CNC programming.**

Program

Program is required to operate NC machine tools. Programming is the creation of a serious of commands statements. All operations of the machine including "spindle rotation", "tool movement" can be controlled by a program.

Program number

Program number is use for identify the program, it must be always on the top of the program it denotes by the address "O"

00001 to **0**9999 (any number can be used from 0001 to 9999) but total number of program can be stored depends upon machine series.

End of block (EOB)

: → EOB means end of the block which must be always used at the end of the each block.

Decimal point

Decimal points are used to input the units like distance, time, and angles. If the decimal points eliminated, the systems read in microns. Decimal points can be input for the following address.

X, Y, Z, A, B, C, I, J, K, R, Q, F

1 mm	**=**	**1000 microns**
0.1 mm	**=**	**100 microns**
0.01 mm	**=**	**10 microns**
0.001 mm	**=**	**1 microns**

Character coding

The EIA (Electronic Industry Association) and AIA (Aerospace Industries Association) used this code for the machine models.

This code has eight track with seven hits of information codes, as

Character coding is divided into two types:

1 Dimension coding, and

2 Alphabet coding.

EIA codes for characters

The list of characters used by EIA in NC machines is presented in table 1.

TABLE 1

EIA character codes

Character	Purpose
N	Sequence number of data book
G	Preparatory function
X	Amount of X axis travel (inch or mm)
Y	Amount of Y axis travel (inch or mm)
Z	Amount of Z axis travel (inch or mm)
R	Clearance plane for fixed cycle (inch or mm)
I	Arc centre coordinate parallel to X - axis (inch or mm)
J	Arc centre coordinate parallel to Y - axis (inch or mm)
K	Arc centre coordinate parallel to Z - axis or depth increment for G83 cycle (in or mm)
F	Feed rate
S	Spindle speed
T	Tool number
H	Tool length compensation
D	Cutter radius compensation
E	Fixture offset
L	Dwell time for fixed cycles
M	Miscellaneous functions
Q	Sub routine (macro)
SP	To give space between any two NC words
DEL	Delete, when the wrong code is punched into the tape, then the error is cleared by this code.

Preparatory functions (G - words)

These are preset function associated with the movement of machine axes and the associated geometry. It prepares the machine control unit for the instruction and data contained in the block. The common G -codes and their functions are given in table 2.

Modal and non - modal G - codes

Some of the G - codes are modal, which means that they behave as settings to the control. Once given they remain operational till cancelled by another G codes from the same group. A few other G - codes are non - modal, which means that they remain operational in the block in which they are programmed.

For example, G20 (Inch input) is a modal command, which can be cancelled by G21 (Metric input) command.

List of G codes and the format (Table 2)

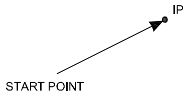
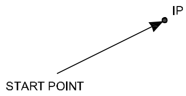
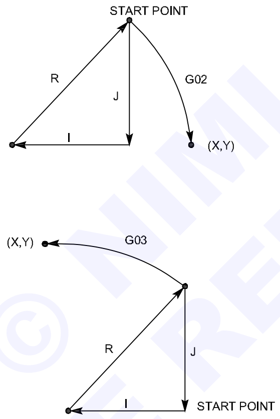
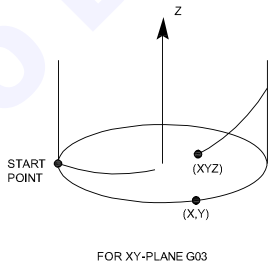
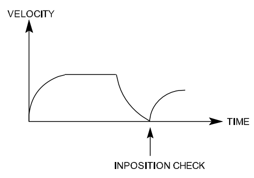
Some functions cannot be addes as options depending on the model. In the tables below. IP_presents a combination of arbirary axis addresses using X, Y, Z, A, B and C (such as X_Y_Z_A_)

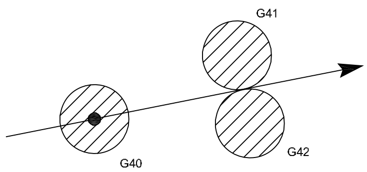
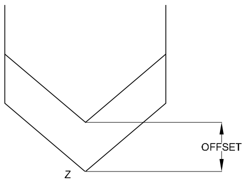
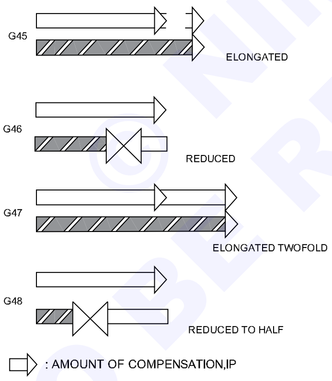
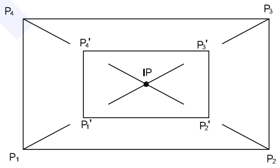
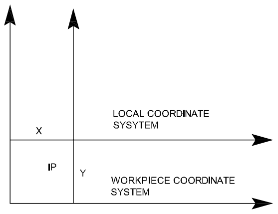
x = List basic axis (X usually)

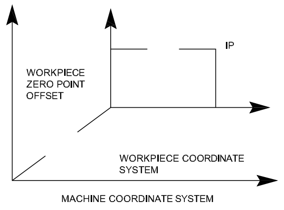
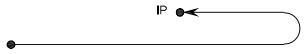
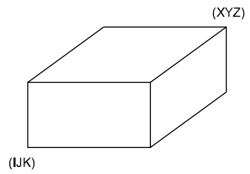
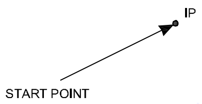
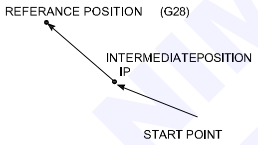
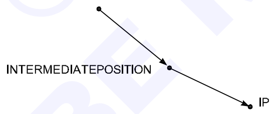
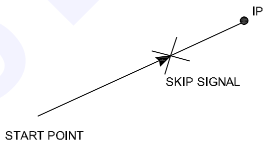
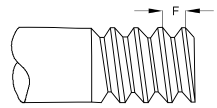
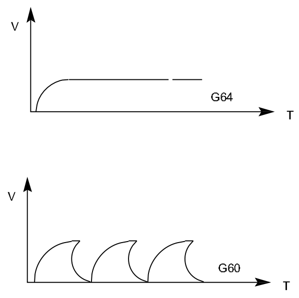
y = 2nd basic axis (Y usually)

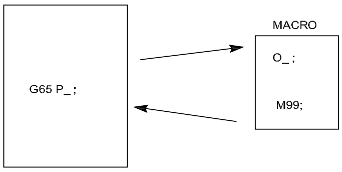
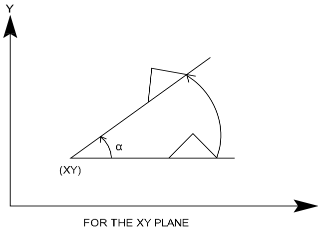
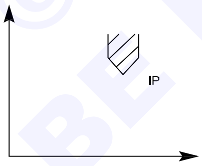
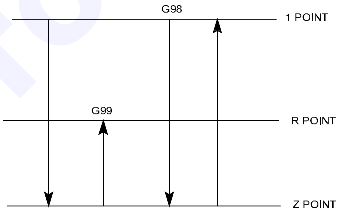
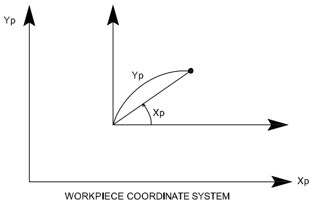
z = 3rd basic axis (Z usually)

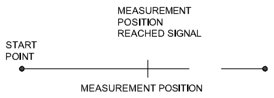
Table 2

Functions	Ilustration	Tape format
Positioning (G00)		G00P_;
Linear interpolation (G01)		G01 IP_F_;
Circular interpolation (G02, G03)		$G17\begin{Bmatrix}G02\\G03\end{Bmatrix}X\_Y\_\begin{Bmatrix}R\_\\I\_J\_\end{Bmatrix}F\_;$ $G18\begin{Bmatrix}G02\\G03\end{Bmatrix}X\_Z\_\begin{Bmatrix}R\_\\I\_K\_\end{Bmatrix}F\_;$ $G19\begin{Bmatrix}G02\\G03\end{Bmatrix}Y\_Z\_\begin{Bmatrix}R\_\\J\_K\_\end{Bmatrix}F\_;$
Helical cutting (G02, G03)	 For XY-plane G03	$G17\begin{Bmatrix}G02\\G03\end{Bmatrix}X\_Y\_\begin{Bmatrix}R\_\\I\_J\_\end{Bmatrix}\alpha\_F\_;$ $G18\begin{Bmatrix}G02\\G03\end{Bmatrix}X\_Z\_\begin{Bmatrix}R\_\\I\_K\_\end{Bmatrix}\alpha\_F\_;$ $G19\begin{Bmatrix}G02\\G03\end{Bmatrix}Y\_Z\begin{Bmatrix}R\_\\J\_K\_\end{Bmatrix}\alpha\_F\_;$ α = Arbitary address except a circular interpoation axis
Dwell (G04)		$G04\begin{Bmatrix}X\_\\P\_\end{Bmatrix};$
Exact stop (G09)		$G09\begin{Bmatrix}G01\_\\G02\_\\G03\_\end{Bmatrix}\_;$

Functions	Ilustration	Tape format
Change of offset value by program (G10)		Tool ofset value (offset memory B) G10 L10 P_R_; Wear offset value (offset memory B) G10 L11 P_R_; Work piece origin offset value G10 L2 P_IP_; G10 L20 P_IP_;
Cutter compensation (G40-G42)		$\left\{\begin{matrix}G17\\G18\\G19\end{matrix}\right\}\left\{\begin{matrix}G40\\G41\\G42\end{matrix}\right\}P\_\left\{\begin{matrix}H\_;\\D\_;\end{matrix}\right\}$ H.D : Cutter compensation number G40: Cancel
Tool length offset (G43, G44, G49)		$\left\{\begin{matrix}G43\\G44\end{matrix}\right\}\alpha\_H\_;$ $\left\{\begin{matrix}G43\\G44\end{matrix}\right\}H\_;$ H: Tool offset number G49: Cancel
Tool offset (G45 - G48)		$\left\{\begin{matrix}G45\\G46\\G47\\G48\end{matrix}\right\}IP\_D\_;$ D: Tool offset number P: Scale factor
Scaling (G50, G51)		G51 IP_P_; P ; G50 ; Cancel
Local co-ordinate system setting (G52)		G52 IP_;
Machine co-ordinate system selection (G53)		G53 IP_;

Functions	Ilustration	Tape format
Work piece co-ordinate system selection (G54 - G59)	 	$\left\{\begin{array}{c}G54\\ :\\ G59\end{array}\right\}IP\_;$
Single direction positioning (G50)	 	G60 IP_;
Inch/Millimeter conversion (G20, G21)		G20 ; Inch input G21 ; Millimeter input
Stored stroke check (G22, G23)	 	G22 X_ Y_ Z_ I_ J_ K_ G23 ; Cancel
Reference position return check (27)	 	G27 IP_;
Reference position return (G28) 2nd, 3rd, 4th reference position return (G30)	 	G28 IP_; $G30P\left\{\begin{array}{c}P2\\ P3\\ P4\end{array}\right\}IP\_;$
Return from reference position to start point (G29)	 	G29 IP_;
Skip function (G31)	 	G31 IP_F_;
Thread cutting (G33)	 	Equal lead thread cutting G33 IP_F_;
Cutting mode, Exact stop mode, Tapping mode, Automatic corner override	 	G64_; Cutting mode G60_; Exact stop mode G62_; Automatic corner override mode G63_; Tapping mode

Functions	Ilustration	Tape format
Custom macro (G65, G66, G67)		One-shot call G65P <argument> P: Program number Modal call G66 P <argument> G67; cancel
Co-ordinate system rotation (G68, G69)		$G68\left\{\begin{array}{l}G17X\_Y\_\\G18Z\_X\_\\G19Y\_Z\_\end{array}\right\}R\_\alpha\_$ G69: Cancel
Canned cycles (G73, G74, G80 - G89)	FUNCTIONS TO SIMPLIFY PROGRAMMING	G80 ; Cancel $\left\{\begin{array}{l}G73\\G74\\G76\\G81\\:\\G89\end{array}\right\}X\_Y\_Z\_P\_Q\_R\_F\_K\_$
Absolute/Incremental programming (G90/G91)		G90_; Absolute command G91_; Incremental command G90_G91_; Combined use
Change of work piece co-ordinate system (G92)		G92IP_;
Initial point return/R point return (G98, G99)		G98 - -; Initial point G99 - -; R - point
Constant surface speed control (G96, G97)	m/min or feet/min	G96 S_; G97 ; Cancel
Polar co-ordinate input (G15, G16)	Local co-ordinate system 	G17 G16 Xp_Yp...; G18 G16 Zp_Xp...; G19 G16 Yp_Zp...; G15; Cancel

Functions	Illustration	Tape format
Automatic tool length measurement (G37)		G37 Z...;

Miscellaneous functions or auxiliary functions (M - codes): These functions actually operate some controls on the machine tool and thus affect the running of the machine. The particular machine tool must have the function that is being called.

Miscellaneous commands are normally placed at the end of the block. The common M - codes and their functions are given in table 3.

Miscellaneous functions (M codes) (Table 3)

Number	Operation	Definition
M00	Program stop	A miscellaneous function command to cancel the spindle, coolant functions and terminate further program execution.
M01	Optional (Planned) stop	A miscellaneous function command similar to a program stop except that the control ignores the command unless the operator has previously validated the command.
M02	End of program	A miscellaneous function indicating completion of workpiece. Stops spindle coolant and feed after completion of all commands in the block. Used to reset control and/or machine. Resetting control may include rewind of tape to the end of record character or progressing a loop tape through the splicing leader.
M03	Spindle CW	Start spindle rotation in clockwise direction.
M04	Spindle CCW	Start spindle rotation in counter clockwise direction.
M05	Spindle OFF	Stop spindle in normal manner; brake if available applied; coolant turned OFF.
M06	Tool change	Stops spindle and coolant and retracts tool to full retract position. It should be coded in last block of information in which a given tool is used.
M07 - M08- M09	Coolant, ON, OFF	Mist (No.2) Flood (No.1) tapping coolant or dust collector.
M10- M11 M12	Clamp, unclamp Synchronization code	Can pertain to machine slides, workpiece, fixtures, spindle, etc. An inhibiting code used for synchronization of multiple sets of axes.
M15 - M16	Motion +, Motion -	Rapid traverse of feed direction selection where required.
M19	Oriented spindle stop	A miscellaneous function which causes the spindle to stop at a predetermined or programmed angular position.
M26	Pseudo tool change	Retracts tool from gage height to tool change position. Used primarily to avoid clamps or part obstructions.
M30	End of data	A miscellaneous function which stops spindle coolant and feed after completion of all commands in the block. Used to reset control and/or machine. Resetting control will include rewind of tape to the end of record character progressing a loop tape through the splicing leader or transferring to a second tape reader.
M31	Interlock by - pass	A command to temporarily circumvent a normally provided interlock.
M47	Return to program start	A miscellaneous function which continuous program execution from the start of program unless inhibited by an interlock signal.
M49	Over ride by - pass	A function which deactivates a manual spindle or feed override and returns the parameters to the programmed value. Cancelled by M48.
M59	CSS by - pass updating	A function which holds the RPM constant at its value when M59 is initiated/ cancelled by M58.
M90 - M99	Reserved for user	Miscellaneous function outputs which are reserved exclusively for the machine user.

Explanation of M-codes

Objective: At the end of this lesson you shall be able to
- **specify the explanation of M codes.**

M codes detailed description

M00 → Program stop (Unconditional stop)

The M00 function is defined as a unconditional or compulsory program stop. Any time the control system encounters this function during program processing, all automatic operations of the machine tool will stop.

Motion of all axes

Rotation of the spindle

Coolant function

Further program execution

The control will not be reset when the M00 function is processed. All significant program data currently active are retained (feed rate, coordinate setting, spindle speed, etc.) The program processing can only be resumed by activating the cycle start key. The M00 functions cancels the spindle rotation.

The M00 code is used to stop a program. It also stops the spindle and turns off the coolant and stops interpretation look ahead processing. The program pointer will advance to the next block and stop. A cycle start will continue program operation from the next block. If the through the spindle coolant option is ON, M00 will shut it OFF.

M01 → Optional program stop (Conditional stop)

The M01 code is identical to M00 except that it only stops if optional stop is turned on from the front panel. A cycle start will continue program operation from the next block. If the though the spindle coolant option is on, M01 will shut it OFF.

M02 → Program end

The M02 code will stop program operation the same as M00 but does not advance the program pointer to the next block.

M03 → Spindle on clockwise direction

The M03 code will start the spindle moving is a clockwise direction at whatever speed was previously set.

M04 → Spindle on counter clockwise direction

The M04 code will start the spindle moving is a counter clockwise direction at whatever speed was previously set.

M05 → Spindle stop

The M05 code is used to stop the spindle.

M06 → Tool change

The M06 code is used to initiate a tool change. The previously selected tool (Tn) is put into the spindle. If the spindle was running, it will be stopped. No previous axis commands are required before the tool change unless there is a problem with tool/part/fixture clearance. The Z - axis will automatically move up to the machine zero position and the selected tool will be put into the spindle. The Z - axis is left at machine zero. The spindle will not be started again after the tool change but the snnnn speed and gear will be unchanged. The Tnn must be in the same block or in a previous block. The coolant pump will be turned off during a tool change.

Example: M06 T01

M08 → Coolant ON

The M08 code will turn on the coolant supply. Note that the M code is performed at the end of a block; so that if a motion is commanded in the same block, the coolant is turned on after the motion. The low coolant status is only checked at the start of a program so a low coolant condition will not stop a program which is already running.

M09 → Coolant OFF

The M09 code will turn off the coolant supply.

M19 → Orient spindle (P,R) (option)

The M19 code is used to orient the spindle to a fixed position. The spindle is oriented electronically. A p value can be added as an option that will cause the spindle to be oriented to a particular angle (in degrees). For example, M19 P270 will orient the spindle to 270 degrees. An R value will recognize up to four places to the right of the decimal point.

M21- M28 → Optional user M code

The M21 through M28 codes are optional for user interfaces. They will activate one of relays 1132 through 1139, wait for the M - fin signal, releases the relay, and wait for the M - fin signal to cease. The reset button will terminate any operation that is hung - up waiting for M - fin.

M30 → Program end and reset

The M30 code is used to stop a program. It also stops the spindle and turns off the coolant. The program pointer will be reset to the first block of the program and stop. The parts counters displayed on the current commands display are also incremented. M30 will also cancel tool length offsets. When the through the spindle coolant (TSC) option is On, M30 will shut it OFF, and then perform and M30 operation.

M97 → Local sub - routine call (P, L)

This code is used to call a sub - routine, referenced by a line number N within the same program. A Pnnnnn code is required and must match the N line number. This is used for simple sub - routines within a program and does not require the complication of having a separate program. A local sub routine must still end with an M99. If there is an L count on the M97 line, the sub- routine will be repeated that number of times.

Main Program

```
O04321 (Start of main program)
.........    (Part program)
.........
M97 P123 (Jumps to line N123, after the M30, to execute
a local sub - routine.)
... (The M99 at the end of the sub - routine will cause it to
jump back here.)
...
... (Finish part program)
...
M30 (End of main program)
N123 (Identifies the start of the local sub - routine called
up by M97 P123)
...
... (Local sub - routine portion of part)
...
M99 (Jumps back to the line after the local sub - routine
call in the main program)
```

A canned cycle consists of a sequence of six operations (Fig 1)

Operation 1----------Positioning of axes X and Y(including also another axis)

Operation 2----------Rapid traverse up to point R level

Operation 3----------Hole machining

Operation 4----------Operation at the bottom of a hole

Operation 5----------Retraction to point R level

Operation 6----------Rapid traverse up to the initial point

Important restrictions

1. Drilling is not performed when cycle call block does not contain X, Y, Z, R or any other axis.
2. Specify Q and/or R in blocks that perform drilling. If they are specified in a block that does not contain drilling they cannot be stored as modal data.
3. In a cycle call block do not specify 01 Group G codes (G00 to G03). This will cancel the canned cycle Gcodes.

Canned cycles

Objectives: At the end of this lesson you shall be able to
- **state the canned cycles and its specific uses**
- **explain the six operations in canned cyles**
- **brief the format for G81 to 89.**

Canned cycle

Canned cycles make it easier for the programmer to create programs. With a canned cycle, a frequently–used machining operation can be specified in a single block with a G function; without canned cycles, normally more than one block is required. In addition, the use of canned cycles can shorten the program to save memory. Table 1, Lists canned cycles.

Table 1

Canned cycles

G code	Drilling (–Z direction)	Operation at the bottom of a hole	Retraction(+Z direction)	Application
G73	Intermittent feed	–	Rapid traverse	High–speed peck drilling cycle
G74	Feed	Dwell→Spindle CW	Feed	Left–hand tapping cycle
G76	Feed	Spindle orientation	Rapid traverse	Fine boring cycle
G80	–	–	–	Cancel
G81	Feed	–	Rapid traverse	Drilling cycle, spot drilling cycle
G82	Feed	Dwell	Rapid traverse	Drilling cycle, counter boring cycle
G83	Intermittent feed	–	Rapid traverse	Peck drilling cycle
G84	Feed	Dwell”Spindle CCW	Feed	Tapping cycle
G85	Feed	–	Feed	Boring cycle
G86	Feed	Spindle stop	Rapid traverse	Boring cycle
G87	Feed	Spindle CW	Rapid traverse	Back boring cycle
G88	Feed	Dwell→spindle stop	Manual	Boring cycle
G89	Feed	Dwell	Feed	Boring cycle

Explanations

A canned cycle consists of a sequence of six operations (Fig 1)

Operation 1---------- Positioning of axes X and Y (including also another axis)

Operation 2---------- Rapid traverse up to point R level

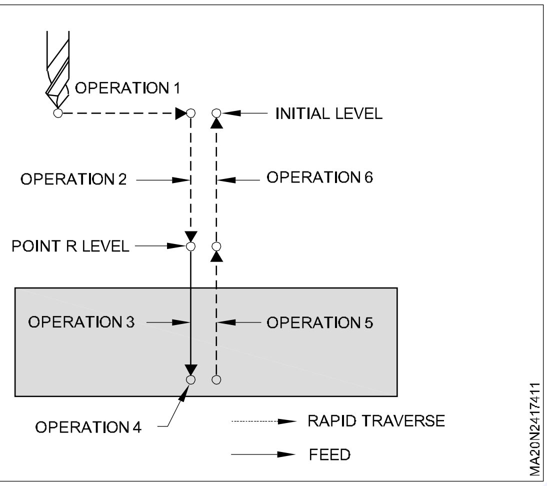
Operation 3---------- Hole machining

Operation 4---------- Operation at the bottom of a hole

Operation 5---------- Retraction to point R level

Operation 6---------- Rapid traverse up to the initial point

Fig 1



Positioning plane

The positioning plane is determined by plane selection code G17, G18 or G19.

The positioning axis is an axis other than the drilling axis.

Drilling axis

Although canned cycles include tapping and boring cycles as well as drilling cycles, in this chapter, only the term drilling will be used to refer to operations implemented with canned cycles.

The drilling axis is a basic axis (X, Y, or Z) not used to define the positioning plane, or any axis parallel to that basic axis.

Positioning plane and drilling axis

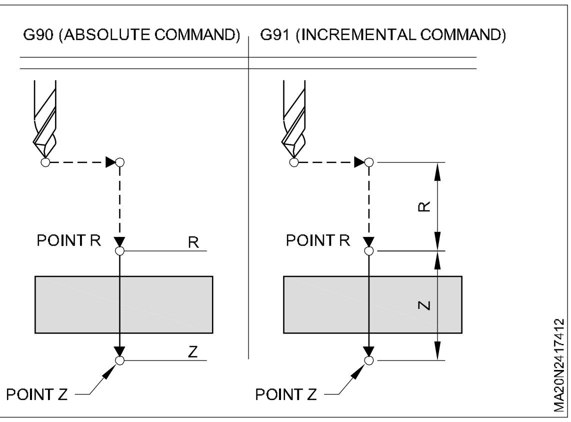
Table 2

G code	Positioning plane	Drilling axis
G17	Xp–Yp plane	Z axis
G18	Zp–Xp plane	Y axis
G19	Yp–Zp plane	X axis

Travel distance along the drilling axis G90/G91

The travel distance along the drilling axis varies for G90 and G91 as follows: (Fig 2)

Fig 2



Drilling mode

G73, G74, G76, and G81 to G89 are modal G codes and remain in effect until canceled. When in effect, the current state is the drilling mode.

Once drilling data is specified in the drilling mode, the data is retained until modified or canceled.

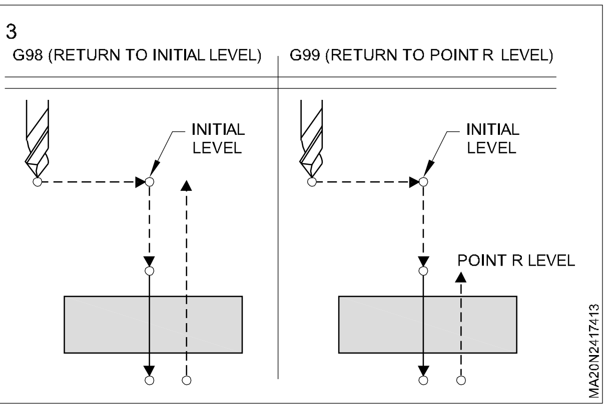
Specify all necessary drilling data at the beginning of canned cycles; when canned cycles are being performed, specify data modifications only.

Return point level G98/G99

When the tool reaches the bottom of a hole, the tool may be returned to point R or to the initial level. These operations are specified with G98 and G99. The following illustrates how the tool moves when G98 or G99 is specified. Generally, G99 is used for the first drilling operation and G98 is used for the last drilling operation.

The initial level does not change even when drilling is performed in the G99 mode. (Fig 3)

Fig 3



Repeat

To repeat drilling for equally–spaced holes, specify the number of repeats in K_.

K is effective only within the block where it is specified.

Specify the first hole position in incremental mode (G91).

If it is specified in absolute mode (G90), drilling is repeated at the same position.

Number of repeats K The maximum command value = 9999

If K0 is specified, drilling data is stored, but drilling is not performed.

Cancel

To cancel a canned cycle, use G80 or a group 01 G code.

Group 01 G codes

G00 : Positioning (rapid traverse)

G01 : Linear interpolation

G02 : Circular interpolation or helical interpolation (CW)

G03 : Circular interpolation or helical interpolation (CCW)

Symbols in figures

Subsequent sections explain the individual canned cycles. Figures in these explanations use the following symbols:

→	Positioning (rapid traverse G00)
→	Cutting feed (linear interpolation G01)
⤳	Manual feed
OSS	Spindle orientation (The spindle stops at a fixed rotation position)
⇒	Shift (rapid traverse G00)
P	Dwell

Drilling cycle, spot drilling (G81)

This cycle is used for normal drilling. Cutting feed is performed to the bottom of the hole. The tool is then retracted from the bottom of the hole in rapid traverse

Format

G81 X_ Y_ Z_ R_ Q_ F_ K_ ;

X_ Y_ : Hole position data

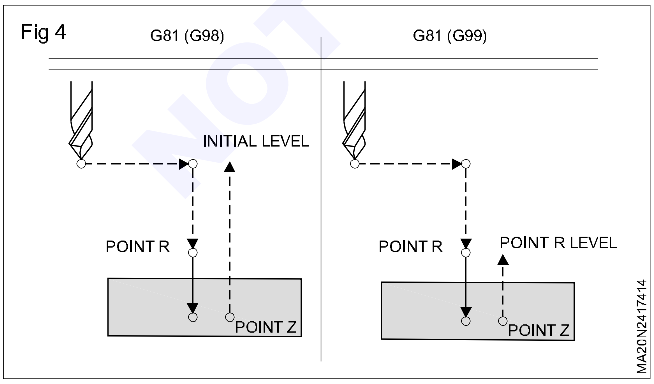
Z_ : The distance from point R to the bottom of the hole

R_ : The distance from the initial level to point R level

F_ : Cutting feedrate

K_ : Number of repeats (Fig 4)

Fig 4



Explanations

After positioning along the X– and Y–axes, rapid traverse is performed to point R.

Drilling is performed from point R to point Z.

The tool is then retracted in rapid traverse.

Before specifying G81, use a miscellaneous function (M code) to rotate the spindle.

When the G81 command and an M code are specified in the same block, the M code is executed at the time of the first positioning operation. The system then proceeds to the next drilling operation.

When K is used to specify the number of repeats, the M code is performed for the first hole only; for the second and subsequent holes, the M code is not executed.

When a tool length offset (G43, G44, or G49) is specified in the canned cycle, the offset is applied at the time of positioning to point R.

Examples

M3 S2000 ; Cause the spindle to start rotating.

G90 G99 G81 X300. Y–250. Z–150. R–100. F120. ;

Position, drill hole 1, then return to point R.

Y–550. ;	Position, drill hole 2, then return to point R.
Y–750. ;	Position, drill hole 3, then return to point R.
X1000. ;	Position, drill hole 4, then return to point R.
Y–550. ;	Position, drill hole 5, then return to point R.
G98 Y–750. ;	Position, drill hole 6, then return to the initial level.
G80 G28 G91 X0 Y0 Z0 ;	Return to the reference position return
M5 ;	Cause the spindle to stop rotating.

Drilling cycle counter boring cycle (G82)

This cycle is used for normal drilling.

Cutting feed is performed to the bottom of the hole. At the bottom, a dwell is performed, then the tool is retracted in rapid traverse.

This cycle is used to drill holes more accurately with respect to depth.

Format

G82 X_ Y_ Z_ R_ P_ F_ K_ ;

X_ Y_ : Hole position data

Z_ : The distance from point R to the bottom of the hole

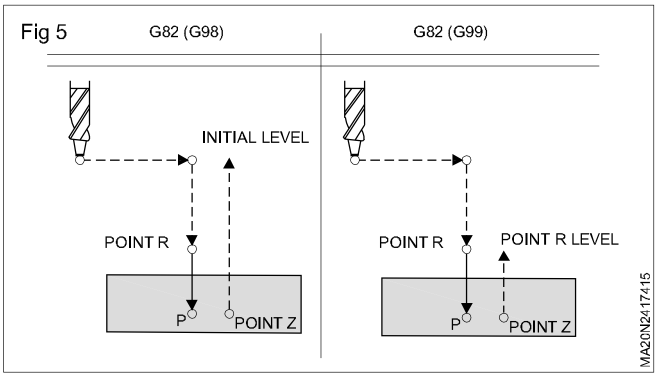
R_ : The distance from the initial level to point R level

P_ : Dwell time at the bottom of a hole

F_ : Cutting feed rate

K_ : Number of repeat (Fig 5)

Fig 5



Explanations

After positioning along the X– and Y–axes, rapid traverse is performed to point R.

Drilling is then performed from point R to point Z.

When the bottom of the hole has been reached, a dwell is performed. The tool is then retracted in rapid traverse.

Before specifying G82, use a miscellaneous function (M code) to rotate the spindle.

When the G82 command and an M code are specified in the same block, the M code is executed at the time of the first positioning operation. The system then proceeds to the next drilling operation.

When K is used to specify the number of repeats, the M code is executed for the first hole only; for the second and subsequent holes, the M code is not executed.

When a tool length offset (G43, G44, or G49) is specified in the canned cycle, the offset is applied at the time of positioning to point R.

Examples

M3 S2000 ; Cause the spindle to start rotating.

G90 G99 G82 X300. Y–250. Z–150. R–100. P1000 F120.;

	Position, drill hole 1, and dwell for 1 s at the bottom of the hole, then return to point R.
Y–550. ;	Position, drill hole 2, then return to point R.
Y–750. ;	Position, drill hole 3, then return to point R.
X1000. ;	Position, drill hole 4, then return to point R.
Y–550. ;	Position, drill hole 5, then return to point R.
G98 Y–750. ;	Position, drill hole 6, then return to the initial level.
G80 G28 G91 X0 Y0 Z0 ;	Return to the reference position return
M5 ;	Cause the spindle to stop rotating

Peck drilling cycle (G83)

This cycle performs peck drilling. It performs intermittent cutting feed to the bottom of a hole while removing shavings from the hole.

Format (Fig 6)

G83 X_ Y_ Z_ R_ Q_ F_ K_ ;

X_ Y_ : Hole position data

Z_ : The distance from point R to the bottom of the hole

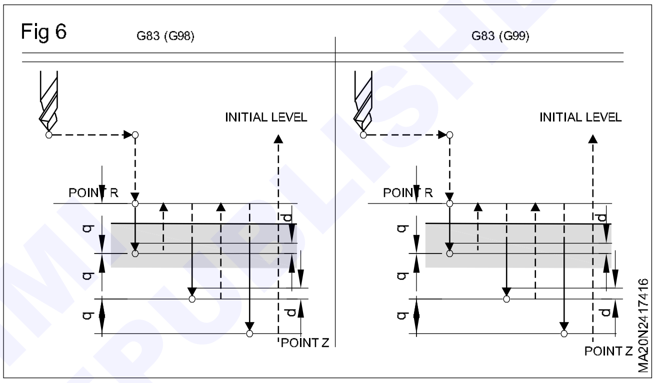
R_ : The distance from the initial level to point R level

Q_ : Depth of cut for each cutting feed

F_ : Cutting feedrate

K_ : Number of repeat

Fig 6



Explanations

Q represents the depth of cut for each cutting feed. It must always be specified as an incremental value.

In the second and subsequent cutting feeds, rapid traverse is performed up to a point just before where the last drilling ended, and cutting feed is performed again. Specify the amount of retraction in parameter No. 532. Be sure to specify a positive value in Q. Negative values are ignored. Before specifying G83, use a miscellaneous function (M code) to rotate the spindle.

When the G83 command and an M code are specified in the same block, the M code is executed at the time of the first positioning operation. The system then proceeds to the next drilling operation.

When K is used to specify the number of repeats, the M code is executed for the first hole only; for the second and subsequent holes, the M code is not executed.

When a tool length offset (G43, G44, or G49) is specified in the canned cycle, the offset is applied at the time of positioning to point R.

Examples

M3 S2000 ; Cause the spindle to start rotating.

G90 G99 G83 X300. Y–250. Z–150. R–100. Q15. F120. ;

Position, drill hole 1, then return to point R.

Y–550. ;	Position, drill hole 2, then return to point R.
Y–750. ;	Position, drill hole 3, then return to point R.
X1000. ;	Position, drill hole 4, then return to point R.
Y–550. ;	Position, drill hole 5, then return to point R.
G98 Y–750. ;	Position, drill hole 6, then return to the initial level.
G80 G28 G91 X0 Y0 Z0 ;	Return to the reference position return
M5 ;	Cause the spindle to stop rotating

Tapping cycle (G84)

This cycle performs tapping.

In this tapping cycle, when the bottom of the hole has been reached, the spindle is rotated in the reverse direction.

Format (Fig 7)

G84 X_ Y_ Z_ R_ P_ F_ K_ ;

X_ Y_ : Hole position data

Z_ : The distance from point R to the bottom of the hole

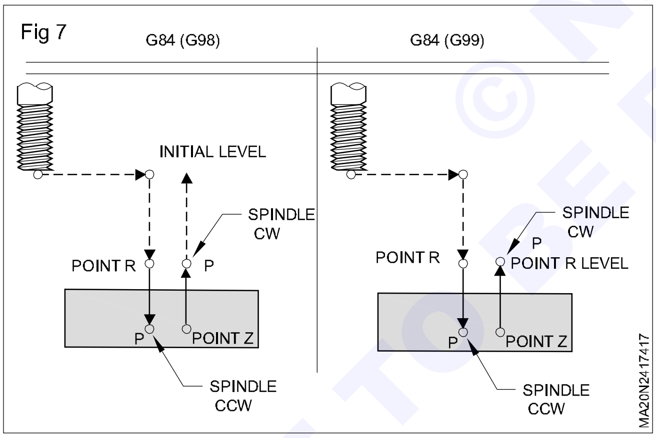
R_ : The distance from the initial level to point R level

P_ : Dwell time

F_ : Cutting feedrate

K_ : Number of repents

Fig 7



Explanations

Tapping is performed by rotating the spindle clockwise. When the bottom of the hole has been reached, the spindle is rotated in the reverse direction for retraction. This operation creates threads.

Feedrate overrides are ignored during tapping. A feed hold does not stop the machine until the return operation is completed.

Before specifying G84, use a miscellaneous function (M code) to rotate the spindle.

When the G84 command and an M code are specified in the same block, the M code is executed at the time of the first positioning operation. The system then proceeds to the next drilling operation.

When the K is used to specify number of repeats, the M code is executed for the first hole only; for the second and subsequent holes, the M code is not executed.

When a tool length offset (G43, G44, or G49) is specified in the canned cycle, the offset is applied at the time of positioning to point R.

Examples

M3 S100 ;	Cause the spindle to start rotating.
G90 G99 G84 X300. Y–250. Z–150. R–120. P300 F120. ;	Position, drill hole 1, then return to point R.
Y–550. ;	Position, drill hole 2, then return to point R.
Y–750. ;	Position, drill hole 3, then return to point R.
X1000. ;	Position, drill hole 4, then return to point R.
Y–550. ;	Position, drill hole 5, then return to point R.
G98 Y–750. ;	Position, drill hole 6, then return to the initial level.
G80 G28 G91 X0 Y0 Z0 ;	Return to the reference position return

M5 ; Cause the spindle to stop rotating

Boring cycle (G85)

This cycle is used to bore a hole.

Format (Fig 8)

G85 X_ Y_ Z_ R_ F_ K_ ;

X_ Y_ : Hole position data

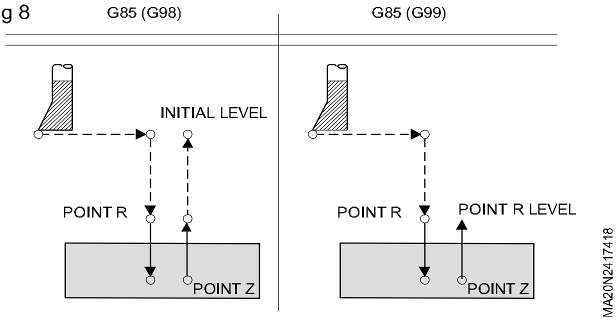
Z_ : The distance from point R to the bottom of the hole

R_ : The distance from the initial level to point R level

F_ : Cutting feed rate

K_ : Number of repeats

Fig 8



Explanations

After positioning along the X– and Y– axes, rapid traverse is performed to point R.

Drilling is performed from point R to point Z.

When point Z has been reached, cutting feed is performed to return to point R.

Before specifying G85, use a miscellaneous function (M code) to rotate the spindle.

When the G85 command and an M code are specified in the same block, the M code is executed at the time of the first positioning operation. The system then proceeds to the next drilling operation.

When K is used to specify the number of repeats, the M code is executed for the first hole only; for the second and subsequent holes, the M code is not executed.

When a tool length offset (G43, G44, or G49) is specified in the canned cycle, the offset is applied at the time of positioning to point R.

Examples

M3 S100 ;	Cause the spindle to start rotating.
G90 G99 G85 X300. Y–250. Z–150. R–120. F120. ;	Position, drill hole 1, then return to point R.
Y–550. ;	Position, drill hole 2, then return to point R.
Y–750. ;	Position, drill hole 3, then return to point R.
X1000. ;	Position, drill hole 4, then return to point R.
Y–550. ;	Position, drill hole 5, then return to point R.
G98 Y–750. ;	Position, drill hole 6, then return to the initial level.
G80 G28 G91 X0 Y0 Z0 ;	Return to the reference position return
M5 ;	Cause the spindle to stop rotating.

Fine boring cycle (G76)

The fine boring cycle bores a hole precisely. When the bottom of the hole has been reached, the spindle stops and the tool is moved away from the machined surface of the workpiece and retracted.

Format (Fig 9&10)

G76 X_ Y_ Z_ R_ Q_ P_ F_ K_ ;

X_ Y_ : Hole position data

Z_: The distance from point R to the bottom of the hole

R_ : The distance from the initial level to point R level

Q_ : Shift amount at the bottom of a hole

P_ : Dwell time at the bottom of a hole

F_ : Cutting feedrate

K_ : Number of repeats

Note: Q (shift at the bottom of a hole) is a modal value retained within canned cycles. It must be specified carefully because it is also used as the depth of cut for G73 and G83.

Fig 9

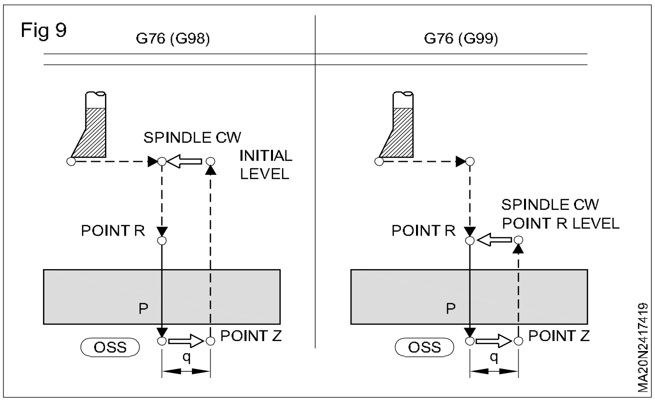
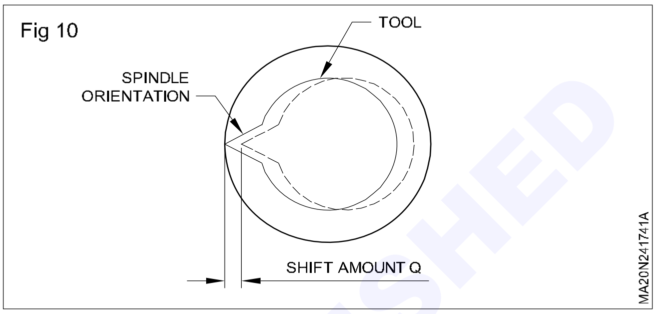


Fig 10



Explanations

When the bottom of the hole has been reached, the spindle is stopped at the fixed rotation position, and the tool is moved in the direction opposite to the tool tip and retracted. This ensures that the machined surface is not damaged and enables precise and efficient boring to be performed.

Before specifying G76, use a miscellaneous function (M code) to rotate the spindle.

When the G76 command and an M code are specified in the same block, the M code is executed at the time of the first positioning operation. The system then proceeds to the next operation.

When K is used to specify the number of repeats, the M code is executed for the first hole only; for the second and subsequent holes, the M code is not executed.

When a tool length offset (G43, G44, or G49) is specified in the canned cycle, the offset is applied at the time of positioning to point R.

Examples

M3 S500 ; Cause the spindle to start rotating.

G90 G99 G76 X300. Y–250. Position, bore hole 1, then return to point R.

Z–150. R–120. Q5. Orient at the bottom of the hole, then shift by 5 mm.

P1000 F120. ; Stop at the bottom of the hole for 1 s.

Y–550. ; Position, drill hole 2, then return to point R.

Y–750. ; Position, drill hole 3, then return to point R.

X1000. ; Position, drill hole 4, then return to point R.

Y–550. ; Position, drill hole 5, then return to point R.

G98 Y–750. ; Position, drill hole 6, then return to the initial level.

G80 G28 G91 X0 Y0 Z0 ; Return to the reference position return

M5 ; Cause the spindle to stop rotating.

Sub programming

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

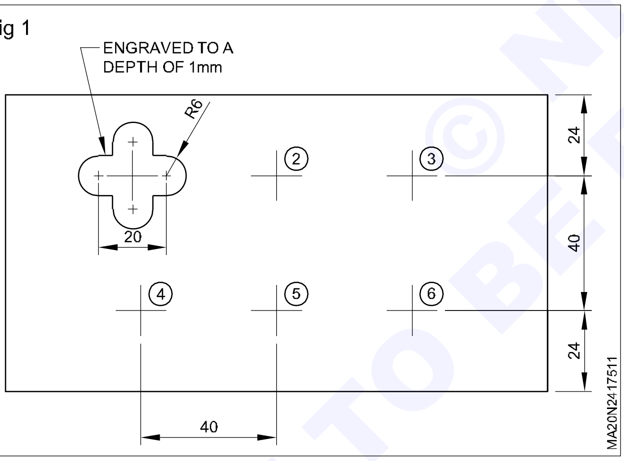
- **state the purpose of a sub-programme**
- **explain the difference between the main programme and a sub-programme.**

CNC programme may be classified into two types, namely, (1) Main programme and (2) Sub-programme.

The programme we have seen so far in the previous examples are actually main programmes.

Sub programmes, which are also called as sub-routines, are very similar to the main programme in its construction. The main purpose of a sub-programme is to-avoid repetition of programme blocks. To understand this clearly, consider the following example. The example in figure (1) shows a part with 6 identical patterns to be engraved on six locations. In such cases if we use sub-programming technique there will be no need to write the programme of the pattern six times. Instead, it is enough that we write the programme for the pattern only once. The programme so written will be a sub-programme and it should have a separate identification number. In sinumeric system sub-programme are identified by the letter L, whereas in main programmes are identified by the symbol %.

Fig 1



A sub-programme can be executed only when it is inserted in a main programme. To machine the patterns in Fig 1, the sub-programme is to be inserted at six places in the main programme are given below.

% 125 (Main programme siemens control)

```
N1 G54
N2 G90 G71 G94
N3 T1 D1 L96 S1000 M03
N4 G00 X40 Y64 Z10 - (Tool comes to position 1)
N5 L 100 P1
N6 G00 X64 - (Tool comes to position 2)
N7 L 100 P1
N8 G00 X104 - (Tool comes to position 3)
N9 L 100 P1
N10 G00 Y24 - (Tool comes to position 4)
N11 L 100 P1
N12 G0 X64 - (Tool comes to position 5)
N13 L 100 P1
N14 G00 X40 - (Tool comes to position 6)
N15 L100 P1
N16 G00 Z200
N17 M30
```

Sub-programme

```
L100
N1 G91
N2 G0 X-6 Y-6
N3 G01 Z-11 F60
N4 G01 X-4
N5 G03 X0 Y12 I0 J6
N6 G01 X4
N7 G01 Y4
N8 G03 X12 I6 J0
N9 G01 Y-4
N10 G01 X4
N11 G03 Y12 I0 J-6
N12 G01 X-4
N13 G01 Y-4
N14 G03  X-12 I0 J6
N15 G01 Y4
N16 G00 Z11
N17 G0 X6 Y6
N18 G90
N19 M17
```

Notice that the sub-programme is written in incremental mode. (These are special case where the programme must be written only in incremental mode.) To make this programme work, the tool must begin at identical posttions on every pattern and this is taken care of in ihe main programme blocks N4, N6, N8, N10, N12 and N14.

It may also be observed that the sub-programme for the pattern is inserted in blocks N5, N7, N9, N11, N13 and N15 by its number followed by the word P1. Had we not used a sub-programme, the entire block in the sub-programme will have to be written at six places.

The number value in word P1 indicates the number of times the sub-programme, is to be repeated at the inserted position. For example, if we write L 100 P3 the sub-programme will get executed 3 times in the same position.

In the programme the main programme is executed till the tool reaches the first location of operation, then to do the machining subordinate L100 is referred, after machining the tool moves to second location for which it refers the main programme again.

Subprogram

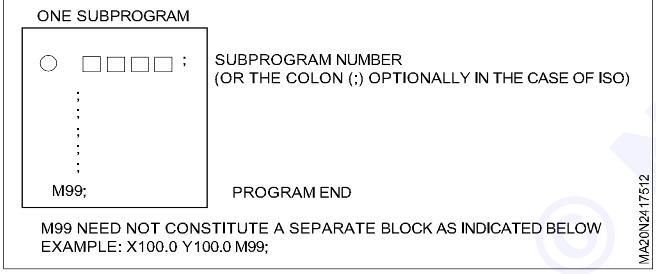
If a diagram contains a fixed sequence or frequently repeated pattern, such a sequence of pattern can be stored as a subprogram in memory to simplify the program.

A subprogram can be called from the main program.

A called subprogram can also call another subprogram.

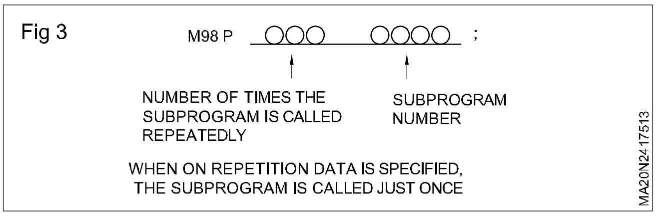
a Subprogram configuration (Fanuc control)

Fig 2



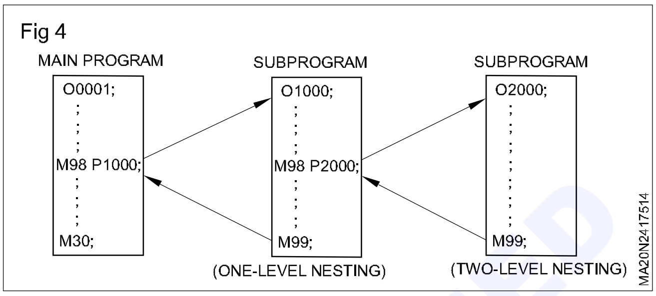
b Subprogam call

Fig 3



c Subprogam explanations

Fig 4



When the program calls a subprogram. It is regarded as a one-level subprogram call. Thus subprogram calls can be rested up to two levels as shown.

A single call command can repeatedly call a subprogram up to 999 times. For compatability with automatic programming systems, in the first block Nxxxx can be used instead of a subprogram number that follows O (or) A sequence number after N is registered as a subprogram number.

Cutter radius compensation (CRC)

Objectives: At the end of this lesson you shall be able to
- **state the significance of the term CRC**
- **explain the use of codes G42, G41 and G40.**

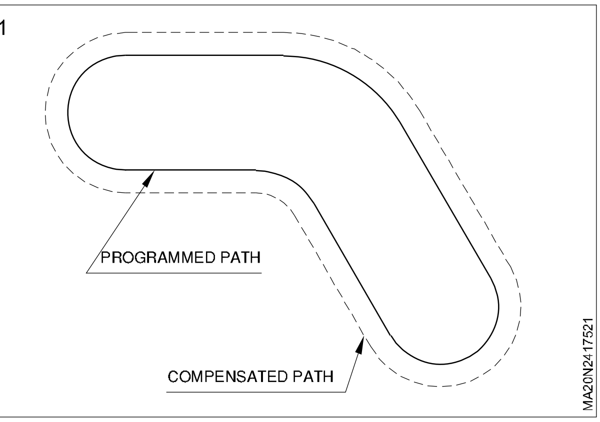
Usually the tool reference point being at the centre of the cutter, it will be the centre of the cutter that moves along the programmed path. But in a profiling operation, it is the periphery of the cutter that should travel along the programmed path and it is possible only if the centre of the cutter offsets itself from the programmed path by a distance equal to the cutter radius.

The term CRC signifies that the centre of the cutter should take a path away from the programmed path by a distance equal to the radius of the cutter. Such deviated path taken by the cutter centre is called compensated path. (Fig 1)

CRC commands (G41 and G42)

These commands instruct the control that the cutter centre should take a compensated path. These commands can be effective only if the control knows the radius of the cutter. For this purpose the radius of the cutter is stored in the tool data memory along with the length of the cutter. In the programme tool data memory is referred to by the D command.

Fig 1



In motion programme the command G41 or G42 is put in the block that takes the tool to the starting point of the profile. (Fig 2)

In Fig 2, the starting point of profile is marked 'S' and its X, Y coordinates are 40 and 40.

Difference between G41 and G42: If we write a block as 'G00 x 40 Y40' without either G41 and G42, the tool takes the path TS (Fig 2a) and the cutter will reach the point S, so that its centre coincides with S.

But if we add G41 in the block and write the block as G00 G41 x 40 Y40, the tool will take the path TK (Fig 2b) in such a way that the tool is on the left side of the profile with its periphery touching the point 'S'. Here TK is the compensated path.

If we write the block with G42 in place of G41, (G00 G42 X60 Y40) the compensated path will be TM such that the tool will be on the right side of the profile. (Fig 2C)

It may then be concluded that code G41 makes the compensated path to the left side while G42 makes the compensated path to the right side of the programmed path. The left/right has its reference to the direction of the cutter path. (See arrows in Fig 2a,b & c).

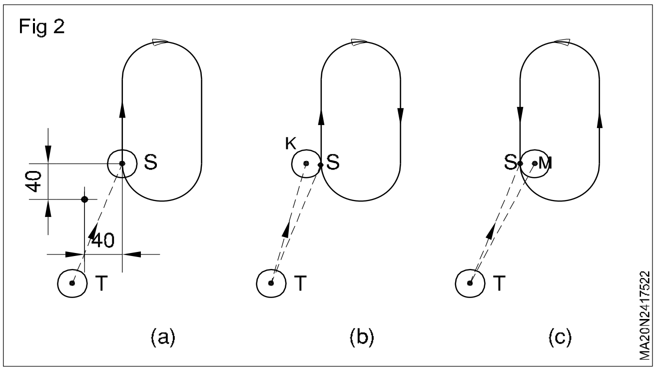


Fig 2

Cancellation of CRC G40

Once the command G41 or G42 is used the cutter will continue to take a compensated path. But when we want the cutter centre to revert back to the original programmed path the effect of G41/G42 has to be cancelled and this is done by the command G40.

Cutting tool materials

Objective: At the end of this lesson you shall be able to
- **state the different types of milling cutter tool materials, their properties and uses.**

The common materials used for the manufacturing of cutting tools are

- Carbon steel
- High speed steel
- Sintered carbide
- Ceramics
- Industrial diamonds

Carbon steel: This is used to produce cutters which are used for machining non-ferrous materials. These are cheap and are also of short life. While using these cutters, a cutting fluid must be used and low cutting speeds and feed rates should be maintained.

High speed steel: The high speed steel cutters are the most widely used cutters in general shop work. The composition of high speed steel may vary. The wear, shock resistance and hardness characteristics of these alloys make the cutters suitable for use in high speed milling. Standard high speed cutters have 18% tungsten, 4% chromium and 1% vanadium.

Sintered carbides: Tungsten carbide, cobait and tantalum carbide are used for producing these cutters, which are known as sintered carbide cutters. To maintain a wide range of hardness, toughness and wear resistance, the composition may be varied. These tools are expensive. They are used only as cutting tips or as edges fitted to low grade steel cutter bodies. The tips are brazed or clamped on the cutter body. They are used where a high rate of production is desired.

Ceramics: Ceramcis are produced in the form of inserts which are fitted to the cutter body. These cutters bear high wear resistance but are very brittle. These are used for light, high speed finishig cuts. These cutters are made from aluminium, silicon or magnesium oxides.

Sialon (Si- Al - O - N): The silicon nitride (Si3H3) based materials with aluminium and oxygen additions known as sialon which is used as a tool material. This is produced by melting together SI3N4, aluminium nitride, alumina and tetria, dried, pressed to shape and sintered at a temperature of about 1800°C.

This is considerably tougher than alumina and therefore can be successfully used during machining involving interrupted cuts. Two to three times more cutting speeds are used than the carbides tools.

Diamond: Diamond is extremely hard material and show large resistance to abrasion. It possesses high heat conductivity and melting point. However, oxidation of diamond starts at about 450°C and thereafter it can even crack. So, this reason the diamond tool is kept flooded by the coolant during cutting and light feeds are used. It offers highest tool life and mirror like finish by direct turning.

Diamond tools are used in production of rocket motors, electronic components made of ceramics etc.

Boron nitride: It is the hardest material next only to diamond. It is polymorphic in nature and can occur in three structural forms, namely.

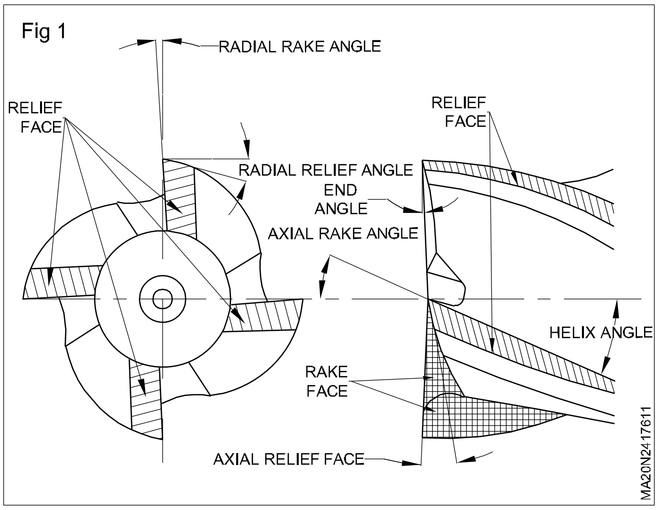
- Hexagonal graphite - like structures.
- Ultra - hard hexagonal wurtzite structure and
- Ultra - hard cubic structure.

Cutting tool geometry for milling operations

Objectives: At the end of this lesson you shall be able to
- **describe the HSS cutting tool geometry for milling operations**
- **explain the insert type tool geometry for milling operations**
- **explain the method of replacing shim and inserts.**

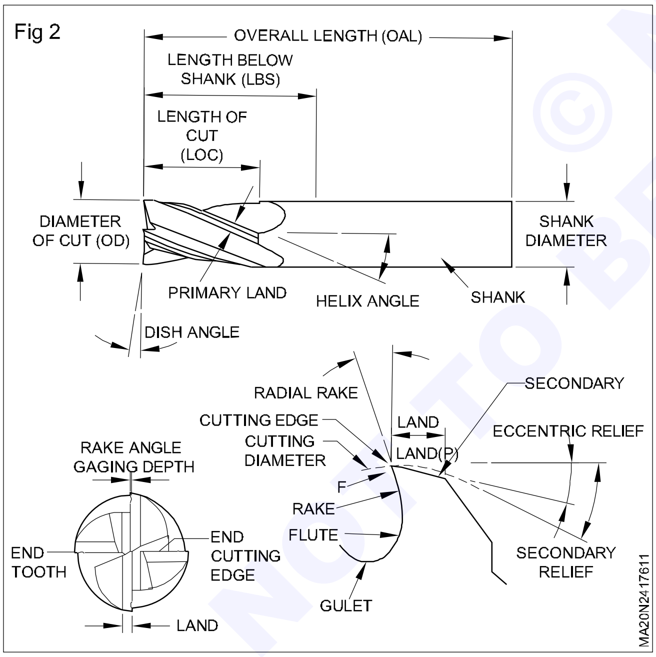
Face mill and its geometry (Fig 1)

Fig 1



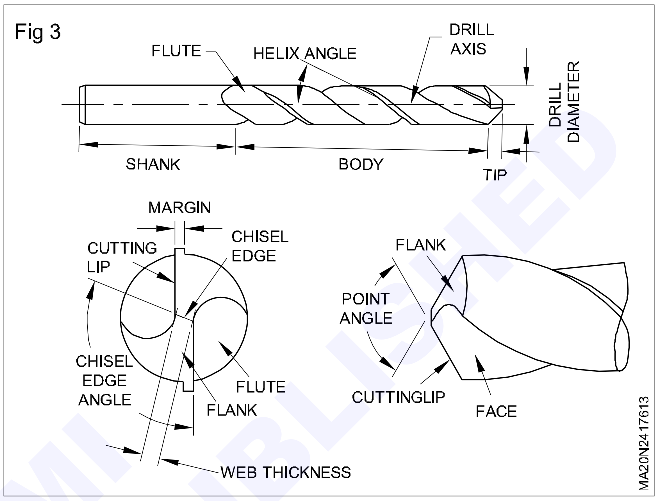
End mill and its tool geometry (Fig 2)

Fig 2



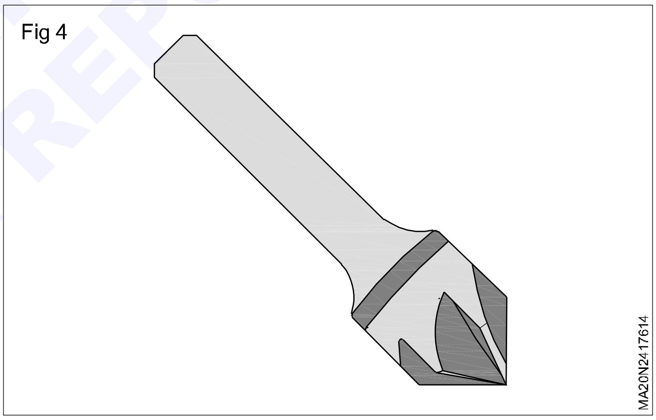
Drill and its geometry (Fig 3)

Fig 3



Countersink and its tool geometry (Fig 4)

Fig 4

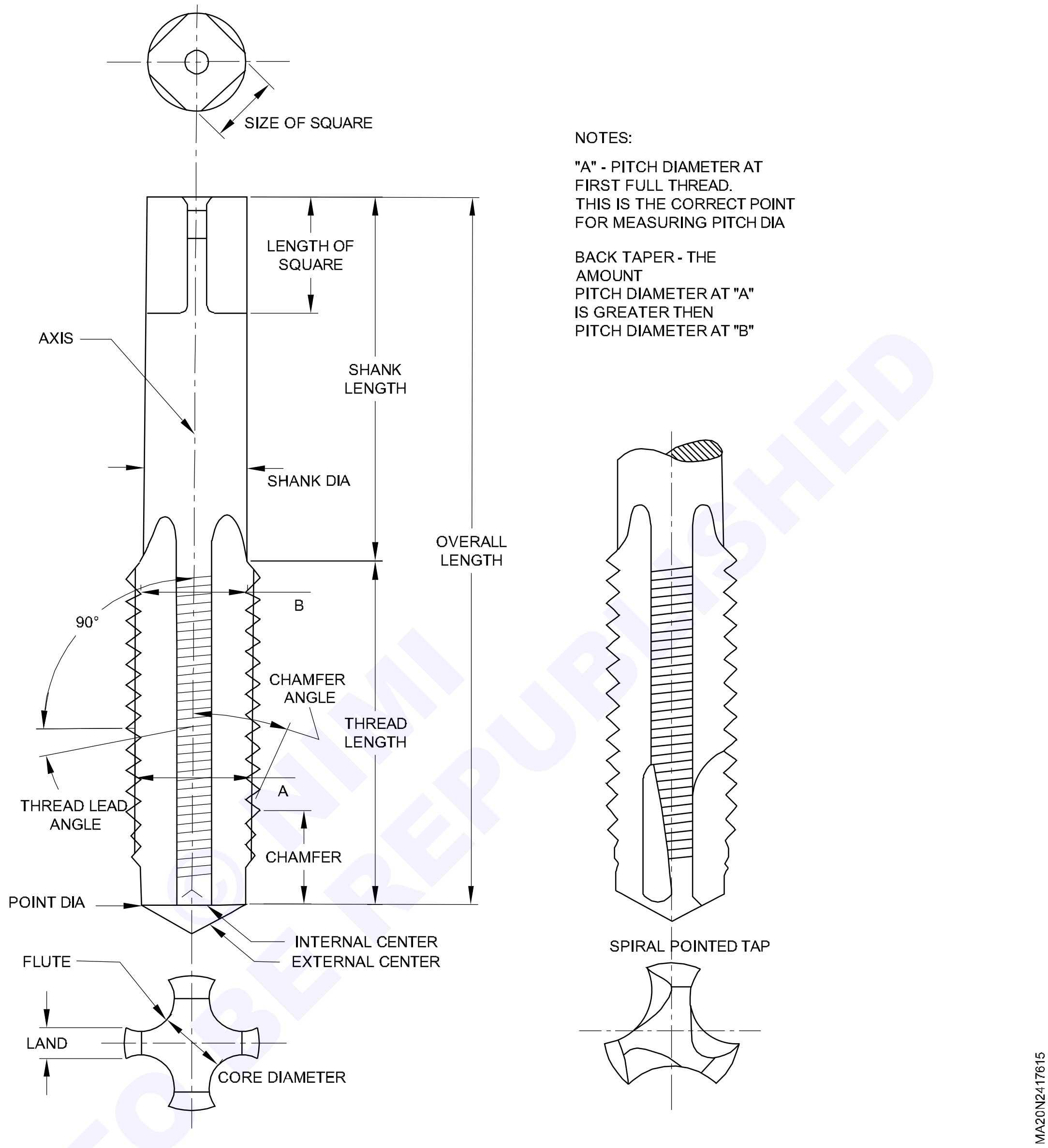


Engagement angle: Countersinks are fashioned to a wide range of engagement angles (60°, 82° 90°, 100°, 110°,120°, 130°, 140°, etc), and to either metric or english standards.

Body diameter: Diameter of counter body above the angle

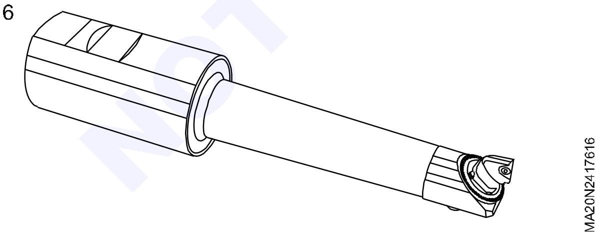
Tap and its geometry (Fig 5)

Fig 5



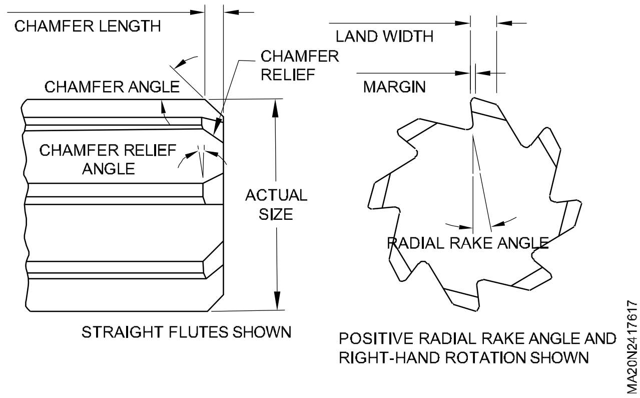
Finish boring tool (Fig 6)

Fig 6



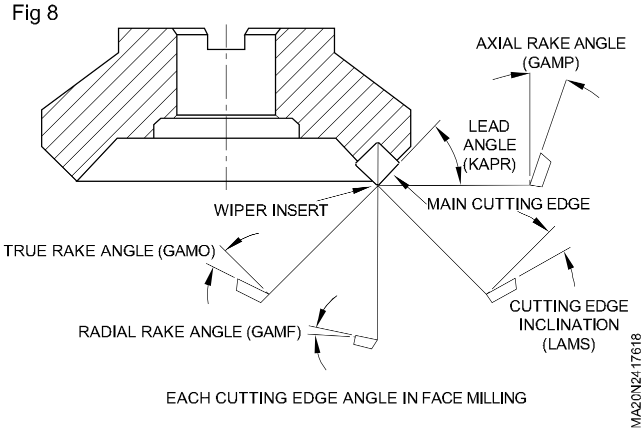
Reamer and its geometry (Fig 7)

Fig 7



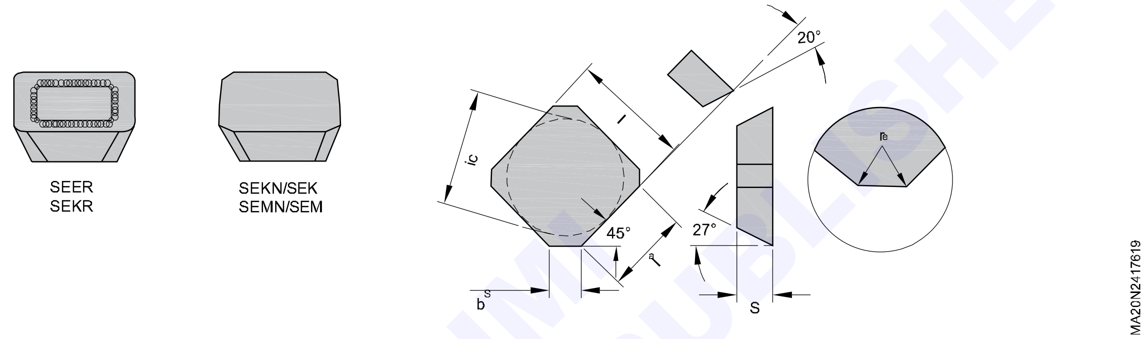
Carbide insert (Fig 8)

Fig 8



Insert for milling (Fig 9)

Fig 9



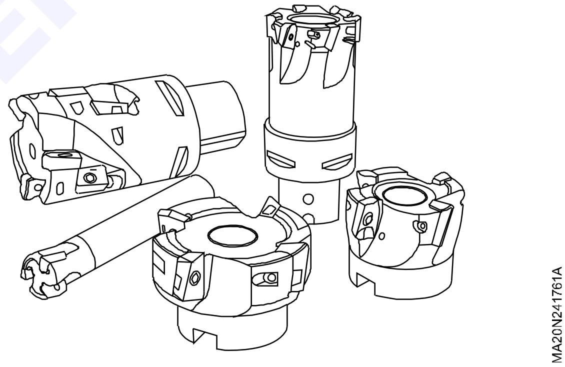
Square shoulder mill or end mill geometry (Figs 10)

Square shoulder milling cutter

Insert type drill and its geometry

Soild carbide drill

Fig 10



Cutting speed, feed and depth of cut

Objectives: At the end of this lesson you shall be able to
- **state about cutting speed**
- **calculate the spindle speed**
- **calculate the feed rate in mm/minute**
- **explain the spindle speed depending upon the diameter of the cutter, and the material to be cut.**

Cutting speeds, feeds and depth of cut

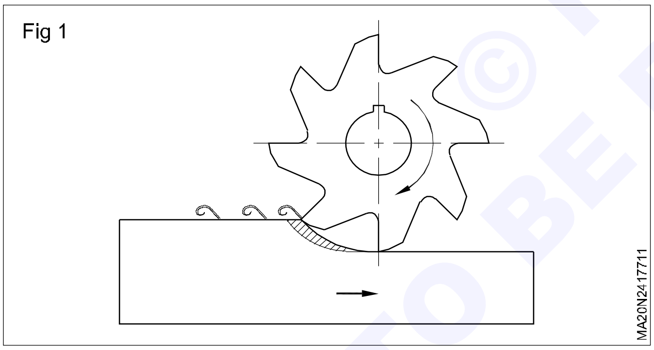
The efficiency of a milling operation depends upon the cutting speed, feed, and depth of cut.

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. Some where 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 evantually 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 important in any milling operation.

Cutting speed (Fig 1)

Fig 1



The cutting speed for a milling cutter is the speed at which the cutting edge or tooth cuts into the workpiece.

It is expressed in metres per minute.

The following important factors must be considered when determining the proper revolutions per minute.

- 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 harness, 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 cutter are shown in table 1.

Calculation

$$\text{Cuttingspeed}(V) = \frac{\pi DN}{1000} \text{m/min}$$

$$N(\text{rpm}) = \frac{V \text{ x } 1000}{3.1416 \text{ x } D}$$

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 π (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.

$$N\,(\text{rpm}) = \frac{V\,(m) \text{ x } 320}{D\,(mm)}$$

where 'D' is diameter of the cutter.

Example

Calculate the revolution per minute required for 75 mm high speed steel cutter when cutting machine steel.

(V = 30 m/min.)

$$\text{rpm} = \frac{30\text{x}320}{75} = \frac{9600}{75} = 128$$

From the chart1, the intersection of 75 mm and cutter speed of 30 m/min. is in between 115 and 140 rpm.This can be taken as 128 rpm 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 efficency of a milling operation are the milling Feed, of 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 (mm/ min.)

Feed rate is specified in mm/min.

The feed is expressed in milling machines by the following three different methods: feed per tooth, feed per rotation feed per minute and depth of cut.

Feed per tooth (Fig 2 & 3)

Feed per tooth is defined by the distance the work has advanced and the time between the engagement of two successive teeth. It is expressed in mm/tooth of the cutter.

Fig 2

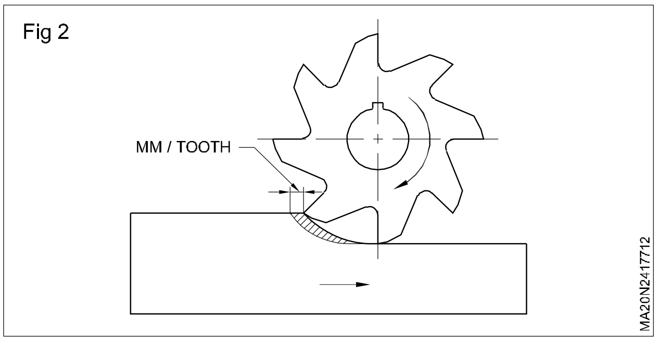
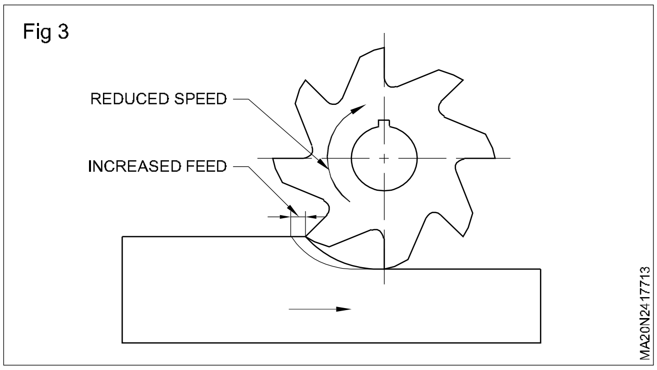


Fig 3



Feed per minute (Fig 4 & 5)

Fig 4

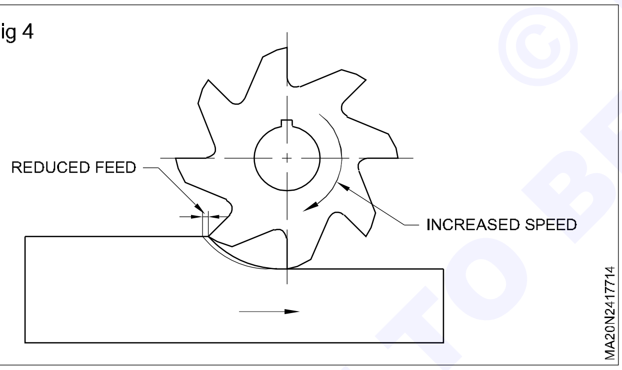
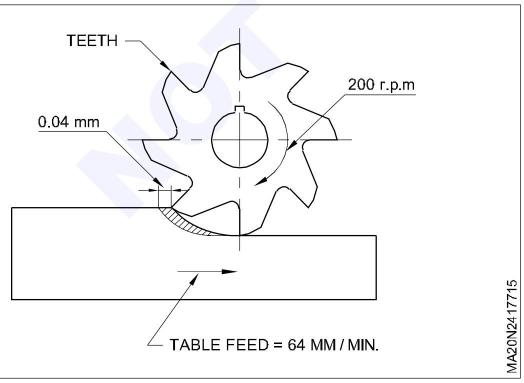


Fig 5



Feed per minute is defined as the distance the work has advanced 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 reduced when feed is increased.
- Cutting speed should be increased when feed is reduced.

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. However, the cutting speed should not be reduced too much as the cutter would then be operating under very heavy cutting forces.

For finish milling, the quality of the surface finish is, of course, important. Therefore, a light feed and a high cutting speed are used. (Fig 6 & 7)

Fig 6

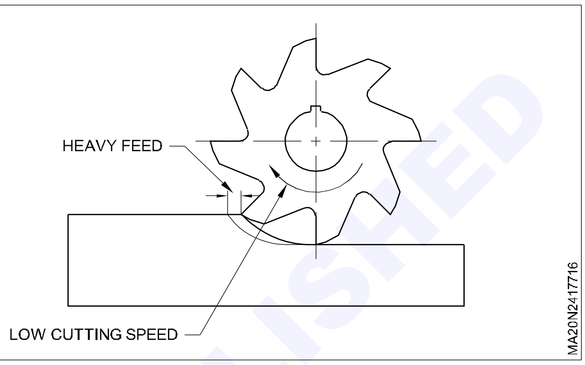
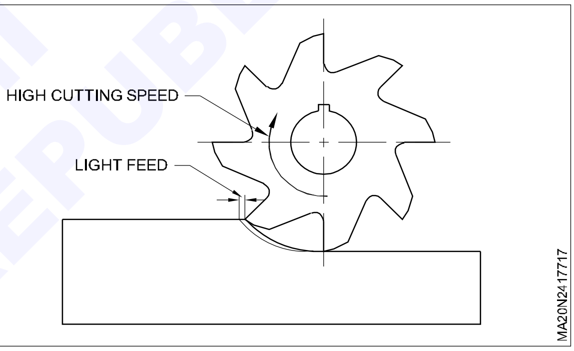


Fig 7



Feed per minute is calculated as follows

Feed in mm/minue = feed per teeth x number tooth x RPM

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 measured between the original and the final surface of the workpiece.

Where a smooth and accurate finish is needed, it is good practice to take roughing and finishing cuts. Roughing cuts should be deep with a feed as heavy as the work and the machine will permit with low cutting speed. 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.

Finishing cuts should be light with a fewer and finer feed than it is used in roughing cuts. The depth of cut should be at least 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. (Fig 8, 9 & 10)

The table 1 shows the cutting speed (V) in miunte (m/min) for various materials, using High Speed Steel (HSS) milling cutters of various types. They must be considered as average values which may vary according to actual working conditions.

Having selected a suitable cutting speed from the table1 this must be coverted into revolutions per minute (rpm) of the machine spindle. This can be done using the chart-1.

Fig 8

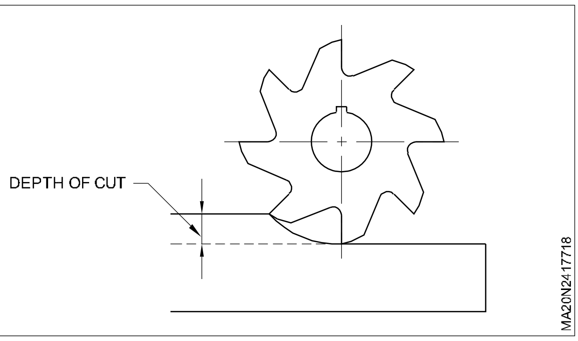


Fig 9

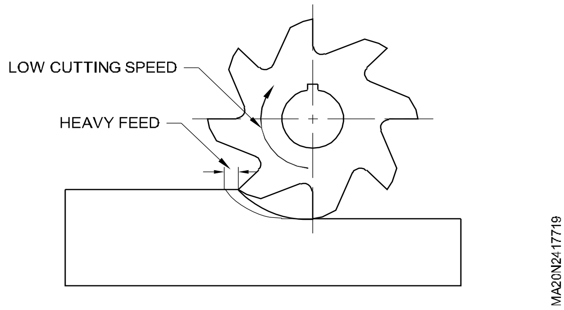


Fig 10

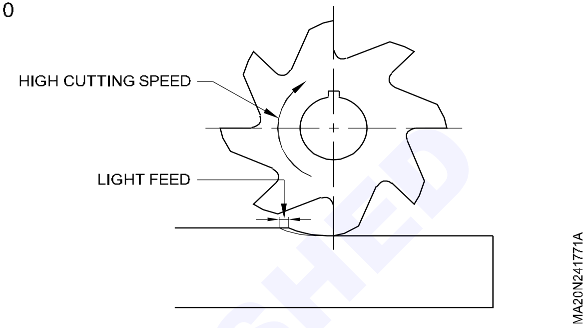


Table 1

Table for selecting cutting speeds for High Speed Steel milling cutters								
Material to be machined	BHN hardness	Shell end mill	End mill	S & F cutter	Cylind cutter	Slot'g cutter	Form cutter	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	18-25
Hard brass &hard bronze	-	40-60	40-60	30-45	30-45	90-90	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

(The chart has rpm values corresponding to the speeds available on a typical milling machine.)

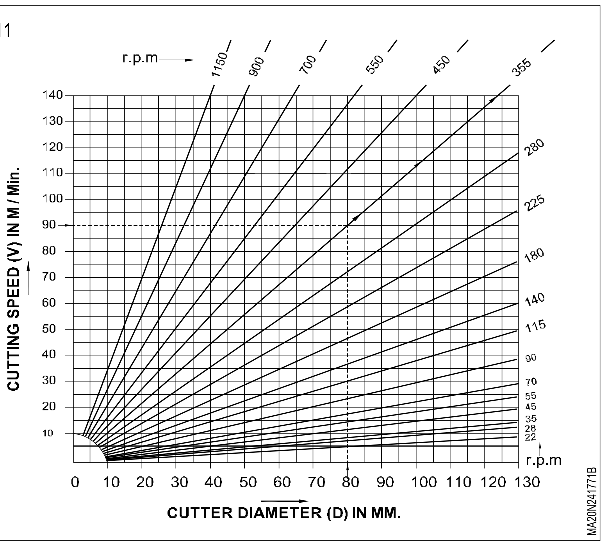
Example

A cutter of 80mm diameter is to cut at 90 M/min. What is the spindle speed to be selected? The two broken lines ----- on the chart shows that the spindle speed should be 355 rpm

Having selected a suitable cutting speed from the table1 this must be coverted into revolutions per minute (rpm) of the machine spindle. This can be done using the chart-1.

(The chart has rpm values corresponding to the speeds available on a typical milling machine.) Fig 11

Fig 11



Tool wear, tool life, relative effect of each cutting parameter on tool life

Objectives: At the end of this lesson you shall be able to
- **brief the criteria for tool wear**
- **describe the rate of tool wear**
- **classify the tool wear**
- **brief the factors influencing the tool wear**
- **state the tool life and factors affecting the tool life**
- **brief the relative effect of each cutting parameter.**

Tool wear is gradual process; created due to:

1 High localized stresses at the tip of the tool

2 High temperatures (especially along rake face)

3 Sliding of the chip along the rake face

4 Sliding of the tool along the newly cut work piece surface

The rate of tool wear depends on

- Tool and work piece materials
- Tool geometry
- Process parameters
- Cutting fluids
- Characteristics of the machine tool

Tool wears are classified as

a Flank and notch wear (Fig 1)

b Crater wear (Fig 2)

c Built up edge (Fig 3)

d Frittering (Fig 4)

e Thermal cracks (Fig 5)

f Insert breakage (Fig 6)

g Plastic deformation (Fig 7)

Factors influencing crater wear are

1 The temperature at the tool-chip interface.

2 The chemical affinity between the tool and work piece materials

•Diffusion rate increases with increasing temperature, crater wear increases as temperature increases

Other Types of Wear, Chipping, and Fracture

- Nose wear is the rounding of a sharp tool due to mechanical and thermal effects
- It dulls the tool, affects chip formation and causes rubbing of the tool over the work piece
- Tools also may undergo plastic deformation because of temperature rises in the cutting zone
- Tools may undergo chipping, where small fragment from the cutting edge of the tool breaks away

Chipping may occur in a region of the tool where a small crack already exists

Two main causes of chipping: Mechanical shock & Thermal fatigue

The tool life can be expressed in different ways

1 Actual cutting time to failure.

2 Length of work cut to failure.

3 Volume of material removed to failure.

4 Number of components produced to failure.

5 Cutting speed for a given time of failure.

Factors affecting tool life:

1 Material of machined work piece.

2 Required surface quality of the work piece.

3 Tool material.

4 Tool geometry and sharpening condition.

5 Fixation of tool and work piece.

6 Machining variables such as, speed, feed, and depth of cut.

7 Type of coolant used. 8. Condition of cutting tool with respect to vibrations.

Tool life criterion in production:

The following are some alternates that are more convenient to use in production:

1 Changes in the sound emitting from operation.

2 Degradation of the surface finish on work.

3 Complete failure of cutting edge.

4 Work piece count.

5 Chips become ribbon form or string

Relative effect of each cutting parameter

1 The cutting speed has an inverse relation with the tool life.

2 The feed rate has inverse relation with the tool life.

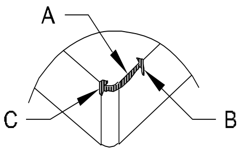
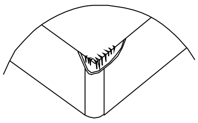
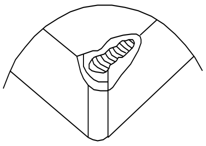
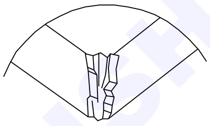
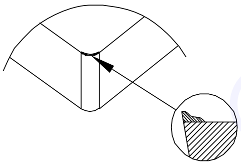
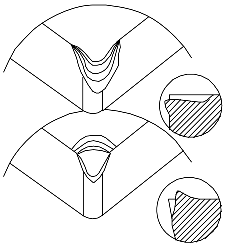
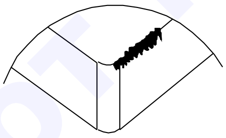
3 The depth of cut has an inverse relation with the tool life.

4 The spindle speed has an inverse relation with the tool life.

5 Various coatings on the tool increase their life.

Thus, by setting the cutting parameters in a proper proportion, the cutting tool life can be optimized.

Classification of tool wear types

Flank and notch wear	**Thermal cracks**
 a Rapid flank wear causing poor surface finish or out of tolerance. b,c Notch wear causing poor surface finish and risk of edge breakage	 Small cracks perpendicular to the cutting edge causing frittering and poor surface edge.
Cratering  Causing a weakened edge	**Insert breakage**  Damage not only the insert by also the shim and workpiece.
Build up edge (B.U.E)  Causing poor surface finish and cutting edge frittering when the B.U.E. is torn away	**Plastic deformation**  a- edge deformation b-flank impression Leading to poor chip control and poor surface finish. Risk of excessive flank wear leading to insert breakage.
Frittering  Small cutting edge fracture causing poor surface finish and excessive flank wear.	

If problems should occur

A few solutions for overcoming the most common problems during

Milling

	Remedies and solutions								
Problems	Reduce the cutting speed	Increase the cutting speed	Reduce the feed/tooth	Increase the feed/tooth	Select a more wear resistant grade	Select a tougher grade	Use a coarse pitch cutter	Change the cutter position	Do not use coolant
Flank wear	X			X	X				
Notch wear	X			X	X				X
Cratering	X				X				
Plastic deformation	X		X		X				
Built-up-edge (BUE)		X		X					X
Small cracks perpendicular to the cutting edge	X					X			X
Small cutting edge fracture (frittering)		X				X			X
Insert breakage			X			X		X	
Vibration				X			X	X	
Bed surface finish		X	X		X				

Guide lines for the Selection of Milling Cutter / Inserts from Catalogue

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

- **select the insert depending upon the operations**
- **select the insert depending upon the material to be machined.**

Step 1: Define the type of material to be milled

"P" for steel low alloy steel

"M" for stainless steel

"K" for cast iron

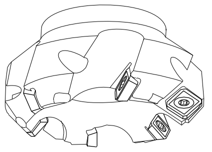
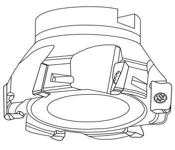
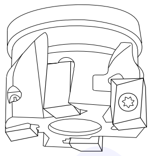
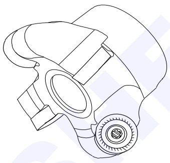
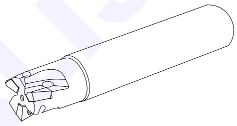
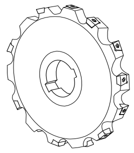
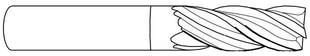
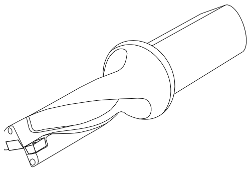
Step 2: Define the milling operation and accordingly select the milling cutter from table 1

Use close pitch for general purpose milling and mixed operation

- use coarse pitch for long over overhang and un stable conditions
- use an extra close pitch for short chipping material, heat resisting material, and a small radial depth of cut

Table - 1

Milling Cutters

Sl.No	Name of mIlling Cutter	Fig.
1	Face Milling Cutter	
2	Shoulder MIlling Cutter	 
3	Profile Milling Cutter	
4	End Mills Cutter	
5	Slot Milling Cutter	
6	Soild Carbide end mills	
7	Short Hole Drills	

Step 3: Define the application of machining

F Fine

M Medium

R Rough

Step 4: Good Condition

Continuous cuts

High cutting speeds

Pre machined or light forged skin

Normal condition

General purpose conditions

Diffcult conditions

Interrupted cuts

Low cutting speed

Heavy forging scale

Step 5: Chose your inserts and note down the spindle speed depth of cut and feed per tooth

- Clamp the selected insert in the milling cutter holder
- Mount the cutter in milling arbor
- Set the spindle speed feed and depth of cut

Preparing part program as per drawing and checking with CNC simulator

Objectives: At the end of this lesson you shall be able to
- **write the part program as per drawing**
- **enter the program in simulator**
- **verify the program on CNC simulator.**

Steps for preparing manual CNC part programing

1 Read the part drawing

2 Decide which machine will produce the part

3 Choose the tooling requirement

4 Decide the machining sequence

5 Do the calculation to find part co-ordinates

6 Calculate the spindle speeds and feed rates required for the tooling and part material

7 Write the CNC program depending on the control

8 Prepare setup sheets and tool lists

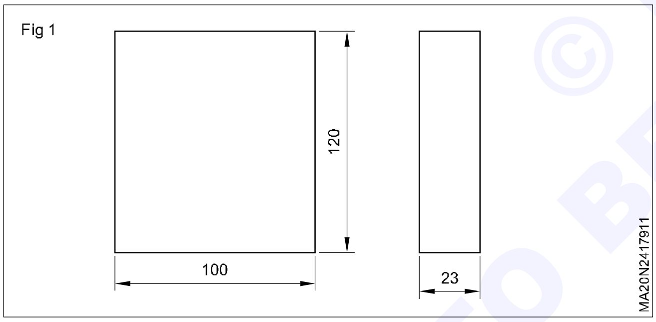
9 Verify the program using a CNC simulator or the actual machine

10 Edit the program if necessary

11 Run the program and produce the part.

Part drawing (Fig 1)

Fig 1



Premachined part 100x100x125mm

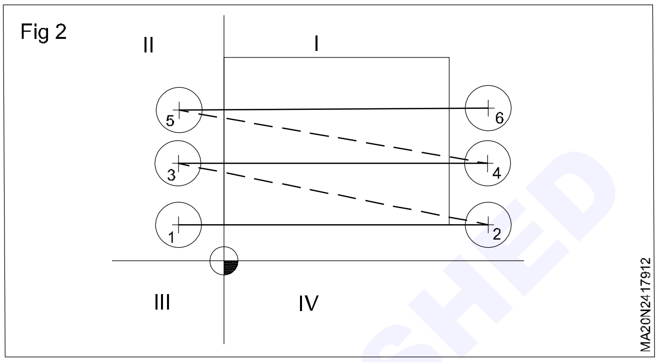
Material - Aluminium.

Operation to be carried out only face milling.

Cutter = Face mill ϕ 50.

Fix the work zero and decide the tool path (Fig 2).

Fig 2



Tool path with CNC code

1 G00 X 20 Y-30 Z50 - Position
G00 Z00;

2 G0I X 20 Y 130 F 100;
G00 Z5;

3 G00 X 65 Y-30;
G00 Z0;

4 G01 X 65 Y130;
G00 Z5;

5 G00 X 110 Y-30;

6 G00 Z00;

7 G01 X110 Y 130;

8 G00 Z50

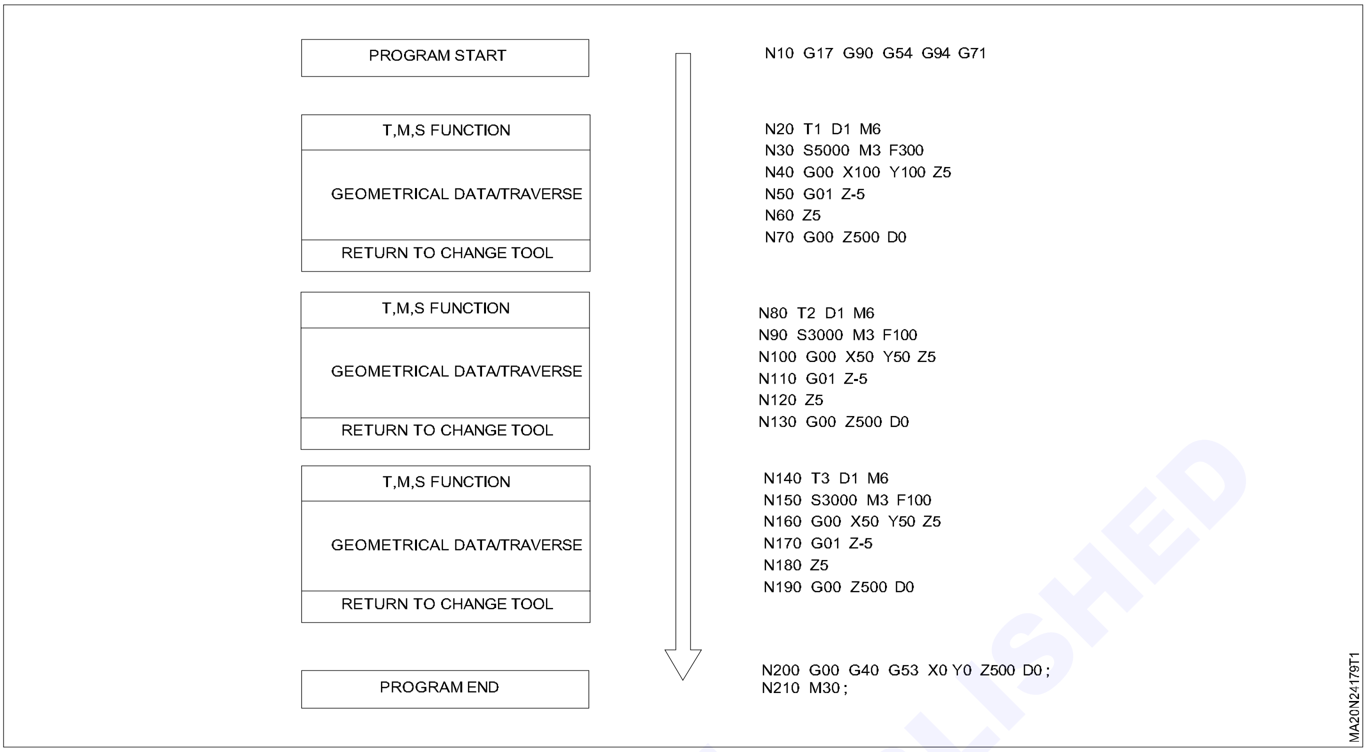
After writing the tool path

Write the program depending up on the control.

Generally to the tool path with G00, G0; G02, G03 and the codes will remain same for the most of the control.

Only the structure of the program will be different.

Standard program structur for siemens control



```
N10  G17  G90  G54  G94  G71

N20  T1  D1  M6
N30  S5000  M3  F300
N40  G00  X100  Y100  Z5
N50  G01  Z-5
N60  Z5
N70  G00  Z500  D0

N80  T2  D1  M6
N90  S3000  M3  F100
N100  G00  X50  Y50  Z5
N110  G01  Z-5
N120  Z5
N130  G00  Z500  D0

N140  T3  D1  M6
N150  S3000  M3  F100
N160  G00  X50  Y50  Z5
N170  G01  Z-5
N180  Z5
N190  G00  Z500  D0

N200  G00  G40  G53  X0 Y0  Z500  D0 ;
N210  M30 ;
```

For

Fanuc control

First cancelation code

Preparation code like (G90, G21, G94)

Tool change command

Spindle rotation

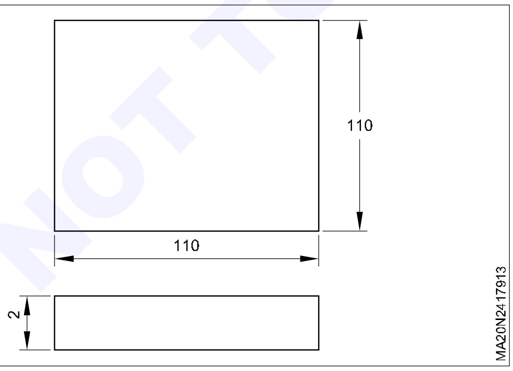
Positioning the tool for operation

Actual tool path as per drawing

Lifting the tool the safe position. Putting off the spindle coolant. Sending the tool to home position ending the program.

Example (Fanuc program) (Fig 3)

Fig 3



Face Milling

Cutter dia 50   Workpiece 110x100

```
O0001:
N5 G40 G49 G50 G80 G69;
N10 G90 G21 G94;
N15 T06 M19;
N20 S600 M03;
N25 G00 M03;
N25 G00 G55 G43 H06 X20 Y-30 Z50;
N30 G00 Z0 F100 M07;
N35 G01 X20 Y 130;
N40 G00 Z5;
M40 G00 X65 Y-30;
N45 G00 Z0;
N50 G01 X65 Y130;
N55 G00 Z5;
M60 G00 x 110 Y-30;
N65 G00 Z0;
N70 G01 X110 Y 130;
N75 G00 Z50 M05;
N80 G91 G28 X0 Y0 Z0 M09;
N85 M30;
```

Program structure for Hase control	
%	:Program must begin and end with %
O01010	:Program number letter "O" with four digit number
(Mill part program example)	:(Comments in parenthsis are ignored by control)
N1 (Drill 4 places)	:(Comments in parenthesis are ignored by control)
T1 M06 (1½ IN. DIA. STUB DRILL)	:Tool change to tool #1, (Notes to operator)
G90 G54 G00 X-1,5 Y1.5 S1400 M03	:ABS posit, work offset#, Rapid X Y, Spindle on CW
G43 H01 Z1. M08	:Tool length comp #1, Z position, coolant ON
G73 G99 Z-0, 625 Q0.2R0.1 F5.	:High speed peck drilling, Drill Z-325 Deep, 2 Peck
Y-1.5	:Drill Another Hole Rapid Plane is AT R.1
/ X1.5	:Drill third hole with an optional Block delete
/ Y1.5	:Drill fourth hole with an Optional block delete
G80 G00 Z1. M09	:Cancel Canned cycle, Rapid Z1. Coolant off
G28 G91 Z0 M05	:Return Z to Machine zero, Spindle off
N2 (Countersink 4 Places)	:(Comments in parenthesis are ignored by control)
T2 M06 (5/8 Dia, 90 Deg, C'Sink)	:Tool change to tool #2, (Notes to operator)
G90 G54 G00 X-1.5 Y1.5 S900 M03	:ABS posit, work offset #, Rapid to posit, spindle ON CW
G43 H02 Z1.M08	:Tool length Comp #2, Z position, Coolant ON
G82 G99 Z-0.27 P0.5R0.1 F12.	:Spot Drill cycle to Z-27 Deep, Dwell 5 Second
Y-1.5	:Second hole, rapid plane is at R-1
/ X1.5	:Third hole with an optional block delete
/ Y1.5	:Fourth hole with an optional block delete
G80 G00 Z1.M09	:Cancel canned cycle, rapid Z1., coolant OFF
G28 G91 Z0 M05	:Return Z to machine Zero, spindle OFF
N3 (Set D3 Dia. Offset value to 500)	:(Comments in parenthesis are ignored by control)
T3 M06 (1½ Dia End Mill)	:Tool changes to tool #3 (Notes to operator)
G90 G54 G00 X-2.3 Y2.3 S1100 M03	:ABS posit, work OFFSET#3, Z position, coolant ON
G43 H03 Z0.1 M08	:Tool length compensation
G01 Z-0.625 F30.	
G41 Y2. D03 F11. X2.0 Y-2.0 X-2.0 Y2.25	:Compensate cutter left of line :Cut A 2.0 in. square CW with tool dia comp D03.
G40 X-2.3 Y2.3	:G40 Cancels comp white positionning away from part
G00 Z1.M09	:Rapid Z1., Coolant OFF
G28 G91 Y0 Z0 M05	:Return Y and Z to machine zero
T1 M05	:Tool change back to tool #1
M30	:Program stop and rewind to begining
%	:Program must begin and end with %

To change tools, all that is needed is an M06 even without a G28 in the previous line. A G28 can be specified to send all axes ti machine home, or it can be defined to send a specific axis home with G28 G91 Z0 and/or Y0 and/or X0 to send just these axis specified to home position.

Methods for creating part programs (Siemens)

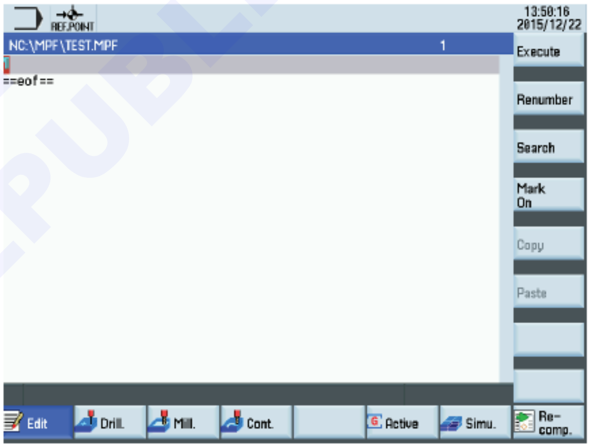
You can create a part program with one of the following methods:

- Creating on a computer and transferring it to the PPU via USB interface.
- Creating on a computer and transferring it to the PPU via Ethernet interface.
- Creating directly on the PPU (see below for details).

Creating a part program on the PPU

Sl. No	Image	
1	PROGRAM MANAGER	Select the program management operating area.
2	NC	Press this softkey to enter the system directory for storing part programs.
3	New	To directly create a new program file, press this softkey and go to Step 4. To create a new program directory first, press this softkey and proceed as follows before you go to Step 4:
4	New directory OK ▲ ▼ INPUT	A Press this softkey to open the window for creating a new directory.  New directory Type: Directory DIR Name: B Enter a desired name for the new directory. Note that some special characters (see table at the end of this section) are invalid for the directory name. C Press this softkey to confirm your entry. D Select the new directory with the cursor keys. E Press this key on the PPU to open the directory.
4	New file	Press this softkey to open the window for creating a new program.  New G code program Type: Main program MPF Name:

Sl. No	Image	
5		Enter the name of the new program. You can enter the file name extension ".MPF" (main program) or ".SPF" (subprogram) to define the program type. The control system identifies a program as a main program if you do not enter any file name extension. The character length of a program name is limited to 24 English characters or 12 Chinese characters. Note that some special characters (see table at the end of this section) are invalid for the program name.
6		Press this softkey to confirm your entry. The part program editor window opens automatically. Now you can edit the program text in the window. The control system saves your editing automatically.



13:50:16
2015/12/22

NC:\MPF\TEST.MPF 1

==eof==

Special characters invalid for program or directory names

The control system does not support the use of the following special characters in program or directory names:

/	<	>	?	:	#	(	)	[	]	$	!	-
+	^	\	*	|	;	&	%	@	=	~	`	Space

Editing a part program

Methods for editing part programs

You can edit a part program with one of the following methods:

- Editing on a computer and transferring it to the PPU via USB interface.
- Editing on a computer and transferring it to the PPU via Ethernet interface.
- Editing directly on the PPU (see below for details)

Editing a part program on the PPU

You can edit a part program only when it is not being executed.

Note that any modification to the part program in the program editor window is stored immediately.

Note:

Steps 1 to 4: Search for a program file

Steps 5 to 9: Edit the selected program in the open program editor window

1 Select the program management operating area. PROGRAM MANAGER

2 Press this softkey to enter the system directory for storing part programs. NC

3 Select the desired program file/directory in one of the following methods:

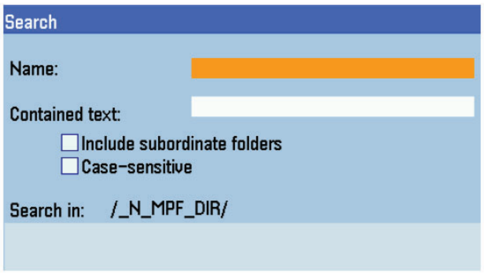
- Navigate to the program/directory with the cursor keys ▲ ▼
- Open the search dialog box and enter the desired search term.

Search

Note:

If you search for a program, the file name extension must be entered in the first input field of the dialog box below (Fig 4).

Fig 4



Search

Name:

Contained text:

☐ Include subordinate folders

☐ Case-sensitive

Search in: /_N_MPF_DIR/

- On the PPU, press the alphabetic or numeric key that contains the first character of the desired program/directory name. The control system automatically highlights the first program/directory whose name starts with that character. If necessary, press the key continuously until you find the desired program/directory.

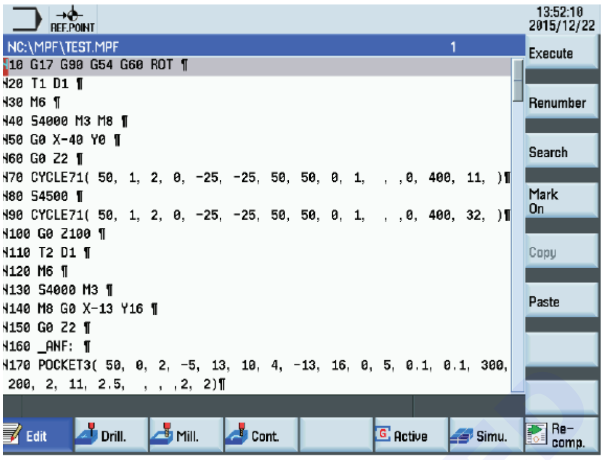
4 Press this key to open either the selected program in the program editor or the selected directory. In the latter case, perform Step 3 and then Step 4 until the selected program is opened in the program editor. INPUT

5 Edit the program text in the program editor window using the following keys on the PPU: (Fig 5)

Fig 5

6 When necessary, select the following vertical softkeys to complete more program editing operations. (Fig 6)

Fig 6



```
NC:\MPF\TEST.MPF                                    1
N10 G17 G90 G54 G60 ROT ¶
N20 T1 D1 ¶
N30 M6 ¶
N40 S4000 M3 M8 ¶
N50 G0 X-40 Y0 ¶
N60 G0 Z2 ¶
N70 CYCLE71( 50, 1, 2, 0, -25, -25, 50, 50, 0, 1,  ,  ,0, 400, 11, )¶
N80 S4500 ¶
N90 CYCLE71( 50, 1, 2, 0, -25, -25, 50, 50, 0, 1,  ,  ,0, 400, 32, )¶
N100 G0 Z100 ¶
N110 T2 D1 ¶
N120 M6 ¶
N130 S4000 M3 ¶
N140 M8 G0 X-13 Y16 ¶
N150 G0 Z2 ¶
N160 _ANF: ¶
N170 POCKET3( 50, 0, 2, -5, 13, 10, 4, -13, 16, 0, 5, 0.1, 0.1, 300,
 200, 2, 11, 2.5,  ,  ,  ,2, 2)¶
```

Renumbering program blocks Renumber

With this softkey, you can modify the block numbering (Nxx) of a program opened in the program editor window. After you press this softkey, the block number is inserted at the beginning of the program block in ascending order and is increased by an increment of 10 (for example, N10, N20, N30).

Searching in programs Search

Pressing this softkey opens the search dialog box. You can use the search function to quickly arrive at points where you would like to make changes, for example, in very large programs. You can search with specified text or line number by selecting the corresponding softkey.

Copying/deleting/pasting program blocks Mark On

A Press this softkey in the open program editor window to insert a marker.

B Select the desired program blocks with the cursor.

▲ ▼
◄ ►

C Press this softkey to copy the selection to the buffer memory. Copy

- OR -

Press this key to delete the selected program blocks and to copy them into the buffer memory. DEL

D Place the cursor on the desired insertion point in the program and press this softkey. The content of the buffer memory is pasted. Paste

7 If you want to program cycles, press the corresponding softkey to open the desired cycle programming window. For more information, see Section "Cycles.

Drill. Mill.

8 If you want to program contours, press this softkey to open the contour programming window.

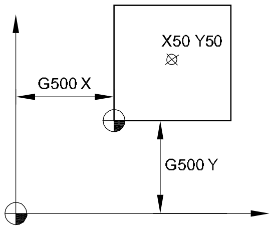
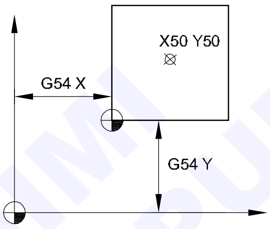
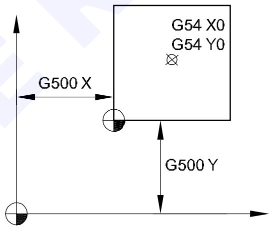
For more information, see Section "Free contour programming.

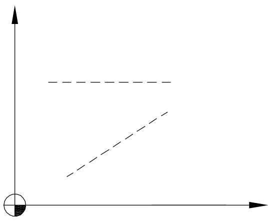
9 Press this key to return to the program management operating area after finishing editing.

Note: When you are in the main screen of the program management operating area, you can select to make the following operations between programs

- Searching for programs
- Copying/cutting/pasting programs
- Deleting/restoring programs
- Renaming programs

Definition of work offset (G54 to G59, G500, G90/G91)

Description	Illustration	Programming example
G500: All absolute path data corresponds to the current position. The position values are written in the G500 (basic) zero offset.		N10 G17 G90 G500 G71 N20 T1 D1 M6 N30 S5000 M3 G94 F300 N40 G00 X50 Y50 Z5 N50 G01 Z-20 N60 Z5 N70 G00 Z500 D0
G54 to G59: With G500 = 0, the offset for the workpiece can be stored in the workpiece offsets G54 to G59.		N10 G17 G90 G54 G71 N20 T1 D1 M6 N30 S5000 M3 G94 F300 N40 G00 X0 Y0 Z5 N50 G01 Z-20 N60 Z5 N70 G00 Z500 D0
G500 + G54: With G500 = 0 activated, the value in G500 is added to the value in G54.		N10 G17 G90 G500 G71 N20 T1 D1 M6 N30 S5000 M3 G94 F300 N40 G00 G54 X20 Y20 Z5 N50 G01 Z-20 N60 Z5 N70 G00 G53 Z500 D0
G90: With G90 (absolute positioning) at the program start, the geometrical data refers to the zero of the coordinate system currently active in the program, usually with G54, G500, or G500 + G54.		- N10 G17 G90 G54 G71 N20 T1 D1 M6 N30 S5000 M3 G94 F300 N40 G00 X100 Y100 Z5 N50 G01 Z-20 N60 Z5 N70 G00 Z500 D0
G91: With G91 (incremental positioning), you can add numerical value of path information (the incremental positioning with the current axis position as the start point) in the program. Subsequently, switch to absolute positioning with G90.		- N10 G17 G90 G54 G70 N20 T1 D1 M6 N30 S5000 M3 G94 F300 N40 G00 X3.93 Y3.93 Z0.196 N50 G01 G91 Z-0.787 N60 Z0.196 N70 G00 G90 Z19.68 D0

Rapid traverse (G00)		
G00: When G00 is active in the program, the axis will traverse at the maximum axis speed in a straight line.		N10 G17 G90 G54 G71 N20 T1 D1 M6 N30 S5000 M3 G94 F300 N40 G00 X50 Y50 Z5 N50 G01 Z-5 N60 Z5 N70 G00 Z500 D0

Tool and traverse (T, D, M6, F, G94/G95, S, M3/M4, G01)

Description	**Programming example**
T, D: A new tool can be selected with the "T" command, and the "D" command is used to activate the tool length offset. **M6:** M6 can be used for automatic tool change on the machine. **G94/G95, F:** The feedrate is defined with "F". G94 F defines feedrate in terms of time (mm/min) and G95 F defines feedrate in terms of spindle revolutions (mm/rev). **S, M3/M4:** The spindle speed is defined with "S". The spindle direction is defined with M3(clockwise) and M4 (counter-clockwise). **G01:** When G01 is active in the program, the axis traverses at the programmed feedrate (as defined by G94 F or G95 F) in a straight line.	N10 G17 G90 G54 G71 N20 T1 D1 M6 N30 S5000 M3 G94 F300 N40 G00 X50 Y50 Z5 N50 G01 Z-5 N60 Z5 N70 G00 Z500 D0.

Simultaneous recording

Simultaneous recording of a program run

Simultaneous recording before machining of the workpiece		
1	Auto	Load a program in the operating mode “AUTO”.
2	NC Pr cn	Press the HSK 1.4 “Prog. cntrl.“ and activate the checkboxes "PRT No axis motion" and "DRY Dry run feedrate" (see section 5.1.5 “Program control”) The program is executed without axis movement. The programmed feedrate is replaced by a dry run feedrate. -OR- Let the “DRY Dry run feedrate” box unchecked. Simultaneous recording is performed with the pro-grammed feedrate.
3	Simult. record.	Press the HSK 7 “Simultan. record“. The “Simultaneous recording” window opens.
4	Cycle Start	Press the “CYCLE START” key on the machine con-trol panel (MCP). The execution of the program on the machine is started and displayed graphically on the screen.
5	Cycle Start	Press “CYCLE STOP” to stop machining and the HSK 7 "Simultan. record" again to close the “Simultaneous recording” window.

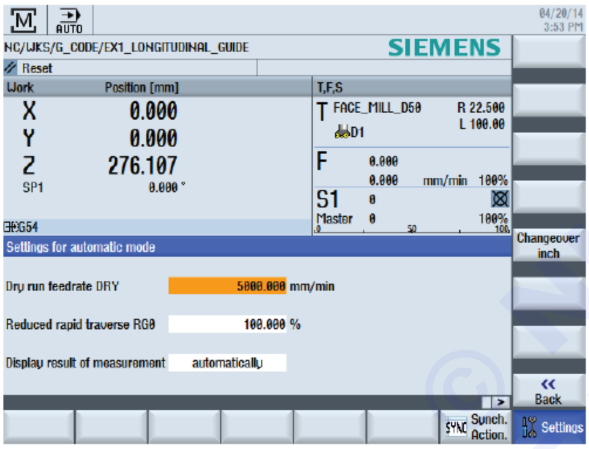
Simultaneous recording during machining of the workpiece

1	Auto	Load a program in the operating mode “AUTO”.
2	Simult. record.	Press the HSK 7 “Simultan. record“. The “Simultaneous recording” window opens.
3	Cycle Start	Press the “CYCLE START”-key on the machine con-trol panel (MCP). The machining of the workpiece is started and graphically displayed on the screen.
4	Cycle Stop Simult. record.	Press the “CYCLE STOP”-key and the HSK 7 "Simultan. record" again to stop the recording and to close the “Simultaneous recording” window.

Settings Settings

By pressing the HSK 2.8 “Settings” the following input mask with the settings for automatic mode is shown on the screen. (Fig 7)

Fig 7



Settings for automatic mode

Dry run feedrate DRY 5000.000 mm/min

Reduced rapid traverse RG0 100.000 %

Display result of measurement automatically

Vertical softkey bar (VSK)

Display area	Description
Changeover inch	By pressing the VSK 5 "Changeover inch" the measuring units are converted from the metric to the imperial (inch) dimension system. New values have to be entered in inches. By pressing this key the key function switches to "Changeover metric".
Changeover metric	By pressing the VSK 5 "Changeover metric" the measuring units are converted from the imperial (inch) to the metric dimension system. New values have to be metric. By pressing this key the key function switches to "Changeover inch". Accept the selection by pressing the VSK 8 “OK” or cancel by pressing the VSK 7 “Cancel”.
Back	By pressing the VSK 8 "Back" you switch back to the main screen of the Sinumerik Operate.

Parameters for "Settings for automatic mode"

In the “Settings for automatic mode“ window all configurations for automat-ic operation can be done.

Parameter	Unit	Meaning
Dry run feedrate DRY	[mm/ min]	The feedrate defined here replaces the pro-grammed feedrate during execution if you have selected „DRY dry run feedrate“ under program control.
Reduced rapid trav-erse RG0	%]	This value entered here reduces the rapid traverse to the entered percentage value if you have selected “RG0 reduced rapid trav-erse” under program control.
Display result of measurement		Using a MMC command, you can display measurement results in a part program:
Automatically		When the control reaches the command, it automatically jumps into the “Machine” oper-ating area and the window with the meas-urement results is displayed.
Manually		The window with the measurement results is opened by pressing the softkey “Measurement result”.

Process planning

Objectives: At the end of this lesson you shall be able to
- **define process planning**
- **list the function of the process planning**
- **explain the ability requirement of the planner**
- **steps involved in process planning**
- **brief the forms used in process planning**
- **study and interpret the sample process planning.**

Definition

Act of preparing detailed work instructions to produce a part

It's a function within the manufacturing facility

Establishes processes and parameters used to convert part from initial form to final form

Predetermined in an engineering drawing

Person who develops process plan: often called process planner

Functions included in process planning

Raw material preparation

Processes selection

Process sequencing

Machining parameter selection

Tool path planning

Machine selection

Fixture selection

Manual Process Planning

Process planner must have following knowledge

Ability to interpret an engineering drawing

Familiarity with manufacturing processes and practice

Familiarity with tooling and fixtures

Know what resources are available in the shop

Know how to use reference books (e.g. machinability data handbooks)

Ability to do computations on machining time and cost

Familiarity with raw materials

Necessary steps to prepare a process planning

1 Study overall shape of part, identify features and all critical dimensions

2 Thoroughly study the drawing; try to identify all manufacturing features and notes

3 Determine best raw material shape to use if raw stock not given

4 Identify datum surfaces; Use information on datum surfaces to determine the setups

5 Select machines for each setup.

6 Determine rough sequence of operations necessary to create all the features for each setup

Process Plan includes

Operation sheet

Route sheet

Operation planning summary

Detailed plan contains

Route

Processes

Process parameters

Machine and tool selections

Fixtures.

Example

CNC Machining Process Design of Transmission Shaft

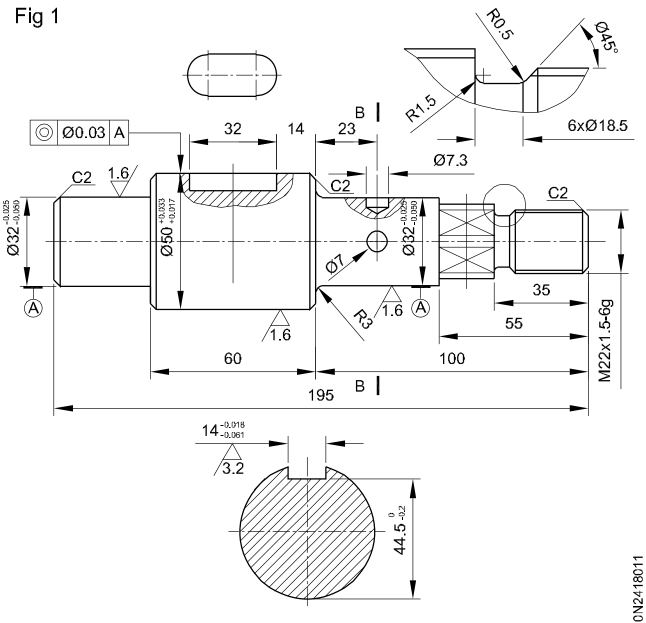
Transmission Shaft Parts Diagram Analysis

Drive shaft has two ϕ32 outer circle shaft segments, middle is ϕ50 Shaft section. So it belongs to the ladder shaft, There are steps in the middle. There is a routine keyway milling and flat milling, Drilling and cutting thread tanker. In addition, Drive shaft in terms of position requirements is: Shaft section Ô50n7 the coaxial tolerance of the reference A is ϕ0.03. Other no special requirements, it is processed according to the general axis parts processing.

The material of the shaft is made of steel 45, Heat treatment for quenching and tempering220-250HB, Note fillet R1.5, Note Dimensional tolerance IT14 level.

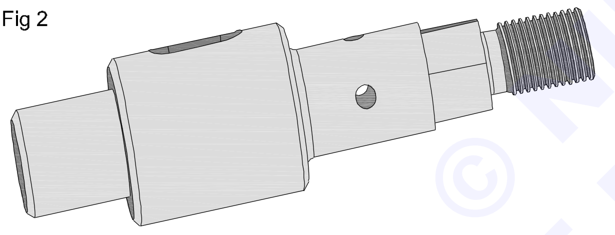
It can be seen from the drawing shaft of the processing content: there are two ∅32f7 outer circles, Ra1.6, on ∅50n7 outer circle, Ra1.6. Processing has a higher accuracy requirements. the ϕ50n7 outer circle with respect to the reference A Coaxial tolerance is ϕ0.03 Position accuracy is relatively high. groove, Chamfer C2 and thread It belongs to the general requirements, the processing time required to complete the semi finishing. The blind hole ϕ7 deep 3. The through hole ϕ7 and keyway (Surface roughness requirement is Ra3.2um) And four plane (surface roughness requirements for Ra3.2um), belonging to the precision requirement. It takes in after finishing CNC milling machine for processing. Transmission shaft overall accuracy requirements higher, process pay attention to three elements of cutting tool and the choice (Fig 1 & 2).

Fig 1



DRIVE SHAFT PART DRAWING

Fig 2



THREE DIMENSIONAL MODEL OF TRANSMISSION SHAFT

CNC Machining Process Analysis of Transmission Shaft

1 Positioning mode and installation

Positioning benchmarks follow the principle of unity, positioning and design datum coincidence. So to improve the programming and numerical simulation in the simplicity and accuracy of determined the axis of the bar, playing center holes at the two ends of the shaft, with both ends of the center hole alternative ideal center line positioning processing. Clamping mode: left the three jaw self centering chuck clamping, the right end of the top tight clamping.

2 Determine the parts of the blank

Selection 45 bar (bar), the size of the blank under: ϕ55x200.

3 Determine the processing route

Processing according to the coarse to fine, from far and near to the principle of determined. Namely, the first from right to left for rough lathing, and from right to left to finish and semi finish lathing. When the rough to interior blunt down and chamfering, in semi finish lathing the completion of key slot and plane and again chamfer processing. And then finish machining. The driving shaft and the specific processing steps roughly as follows: first rough lathe ϕ22, ϕ27, ϕ32, ϕ50 outside circle; Second Semi finish machining ϕ27,ϕ32, ϕ50 outside circle. Grooving•Screw thread; Finish machining ϕ32, ϕ50 outside circle; Keyway milling, plane milling; Last dill the bind hole of ϕ 7 deep 3 and through hole ϕ 7

Table 1 Selection of Machine Tool

No.	Machine Name	Processing contents	Model	Specifications
1	Plain lathe	Rough lathe ϕ32,ϕ50 outer circle	CA6140	ϕ400 ×650
2	CNC lathe	Semi fine lathe and fine lathe$32,$50 outside circle, lathe M22X1.5-6g thread	CK3665	ϕ360×650
3	CNC milling machine	Milling keyway ϕ14, the blind hole of ϕ7, the through hole of ϕ7 and the four plane	XD-40A	600×520×420

Selection of Transmission Shaft Machine Tool

1 Lathe roughing operation selection of horizontal CA6140 ordinary lathe because rough machining should pay attention to the principle of separation of coarse fine, but also to take into account the requirements and efficiency, comprehensive above factors, the use of rough machining horizontal CA6140 ordinary lathe.

2 Semi precision lathe, precision machining selection CK3665 CNC lathe, because the semi precision and precision of the lathe allowance has been relatively small, and the accuracy requirements are relatively high, comprehensive above factors, so the semi finish machining precision machining selection CK3665 CNC lathe.

3 Milling choose FANUC 0i system vertical milling machine.

Fixture Selection

Clamping ways: to determine part axis for the locating datum. Machining contour to ensure an installation process the outer contour, needs the three jaw chuck is clamped on the left, right end of the left with a center hole and tailstock peaked tightly to improve the rigidity of the process system. Milling machine with a V type block and dividing head, such as positioning. Fixture selection are shown in Table 2.

Table 2 Fixture selection

No.	Processing contents Name	Model	Specifications
1	Rough lathe ϕ32,ϕ50 outer circle	CA6140	Three jaw chuck, thimble
2	Semi fine lathe ϕ32, ϕ50 outside circle, lathe M22X1.5-6g thread	CK3665	
3	Fine lathe ϕ32, ϕ50 outside circle	CK3665	
4	Milling keyway ϕ14-0.018 -0.061	XD-40A	V shape block, dividing head
5	Milling keyway ϕ14, the blind hole of ϕ7, the through hole of ϕ7	XD-40A	
6	Milling four small plane	XD-40A	

Choice of cutting tools

The content of shaft parts of the processing is mainly dominated by turning, and must be rough and finish machining separate, intermediate and semi finishing processes. Therefore, processing, the selection of the tool according to the different of the content of the processing, to meet the different requirements of the tool, general in the rough machining focus on efficiency of tool performance, and fine processing more emphasis on tool precision capability. First of all, we should understand the content of the processing, processing sequence and processing precision requirement, and then select the processing necessary tool. Here, There will be the ϕ 32 and the ϕ 50 outer circle meanwhile finishing, should pay attention to is the need about partial knife. The specific choice of tool see Table 3.

Table 3 Specific Choice of Cutting Tools

No.	Operation name	Machining method and tool selection			
		Machining method	Tool content		
			Tool Name	Specifications	Tool No.
1	The end face and the Chamfer	Lathe	45°Turning tool	16	T01
2	The center hole	Drill	ϕ2 Center drill	A-2	T02
3	The 2×ϕ32-0.025 -0.050 Outer circle Ra1.6	Rough lathe outer circle Ra1.6	90°Turning tool	20	T03
		Semi fine lathe	93°Turning tool	22	T04
		fine lathe	Left and right offset knife	22	T05
4	The ϕ50+0.033 +0.017 outer circle Ra1.6	Rough lathe	90°Turning tool	20	T03
		Semi fine lathe	93°Turning tool	22	T04
		Fine lathe offset knife	Left and right	22	T05
5	The M22X1.5-6g thread tool	Lathe	Thread lathe		T06
6	The ϕ14-0.018 - 0.061 keyway	Milling keyway	Milling keyway knife	A-5	T07
	Four small plane The blind hole and through hole of ϕ7	End milling Drill hole	End milling knife ϕ7 drill	A-50 ϕ7	T08 T09

Measuring Tool Selection

Inspection is a part, precision of the outside dimension is in accordance with the requirements of the drawings, first of all, we should choose reasonable measuring. The so-called reasonable is refers to the selected measuring true, correct, effective measurement of the dimension accuracy of the parts. Usually measuring basis for:

1 Considering the measurement of parts of the site and the project. If parts have been measured and the aperture size should choose vernier caliper, inner micrometer or bore dial indicator.

2 Considering the accuracy of parts measurement (nominal dimensions and tolerances). In the choice of measuring tools, first of all, we should first see the dimension accuracy of the parts, smaller tolerances should select micrometer; tolerances is large, for the convenience of measurement, fast vernier caliper.

One of the options is to consider the economy of measurement. The same size can sometimes be achieved with different measuring tools.

Table 4 Measuring tool selection

No.	Processing content and accuracy requirements	Gage name	Specification	Accuracy
1	The f 32-0.025 0.050 and the ϕ 50+0.033+0.017 outer circle ra 1.6	Outside micrometer	50-75 mm	0.001
		Vernier caliper	25-50 mm	0.001 mm
		Surface roughness tester		
2	The M22X1.5-6g thread	Thread pass check		
3	The ϕ 14-0.018-0.061 keyway	Inside micrometer	0-25 mm	0.001 mm
		Depth micrometer	0-25 mm	0.001 mm
4	The blind hole and the through hole of ϕ 7 four small plane	Vernier caliper	0-200 mm	0.02 mm

Selection of cutting parameter

In the processing process, the need for a given amount of cutting (cutting depth, spindle speed and feed rate), so in the process must determine the CNC machining of three cutting factors, under the selected tool wear degree, according to the instructions for use of CNC machine tools, processing material type is cast iron, steel or non-ferrous metals, etc) processing requirements (roughing, semi finishing processing or finishing) and other technical requirements, and requires a combination of practical experience to determine the amount of cutting. At the same time, cutting parameters selection should also consider the dynamics of the machine tool should be high cutting speed and small amount of feed.

The principle of selection of cutting parameters is: to ensure the machining accuracy and surface roughness of the parts, to give full play to the performance of machine tools, to maximize production efficiency and reduce costs.

The cutting speed V c is converted into a speed formula as follows:

n= 1000V c/3.14d (r/min)

D-cutting surface diameter (mm)

Vc-cutting speed m/min

Table 5 Selection of Cutting Parameter

No.	Process contents	Three elements of cutting		
		Cutting depth (mm)	Feed rate (mm/r)	Cutting speed (m/min)
1	Rough lathe ϕ22, ϕ27, ϕ32 outer circle	1.5	0.4	110
2	Rough lathe ϕ32, ϕ50 outer circle	1.5	0.4	110
3	Semi fine lathe ϕ32 outer circle	0.5	0.2	132
4	lathe thread	-	-	80

5	Semi fine lathe ϕ27, ϕ32, ϕ50 outer circle	0.5	0.2	132
7	fine lathe ϕ32, ϕ50 outer circle	0.2	0.1	150
6	fine lathe ϕ32 outer circle	0.2	0.1	150
8	Milling keyway	2.75	0.3	120
9	Milling plane	2.5	0.3	20
10	drill the blind hole and through hole of ϕ7	7	0.15	16

Preparation of the Drive Shaft CNC Machining Process Card

Shaft CNC Machining Process Card Production

The CNC machining process card is a kind of process document that the process is a unit to explain parts machining process. It briefly process picture, is the foundation for formulating other process documents. This kind of card to each working procedure stipulation and the description is not concrete, therefore in mass production and mass production in general can not know the worker operation, but many as the production management numerical control processing use, Need to prepare the CNC machining process card, to guide the operator. But in a small batch production, usually no longer the preparation of a more detailed process files, it can directly guide the production, to guide the operation of workers.. In this case, the transmission shaft of CNC machining process card preparation and preparation of CNC machining process card, as shown in Table 6. The table is the transmission shaft CNC machining process card. You can use the computer aided process design for the preparation.

Table 6 - CNC Machining process card of drive shaft

							Product model			Part drawing number							
School Enterprice cooperation unit				**NC Machining process card**			Product name			Part name		Output shell	Page 1 of 1	Test pages			
Material card number	45			Work stock		Round steel	Blank dimen-sions		55x200	Each blank number		1	Number per piece	1	Remark		
											process equipment		Card			Work line	
No	Process name			Work order volume			Workshop	Workshop section	Set up	Texture	Case	Working procedure	Working procedure	Coopera-tion	Test	Quest final	Com-pletion
1	Rough lathe 1	1 lathe the right and face of the center hole 2 Rough lathe 22 27 32 under circle and pour lines arc and chamber					Machining		CA 140								
2	Rough lathe 2	1 lathe the left end face hit the center hole 2 Rough lathe 32 50 outer circle and chamber					Machining		CA 140								
3	Heat treatment countersinking HB220-250 HR						Heat treatment										
4	Fine lathe 2 1 Semi line lathe front end face connect center hole						Machining		CK3885								
5	Fine lathe 1	1 Center line lathe left end face, correct center hole 2 Semi line lathe 22 27 32 outside circle and chamber					Machining		CK3885								
6	Fine lathe 2	1 fine lathe line 92 and 50 center circle and chamber					Machining		CK3885								
7	Milling	Keyway milling place milling blind milling and through holes milling					Machining		XD-40A								
8	Fitter	To burr blunt					Machining										
9	Crack	Inspection					Inspector										

Programming

FANUC Oi Mate-TD system

```
O1005
N010 G54 G99 G21
N012 T0101
N013 M03 S1200
N014 G00 X60.0 Z5.0
N016 G71 U1.5 R1.0
N018 G71 P020 Q032 U0.5 W0.3 F0.3
N020 G00 X0.0
N022 G01 Z0.0 F0.1
N024 X28.0
N026 X32.0 Z-2.0
N028 Z-35.0
N030 X46.0
N032 X50.0 Z-98.0
N034 M30
O1006
N010 G54 G99 G21
N020 M03 S8000
N030 T0202
N040 GOO X60.0 Z5.0
N050 G71 U1.5 R1.0
N060 G71 P070 Q170 U0.5 W0.5 F0.2
```

```
N070 G00 X0.0
N080 G01 Z0.0 F0.1
N090 X18.0
N100 X21.8 Z-2.0
N110 Z-37.0
N120 X31.0
N130 Z-55.0
N140 X32.0 Z-97.0
N150 G02 X38.0 Z-100.0 R3.0
N160 G01 X46.0
N170 X50.0 Z-102.0
N180 G00 X100.0 Z100.0
N182 M05
N190 M00
N200 M03 S1500
N202 G00 X60.0 Z5.0
N204 G70 P070 Q170
N206 G00 X100.0 Z100.0
N210 T0303
N212 M03 S500
N220 G00 X60.0 Z60.0
N230 G75 R0.5
N240 G75 X18.5 Z-35.0 P0 Q0 F0.05
N250 G00 X100.0 Z100.0
N260 M05
N262 M00
N270 M03 S600
N280 T0404
N290 G00 X30.0 Z4.0 F1.5
N300 G92 X21.5 Z-33.0
N310 X20.7
N320 X20.1
N330 X19.8
N340 X19.7
N350 X19.6
N360 X19.5
N370 X19.5
N390 G00 X100.0 Z100.0
N400 M30
```

The technical requirements of the transmission shaft NC machining process design, parts of the material and blank forms are briefly introduced. The precision, performance and service life of the machine are directly influenced by the machining quality of shaft parts. The main contents of the shaft are introduced in detail— lathing and milling, At the same time, the process characteristics of the parts were described, the method and the steps of making the part machining process are also analyzed. Requires lathe the end face, lathe outside circle, cutting groove, lathe thread and other processes. The parts both in shape and size precision or accuracy of form and position are considered relatively simple one, but it also has unique is refined processing required about partial knife, but parts of the shape is ladder type, processing milling keyway, milling plane. It can be processed according to general shaft parts of the processing. When making decision, according to the properties of the material, the shape and the position accuracy, the choice of the appropriate plan, and then according to the surface roughness requirements, select the processing plan.

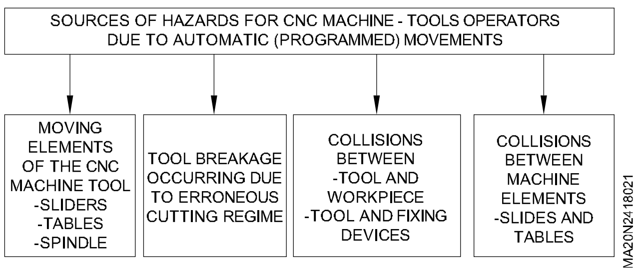
Collision in CNC turning machine tools

Objective: At the end of this lesson you shall be able to
- **describe the causes of collision and its impacts.**

Compared with ordinary machine tools, CNC turning-milling compound machine tools have high machining accuracy, good dimensional stability, low labor intensity, and convenient modern management. However, due to improper operation or programming errors, it is easy to cause the tool or tool holder to hit the work piece or the machine tool, the lighter will damage the tool and the processed parts, and the severer will damage the machine components, making the CNC turning and milling composite machine tool processing. Loss of accuracy, even causing personal accidents.

Fig 1 Sources of hazards which can affect the safety of the computer numerical control (CNC) operator.

Fig 1



Moving elements of the machine (machine slides, rotating tables, rotating main spindle) which move automatically under the control of the NC program;

Tool breakage occurring due to an erroneous cutting regime (higher cutting depths and speeds than are allowable), which can project tool fragments outside the working space of the machine;

Collisions between tools and the work piece and tools and fixing devices, occurring at high positioning speeds, which can project moving debris outside the working space of the machine. These collisions can occur due to erroneous programming or erroneous setup operations (fixing the work piece in other positions than the one considered in the programming stage);

Collisions between machine elements due to erroneous programming (related to multi-axis machining operations). In this case, not only should tools and work pieces be checked for collisions, but machining slides should also be checked for collisions against the rotating axes (rotating tables).

The following also the main reasons for tool collision failure of CNC turning-milling compound machine tools:

1 The simulation processing machine is not locked

Since the CNC turning-milling compound machine tool is locked by software, it is impossible to visually see whether the machine tool is locked in the simulation interface when the automatic operation button is pressed during simulation processing. There is often no tool setting during simulation. If the machine tool is not locked and running, tool collision is very likely to occur. Therefore, you should go to the running interface to confirm whether the machine tool is locked before simulating processing.

2 Forgetting to turn off the dry run switch during processing

In order to save time during program simulation, the dry run switch is often turned on for the CNC turning-milling compound machine tool. Dry run means that all motion axes of the machine tool run at the speed of G00. If the dry run switch is not turned off during processing, the machine tool ignores the given feed rate and runs at the speed of G00, causing accidents of hitting the tool and hitting the machine tool.

3 No reference point return after dry run simulation

When the program is verified, the machine tool is locked and the tool is in the simulation operation relative to the work piece processing (absolute coordinates and relative coordinates are changing). At this time, the coordinates do not match the actual position. The method of returning to the reference point must be used to ensure the machine. The zero point coordinate is consistent with the absolute and relative coordinates. If no problems are found after the program is checked, the machining operation will be carried out, which will cause tool collisions.

4 The direction of overt ravel release is wrong

When the turning and milling compound machine tool is over travel, you should press the over travel release button and move it in the opposite direction manually or manually to eliminate it. However, if the direction of release is reversed, it will cause damage to the machine tool. Because when the over travel release is pressed, the over travel protection of the machine tool will not work, and the travel switch of the over travel protection is already at the end of the travel. At this time, it may cause the workbench to continue to move in the over travel direction, eventually pulling the lead screw and causing damage to the machine.

5 Inappropriate cursor position during specified runtime

When specifying the operation, it is often executed downward from the cursor position. For the lathe, it is necessary to call the tool offset value of the used tool. If the tool is not called, the tool of the running program segment may not be the desired tool, and it is very likely that the tool collision accident may occur due to different tools. Of course, the coordinate system such as G54 and the length compensation value of the tool must be called first on the machining center and CNC milling machine. Because the length compensation value of each tool is different, it may cause tool collision if it is not called.

As a high-precision machine tool, the CNC turning and milling compound machine tool is very necessary to avoid collisions. The operator is required to develop the habit of being careful, careful, and operating the machine in the correct way to reduce the occurrence of machine tool collision.

Cost associated with Collision

The expenditure associated with collision are depends up on the severity of the collision.

If the tool breakage occurs only the cost of replacing the tool

If the spindle is hit on the work or table the expenditure will be more

At any cost if collision occurs one as to check machine thoroughly for its alignments and accuracy

If the machine is under maintenance contract the expenditure may be minimum.

Minimizing manual data entry will minimize the risk of human error. As long as an over-reliance on human intervention exists, CNC machining mistakes will continue to occur. Investing in CNC machine monitoring systems will contribute to efficiency improvements and eliminating human error.

Procedure of programme execution in different modes

Objective: At the end of this lesson you shall be able to
- **brief the operations of single block mode and automatic mode.**

1 MDI Automatic mode

MDI - means Manual Data Input. In this mode we can enter just one program block and execute it then and there. The program block so entered with not remain in Machine memory permanently. The program remains in the machine memory only till it is executed.

Procedure

- Switch on the machine.
- Auxillary on.
- Take the axis to reference points.
- Mount the job and take the zero offsets.
- Enter zero offsets and tool offsets in machine memory.
- Select MDI automatic mode.
- Enter the program block at the alpha numeric key board of the machine.
- Executive the program by pressing program execution key.

This mode of programe execution is useful for testing tool offsets and doing simple operations like facing etc.

Watch carefully executive of G00 commands.

Regulate the feed using feed selector. Stop the feed well in advance if the tool is found approaching any obstruction.

Single block mode

This mode is adopted for block by block execution of a programme which has already been entered in the machine memory. This mode is mainly used for programme proving by actually machining the part. During this mode of execution the machine can be kept idle between blocks as long as needed for checking purposes.

Procedure

- Power on.
- Auxiliary on.
- Reference points.
- Mount the job and take zero offsets.
- Mount the tool and take tool offsets.
- Enter zero offsets and tool offsets.
- Select automatic mode.
- Select single block mode.
- Press execution key for first block.
- Perform the required verification.
- Execute the next block. verify and continue till all blocks are executed.

Automatic mode

Selection of this mode enables automatic machining. This should be attempted only after the programme has been proved to be correct in all aspects. It is very important that there should be no mistake in the programme and one should satiffy himself during the trial run. It is also important to realize that once a programme is proved, we should not make even minor or innocent looking changes in it. In case any change is warranted, we may incorporate the same, but we should again do the trial run of the changed version of the programme in single block mode.

Procedure

- Power on.
- Auxiliary on.
- Reference points.
- Mount the job and measure zero offsets.
- Mount the tool and measure tool offsets.
- Enter the zero offsets and tool offsets.
- Select automatic mode.
- Enter the number of programme to be executed.
- Ensure that default settings displayed are correct.
- Start execution by pressing the relevant button on the control panel.
- Inspect the part for dimensions and geometry.

Types of offsets

Objectives: At the end of this lesson you shall be able to
- **list the types of offsets used in CNC machine**
- **describe the each types of offsets used in CNC machine.**

Offsets

Offsets are a platform for programing it does not require fixing the machine

Types of offsets

There are three types of offsets used in CNC machine, they are

1 Work offset (or) zero offset

2 Geometrical offset

3 Wear offset

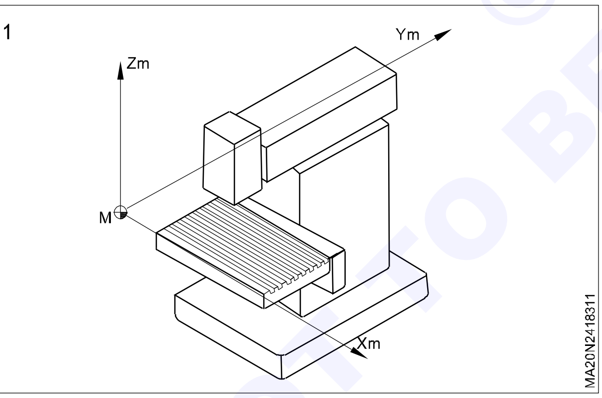
Work offset or Zero offset

Every manufacturer fix a reference point on the spindle of the machine, the fixing of reference point on the spindle position may vary CNC control to CNC control. The reference point is in relation to the machine zero or machine coordinate system (MCS)

Machine coordinate system

The machine coordinate system comprises all the physically existing machine zero reference point and tool changing point are defined in the machine coordinate system Fig 1.

Fig 1



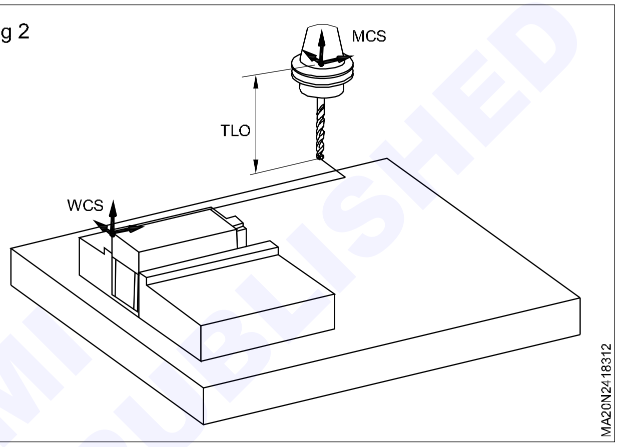
The tool will take the command and move with the reference to the machine reference point. To make the tool to move with reference to work piece. Work offset or zero offset has to be taken.

Work offset or work coordinate system (WCS)

Work coordinates system are positions set, that are referenced from machine home position with relation to the machine zero one can set multiple offset in a program. This is usually done when machining several parts at the same time.

There are specific components with in work offset. Work offset represents, the position of the work piece at which all the axes line at zero position . Tool offset represents the position of the cutter at which all the lines at position. (Fig 2)

Fig 2



Zero offset are the distance X0, Y0, and Z0 from machine zero to work zero in X,Y,Z directions. The set table zero offset specifies the position of the workpiece of the zero with reference to the machine zero. These offset values table G54 to G59. The values is enable by the program with work offset command CG54-G59.

Zero offset measurement

Call tool number one in MDI mode.

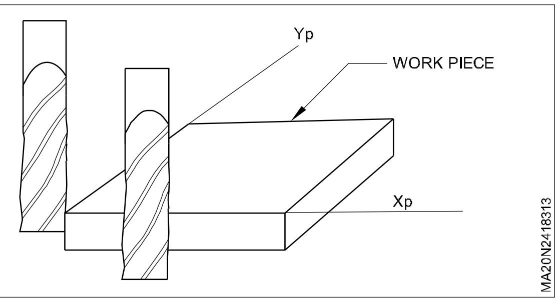
Go to jog mode. Remove the available tool if any.

Mount one reference tool / cylindrical pin of about 100mmlength / position finder. Make the zero offset value G54 X0, Y0, Z0.

Just touch the X surface of the workpiece as shown in Fig 3 below and take the absolute display reading of X-axis.

Just touch the Y surface of the workpiece as shown in Fig 3 below and take the absolute display reading of Y-axis.

Fig 3



Just touch the Z surface of the workpiece as shown in Fig 4 below and take the absolute display reading of Z-axis.

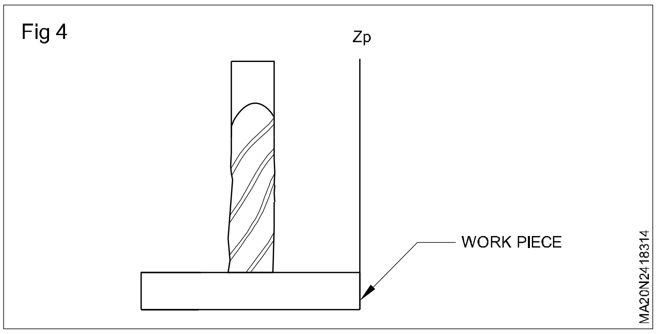


Fig 4

Reduce the radius value of the tool / cylindrical pin from the noted X and Y value. This given the zero offset of the corner of the job as shown. This may be entered under X and Y in one of the zero offset value i.e, in G55-G59.

Enter the same absolute display value in 'Z'.

Every time a new job is mounted, new 'Z' value should be taken through the reference tool.

As explained above and must be entered in the appropriate zero offset number i.e. G55 to G59.

Geometrical offset

The tool used in on CNC machine one of different geometry. The length and diameter of the tools vary accordingly.

Even if the work offset is taken for a tool it will not suit the other tools

So for every tool offset s taken and entered in tool offset value in tool offset table.

Tool length offset

The tool length is the distance between the tool holder reference point and the tool trip in general.

The statement varius depending upon the controller (Fig 5)

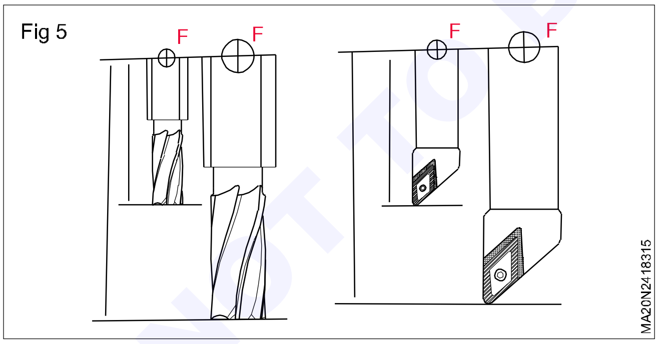


Fig 5

The tool offset measurement

If tool measuring system is available we can measure the tool offset through the tool pre setter.

If the tool pre setter is not available then reference tool method can be used to find out the tool offset.

Steps

Designate always tool No 01 as reference tool and avoid changing the tool.

Bring this tool and just touch the machined surface of the job,

Note down the Abs, value of Z at the time when the G54 Z-value is '0' - say A.

Use this 'Z' value for subsequent measurements of tool - (A)

Bring the next tool (No 02) for which measurement is to be set.

Touch the tool on the same workpiece surface.

Note the display value and subtract the Ref. Tool 'Z' value (A)

This will be the tool offset value for Tool No 02.

The same procedure can be adopted for other tools.

Enter all the value of length offsets in the CNC, against the Tool No.

The tool-offset can be measured directly if the value A is entered against G54 Z value.

Tool name, offset number along with the length offset may be kept separately and safely in a tool register for further use and reference.

The value A should betaken as zero Z-value for G55-G59.

To set zero offset of 'Z' for a new job just touch the reference tool in 'Z' direction and note down the Z value in the required offset number from G55 to G59. Please note that G54, z-value is 0. This avoids measuring of tool offset for other tool again.

Wear offset

Because of long running the tool tip may wear out and the dimension of the work piece may increase or decrease from the original programmed dimension.

To rectify this defect wear offset is used. If the programmed size is X40,48 and the machined size is 40.44mm in this 0.04mm is reduced due to wear.

To rectify this enter the 0.04 in corresponding tool and wear offset number.

In the inner dimension defects instead of positive value enter -0.04 in the corresponding wear offset number.

In general the tool offset number and wear offset number are same.

Work holding with temporary holding fixtures

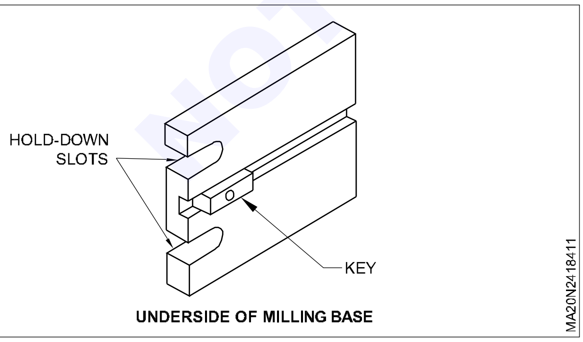
Objectives: At the end of this lesson you shall be able to
- **state about fixture and its purpose**
- **brief the use milling fixture and its alignments with machine table**
- **differentiate between Permanent & Temporary fixtures**
- **describe the modular type of fixture.**

A fixture is a work-holding or support device used in the manufacturing industry. Fixtures are used to securely locate (position in a specific location or orientation) and support the work, ensuring that all parts produced using the fixture will maintain conformity and interchangeability. Using a fixture improves the economy of production by allowing smooth operation and quick transition from part to part, reducing the requirement for skilled labor by simplifying how work pieces are mounted, and increasing conformity across a production run.

Milling Fixture

This holds the part in correct relation to the milling cutter. Fixture is attached to milling machine table. Milling fixture consists of the base, clamps, rest blocks or nest, locating points and gauging surfaces. The base of milling fixture consists of a base plate. A base plate has a flat and accurate undersurface and forms main body on which various components are mounted. This surface aligns with the surface of the mill table and forms the reference plane with respect to the mill feed movement. It may be constructed of steel plate or cast iron, depending upon the size and complexity of the part. The slots are provided in the base for clamping the fixture to the mill table. The base plate also has keyways along with length of the base for two keys. These keys are used to align the fixture on the milling machine table. The keys are pressed into the keyway at both ends of fixture and held there by socket head caps screw. This arrangement is shown in Figure 1. It is necessary to adjust the table by using feed movements until the correct position is attained. This can be done by trial and error cuts in the work piece. Milling is always first operation.

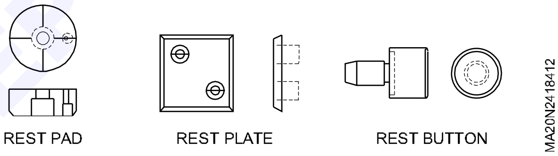
Fig 1



One must know the dimension of milling machine for designing the fixture. The various dimensions include the dimension of T-slots, centre-to-centre distance of T-slot, dimension of milling machine table and length of table travel in all three feed movements. Tool designer should provide enough clearance space around hold down slots for a nut, washer and wrench. Clamps on mill fixture must be extremely rigid. Cutting forces may change as the cutter enters or leaves the work piece and throw an extra load on clamps. Clamps should not be loosened by vibrations, which are caused by interrupted cutting by the mill cutter at the beginning and at the end of the cut. Clamp should be located opposite to bearing surfaces and locating points. These should be designed in a way so that these can be easily operated by the operator.

Rest blocks or bearing surfaces are located with the nest and provide support for the work piece. These surfaces change in design according to the shape and size of work piece. These are usually in the form of pins, pads or plates that are accurately placed in the base of fixture as shown in Figure 2. These surfaces are raised above the surface of base to permit chips to fall away and allow easy cleaning (Fig 2).

Fig 2



Permanent & Temporary Work holders

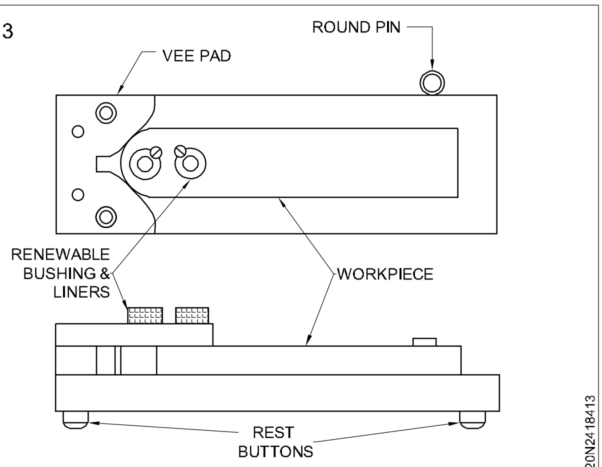
Jigs and fixtures are most often found where parts are produced in large quantities, or produced to complex specifications for a moderate quantity. With the same design principles and logic, work holding devices can be adapted for limited-production applications. The major difference between the various types of work holders, from permanent, to flexible, to modular and general-purpose work holders is the cost/benefit relationship between the work holder and the process. Some applications require jigs and fixtures solely for speed; others require less speed and higher precision. The requirements of the application have a direct impact on the type of the jig or fixture built and, consequently, the cost.

Permanent Jigs and Fixtures: Work holders for high-volume production are usually permanent tools. These permanent jigs and fixtures are most often intended for a single operation on one particular part. The increased complexity of permanent work holders yields benefits in improved productivity and reduced operator decision-making, which result in the tool having a lower average cost per unit or per run. Therefore, more time and money can be justified for these work holders.

In the case of hydraulic or pneumatic fixtures, inherent design advantages can dramatically improve productivity and hence, reduce per-unit costs even further, even though the initial cost to construct these fixtures is the most expensive of all fixture alternatives. In some cases, where machine-loading considerations are paramount, such as a pallet-changing machining center, even duplicate permanent fixtures may be justified (Fig 3).

Permanent jigs and fixtures are typically constructed from standard tooling components and custom-made parts. Figure 3 shows a typical permanent work holder for a drilling operation.

Fig 3



A PERMANENT WORK HOLDER USED FOR A DRILLING OPERATION

Low-volume runs and ones with fewer critical dimensions are often produced with throwaway jigs and fixtures. These tools would typically be one-time-use items constructed from basic materials at hand and discarded after production is complete. Although throwaway jigs and fixtures are technically permanent work holders, in effect they are actually temporary.

General-Purpose Work holders

In many instances, the shape of the part and the machining to be performed allow for the use of a general-purpose work holder such as a vise, collet, or chuck. These work holders are adaptable to different machines and many different parts.

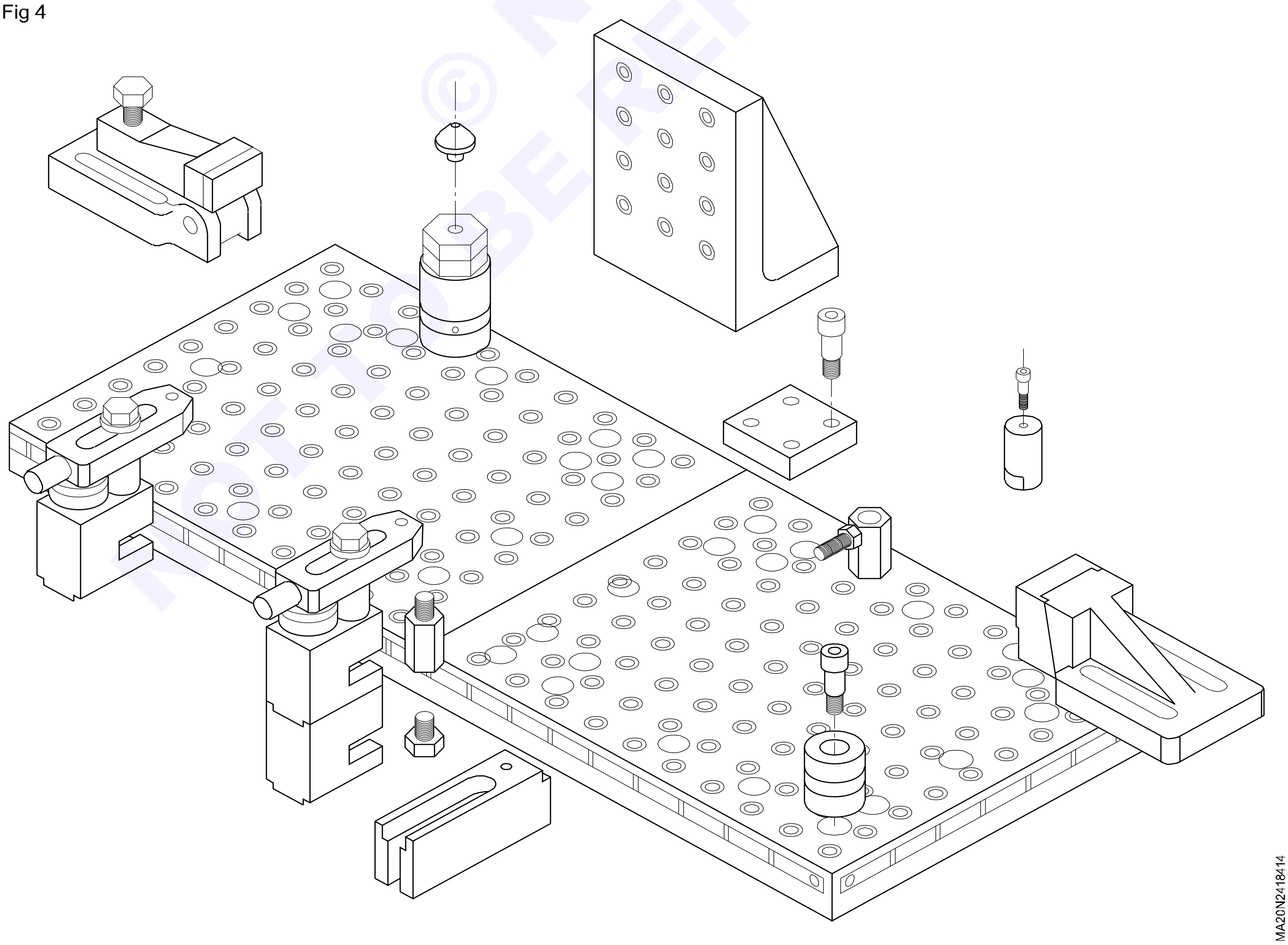
Since they are not part-specific, their versatility allows for repeated use on a variety of different or limited-production runs. The cost of these work holders would usually be averaged over years and might not even be a factor in job-cost calculations. The general-purpose nature of these work holders necessitates a higher level of operator care and attention to maintain consistency and accuracy. For these reasons, general-purpose work holders are not preferred for lengthy production runs.

Modular fixturing's role in workholding

Modular fixturing is not intended for every work holding operation, but when it is appropriate, it both increases production and reduces fixturing costs. Modular fixturing is not a replacement for permanent fixturing; rather it is an upgrade from "no fixturing," just a step below permanent fixturing.

Definition of Modular Fixturing (Fig 4)

Fig 4



Modular fixturing is a work holding system using a series of standardized components for building specialized work holders. As shown in Figure 4, a modular wor kholder is assembled from a variety of standard off-the-shelf tooling plates, supports, locating elements, clamps, and similar components. The components are assembled with socket-head cap screws and locating screws. A modular-fixturing system may have hundreds of different elements. The components can be assembled in different combinations to build an unlimited variety of work holders.

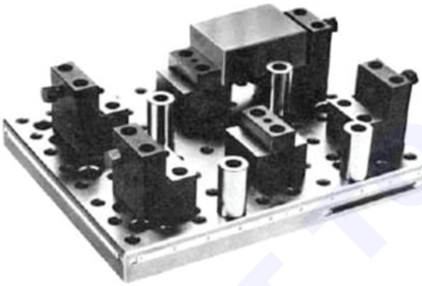
Modular work holders can be assembled entirely from standard off-the-shelf components.

The assembly process is quite simple. Components are designed to be used together, so each has an identical hole pattern. To build a work holder, the components are simply positioned as required and attached with locating or fastening screws. The simplicity reduces training time and permits technicians to begin building work holders almost immediately. Regardless of the manufacturing operations, a modular-fixturing system can provide work holders for almost any work piece.

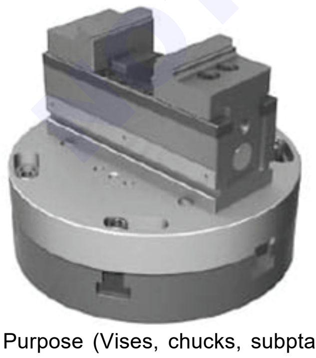
The Hierarchy of Work holding Options (Fig 5)

Fig 5

Permanant Fixing (Special purpose)



Modular Fixturing



General Purpose (Vises, chucks, subptates and table mounted clamps)

To understand how modular fixtures relate to work holding in general, an awareness of the various forms of work holders is necessary. Work holder forms can be grouped into three general categories, Figure 6. The least-complicated tools are the general-purpose work holders. The most-complex and most-detailed work holders are the special-purpose, or permanent, fixtures. Between the two are modular fixtures. Modular fixturing bridges the gap between the general-purpose and special-purpose work holders. Although the three forms of fixturing may seem completely different, each is actually a further development, or refinement, in the work holding process.

General-purpose work holders are the simplest form of fixturing device. This category includes a wide variety of standard clamps, vises, chucks, and similar standard off-the-shelf components. General-purpose components are reusable and normally the least expensive. Although these components represent a smaller initial investment, they are often inadequate or unsatisfactory for complex parts or high-volume production.

Modular fixturing fills the significant gap between general purpose and permanent work holders.

Permanent work holders are specifically designed and constructed for a single work piece or family of parts. These work holders, though usually the most efficient, are also the most expensive. Permanent work holders are built with a variety of standard and custom-made parts to meet specific requirements. These fixtures are the best choice for high-volume or repeated production runs.

Modular work holders, in the most-basic sense, can be described as special-purpose work holders assembled from general-purpose components. Figure 6 shows a typical system of modular-fixturing components. The concept of modular fixturing, rather than a departure from conventional fixturing methods, is actually a combination of the best attributes of both special- and general-purpose work holding methods. Modular fixtures are built with the accuracy and detail of special-purpose work holders, but with reusable and universal components, these fixtures compare favorably in cost to general-purpose work holders.

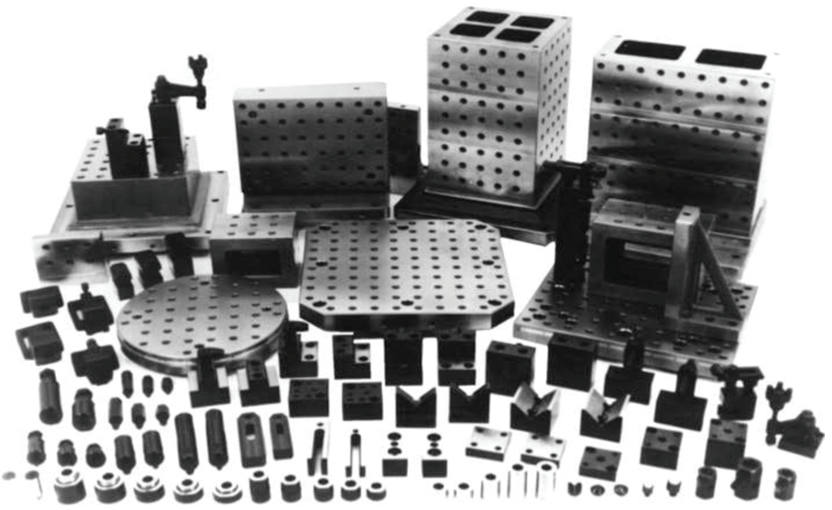
Applications for Modular Fixturing

Modular work holders are particularly well suited for one-time jobs, infrequent productions runs, prototype parts, replacement parts, trial fixturing, and temporary tooling.

One-Time Jobs: One-time jobs, found especially in job shops, are ideal for modular fixturing. With modular fixturing, a work holder can be economically built even for a one-part run. In most job shops, each machine tool does a variety of tasks in a single day. With modular fixturing, jobs can be performed with specialized work holders at a cost very competitive with crude machine-table setups. Here, modular fixtures increase quality and accuracy, yet still maintain competitive costs.

Infrequent Production Runs: Jobs that do not repeat on a regular basis are well suited to modular fixturing. Today shorter lead times are quite common. Modular fixturing permits rapid setup of short-notice production runs. Once again, modular work holders offer many of the benefits of special-purpose tooling at a fraction of the cost.

Fig 6



Prototype Parts: Prototype or experimental parts frequently require special work holders. Since prototype work pieces are often changed or redesigned, the cost of building jigs or fixtures for each new variation is prohibitive. Modular fixturing is the best alternative. With modular work holders, each variation of the work piece can be quickly fixtured with little or no downtime.

Replacement Parts That Are Made to Order: Replacement parts are an expensive problem in many companies. In the past, these parts were made in large lots and placed in storage. Some parts were quickly sold while others were never ordered. Modular fixturing eliminates the need for an inventory of slow-moving replacement parts. Modular fixturing permits a company to respond to orders as they are received. Instead of shipping parts from an inventory, parts can be made as needed.

Trial Fixturing Techniques: Trial fixturing is common throughout manufacturing organizations. Before any product goes into production, workholders must be designed, built, and tested. Assembling a modular workholder for workpiece allows the tool designer to test new tool designs and find problem areas. Modular workholders allow a designer to fine tune tooling ideas before a final production workholder is built.

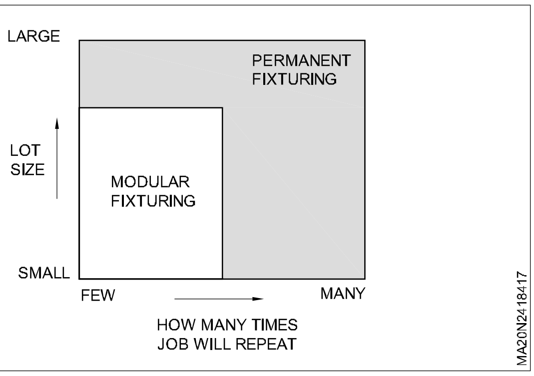
While Permanent Fixtures Are Built or Repaired. No matter how well a permanent jig or fixture is built, most require repair or maintenance. A modular work holder can easily be assembled and placed into production while the permanent work holder is repaired. Modular fixtures also buy time for initial building of permanent fixtures without delaying production.

Jobs That Will Repeat Many Times: Recurring production runs are those that repeat on a regular basis. The choice of fixturing method depends more on the frequency of production runs than on the number of parts per run. As shown in Figure 6, choosing between modular and permanent fixturing for a particular job depends on how often the job will run, not just on lot size.

Modular work holders are usually disassembled after each job: Each time a job is run, the modular fixture requires reassembly. Permanent work holders are normally built for a complete product run. If a job repeats on a regular basis, a permanent fixture is the better choice. Modular fixtures could remain assembled between production runs, thus becoming a permanent work holder, but this negates the economy of reusing modular components.

Where Fixture Compactness Is Important: Another factor to consider in the selection of a work holder is the size of the completed fixture. As a rule, modular work holders tend to be larger than their special-purpose counterparts. Permanent work holders are normally built from custom-made elements and base plates that permit a smaller, more-compact work holder. Modular components are intended for a variety of applications and, though more universal, are larger than comparable custom-made components. So, when space is limited, such as with multiple-part setups, modular work holders may prove to be too large. (Fig 7)

Fig 7



Modes of operation

Objective: At the end of this lesson you shall be able to
- **state the manual mode in CNC machines**
- **describe the jog, incremental and MPG modes**
- **explain the edit and memory modes.**

Manual mode

In the manual mode, the CNC machine behaves like a standard or conventional machine. A CNC machining center behaves like a manual milling machine. A CNC turning center behaves like an engine lathe.

The physical positions of the mode switch that are included as manual modes include manual or jog, incremental, hand wheel and reference return.

With the manual mode, the operator of a CNC machine is allowed to press buttons, turn hand wheels and activate switches in order to attain the desired machine function. The activation of each button or switch in the manual mode has an immediate response. For example, if the correct mode switch position is selected and if the correct button is pressed, the spindle will start. If a switch is turned on the coolant will come on.

Note that those things that cannot be done in the manual mode must be done in the manual data input mode, the second mode of operation that is MDI mode.

Jog mode

In the jog mode, pressing a feed axis and direction selection switch on the machine operator's panel continuously moves the tool along the selected axis in the selected direction.

The jog feed rate can be adjusted with the jog feed rate over ride dial. Pressing the rapid traverse switch moves the tool at the rapid traverse feed rate regardless of the position of the jog feed rate over ride dial. Manual operation is allowed for one axis at a time. 3 axes can be selected at a time.

Steps for jog feed

1 Press the jog switch, one of the mode selection switches.

2 Press the tool is to be moved. While the switch is pressed, the tool moves at the feed rate specified. The tool stops when the switch is released.

3 The jog feed rate can be adjusted with the jog feed rate over ride dial.

4 Pressing the rapid traverse switch while pressing a feed axis and direction selection switch moves the tool at the rapid traverse rate. Rapid traverse over ride by the rapid traverse override switches effective during rapid traverse.

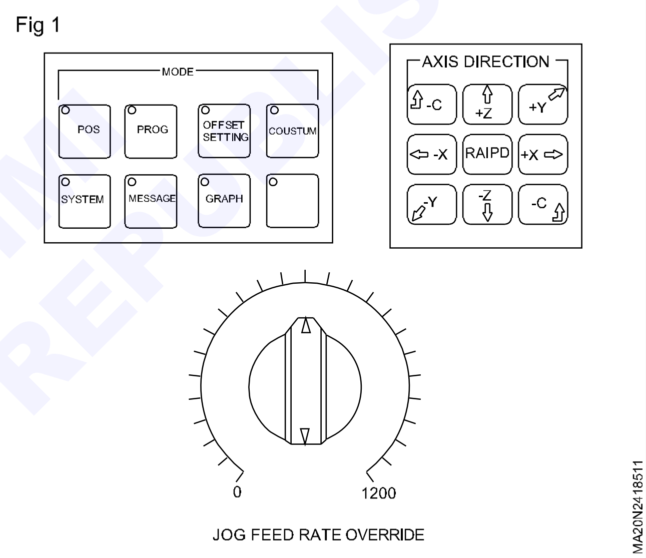
The above is an example. Refer to the appropriate manual provided by the machine tool builder for the actual operations.

Feed rate, time constant and method of automatic acceleration/deceleration for manual rapid traverse are the same as G00 in programmed command.

Changing the mode to the jog mode while pressing a feed axis and direction selection switch does not enable jog feed. To enable jog feed, enter the jog mode first, then press a feed axis and direction selection switch.

If reference position return is not performed after power-on, pushing RAPID TRAVERSE button does not actuate the rapid traverse but the remains at the JOG feed rate (Fig 1).

Fig 1



Incremental feed

In the incremental (STEP) mode, pressing a feed axis and direction selection switch on the machine operator's panel moves the tool one step along the selected axis in the selected direction. The minimum distance the tool is moved is the least input increment. Each step can be 10, 100 or 1000 times the least input increment. This mode is effective when a manual pulse generator is not connected. (Fig 2).

Tool each time a switch is pressed, the tool moves one step in the direction specified by the switch. 'Z', 'X', 'Y'.

Steps for incremental feed

1 Press the STEP switch, one of the mode selection switches.

2 Select the distance to be moved for each step with the magnification dial. (Figs 3 & 4)

Fig 2

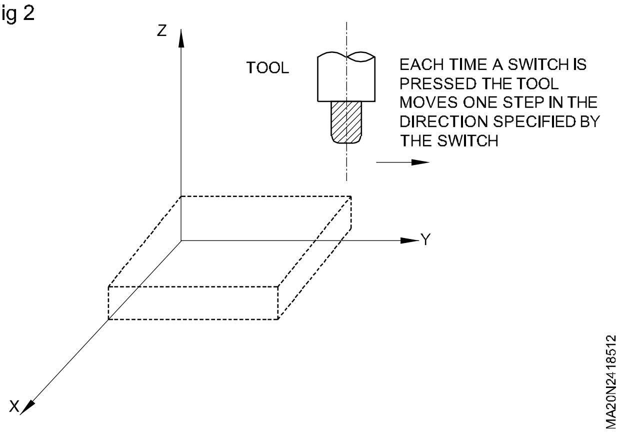


Fig 3

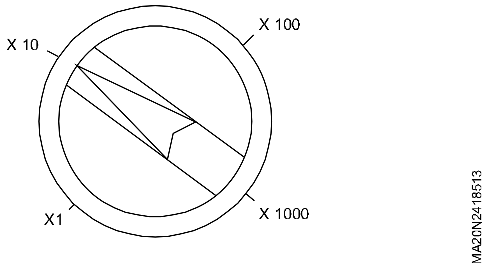
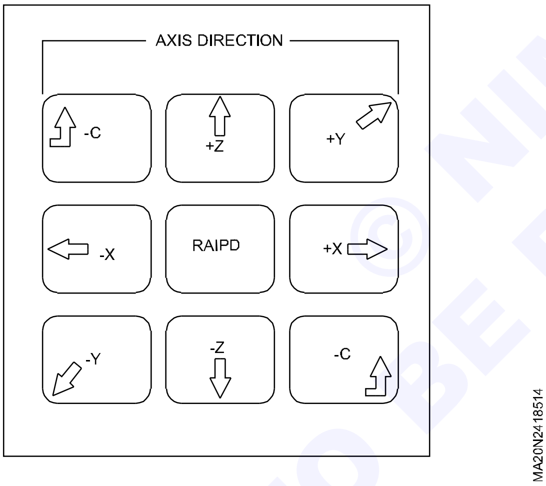


Fig 4



3 Press the feed axis and direction selection switch corresponding to the axis and direction the tool is to be moved. Each time, a switch is pressed, the tool moves one step. The feed rate is the same as the jog feed rate.

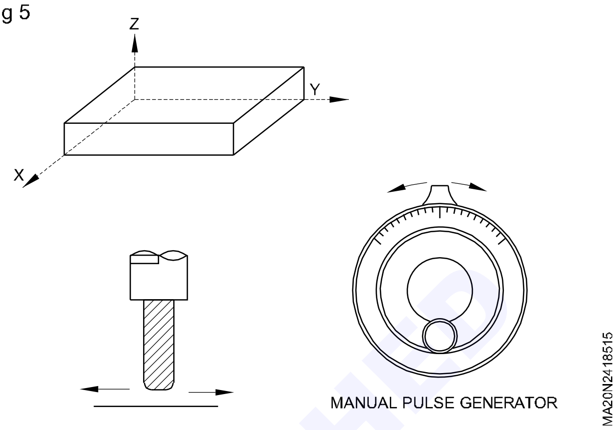
4 If an axis direction selection switch is pressed after rapid traverse has been selected, movement is made at the rapid traverse rate. Rapid traverse over ride by the rapid traverse over ride switch is effective during rapid traverse.

The above is an example. Refer to the appropriate manual provided by the machine tool builder for the actual operations.

Manual pulse generator (MPG)/Manual handle feed

In the handle mode, the tool can be minutely moved by rotating the manual pulse generator on the machine operator's panel. Select the axis along which the tool is to be moved with the handle feed axis selection switches. The minimum distance the tool is moved when the manual pulse generator is rotated by one graduation is equal to the least input increment. Or the distance the tool is moved when the manual pulse generator is rotated by one graduation can be magnified by 10 times or by one of the two magnifications specified by parameters (Fig 5).

Fig 5



Steps for manual handle feed

1 Press the HANDLE switch, one of the mode selection switches.

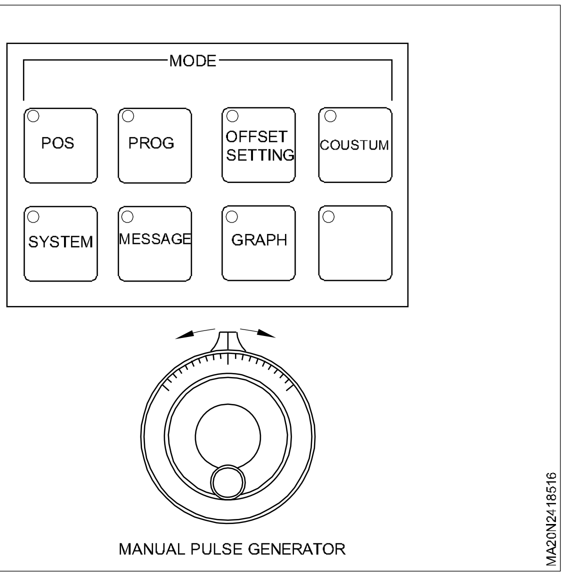
2 Select the axis along which the tool is to be moved by pressing a handle feed axis selection switch.

3 Select the magnification for the distance the tool is to be moved by pressing a handle feed magnification switch. The minimum distance the tool is moved when the manual pulse generator is rotated by one graduation is equal to the least input increment.

4 Move the tool along the selected axis by rotating the handle. Rotating the handle 360° moves the tool the distance equivalent to 100 graduations.

The above is an example. Refer to the appropriate manual provided by the machine tool builder for the actual operations. (Figs 6).

Fig 6



Manual data input mode

This mode includes two positions on the mode switch, the edit position and the manual data input (MDI) position. In each case, the operator will be entering data through the keyboard on the control panel and display screen.

Though these mode switch positions have substantial differences, we consider then together for two reasons. First, both mode switch positions involve entering data through the keyboard. With the edit position of the mode switch, a program is entered or modified. With the MDI position of the mode switch CNC commands are entered and executed.

Second, both provide manual capabilities that can also be done in a more automatic way. In edit mode, an operator can enter CNC programs into the control memory. This can also be accomplished by loading the program from some outside device, such as a computer or tape reader.

In MDI mode, CNC commands are entered through the keyboard and display screen manually and can be executed once. If the command must be executed a second time, the operator must enter the MDI command a second time. However, if the same command is included in a program, it can be executed automatically, over and over without having to be re-typed.

Manual data input

Almost anything that can be done within a CNC program can be done in the MDI mode. If an operator is proficient with CNC commands, many times they can perform functions more quickly with MDI than they can by other manual means. For example, the reference return position can be attained through manual means (the reference return mode of the mode switch) and through MDI commands. A good operator can perform this operation faster in the manual data input mode.

But the most important reason for the MDI mode is to allow the operator to perform manual operations that are not possible by using the manual mode. For example, some machining centers have no manual buttons or switches to select the spindle speed, direction and turn the spindle on and off. With this type machine, if the operator needs to manually turn the spindle on for any reason, they must use MDI to do so. In the same way, some turning centers have no manual means with which to change tools and thus must use MDI commands.

The manual data input mode switch position can even be used for machining a work piece. Since almost all CNC commands are possible by MDI (including G00, G01, G02 and G03), the operator can make machining commands in a way similar to how they are commanded in a CNC program. However, the operator must be very careful when using MDI to machine work-pieces. The command will be executed as entered. If the operator makes a mistake in the MDI mode, disaster could result. There is no chance to verify a CNC command executed by MDI as can be done with a CNC program.

MDI operation

In the MDI mode, a program can be inputted in the same format as normal programs and executed from the MDI panel. MDI operation is used for simple test operations. The following procedure is given an as example. For actual operation, refer to the manual supplied by the machine tool builder.

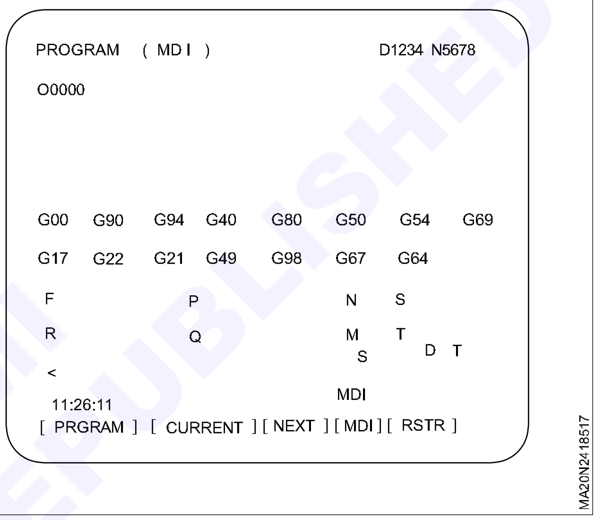
Procedure for MDI Operation - B

1 Press the MDI mode selections switch.

2 Press the PEGRM function key on the CRT/MDI panel to select the program screen.

The following screen appears: (Fig 7)

Fig 7



Program number Ooooo is entered automatically.

3 Prepare a program to be executed by an operation similar to normal program editing. M99 specified in the last block can return control to the beginning of the program after operation ends. Word insertion, modification, deletion, word search, address search and program search are available for programs created in the MDI mode.

4 To entirely erase a program created in MDI mode, use one of the following methods:

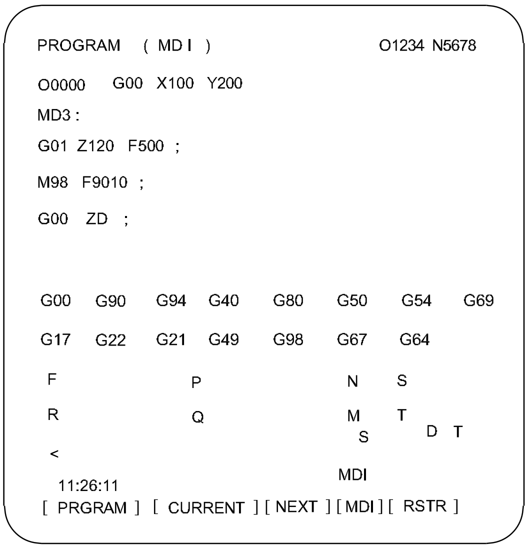
a Enter address O, then press the DELETE key on the MDI panel.

b Alternatively, press the RESET key.

5 To execute a program, set the cursor on the head of the program. (Start from an intermediate point is possible). Push Cycle Start button on the operator's panel. By this action, the prepared program will start.

When the program end (M02, M30) or ER (%) is executed, the prepared program will be automatically erased and the operation will end. By command of M99, control returns to the head of the prepared program. (Fig 8)

Fig 8



```
PROGRAM   ( MD I )                          O1234  N5678
O0000    G00  X100  Y200
MD3 :
G01  Z120   F500  ;
M98   F9010  ;
G00   ZD   ;

G00   G90   G94  G40   G80   G50   G54   G69
G17   G22   G21  G49   G98   G67   G64
 F              P              N    S
 R              Q              M    T
                                S      D  T
 <
  11:26:11                      MDI
[ PRGRAM ]  [ CURRENT ] [ NEXT ] [ MDI ] [ RSTR ]
```

To stop or terminate MDI operation in midway through, follow the steps below.

a Stopping MDI operation Press the feed hold switch on the machine operator's panel. The feed hold lamp goes on and the cycle start lamp goes off. When the cycle start switch on the machine operator's panel is pressed, machine operation restarts.

b Terminating MDI operation.

Press the RESET key o the CRT/MDI panel.

Automatic operation is terminated and the reset state is entered. When a reset is applied during movement, movement decelerates then stops.

The previous explanation of how to execute and stop memory operation also applies to MDI operation, except that in MDI operation B, M30 does not return control to the beginning of the program (M99 performs this function).

Programs prepared in the MDI operation will be erased in the following cases;- In MDI operation, if M02, M30 or ER (%) is executed.

Edit mode

In edit mode we can enter CNC programs into the control memory and modify current programs. Almost all CNC controls allow the operator to store multiple programs. Programs are typically organized by program number (commonly the O word). The operator will be allowed to call up the desired program from within the control's memory, making it the active program.

CNC controls give the user the ability to do atleast three basic things in the edit mode. The operator can insert new information into the program, alter the current information in the program and delete information from the program. Some CNC controls also allow the operator to do global editing, meaning they can cut and paste and find and replace data.

Along with these basic features, the operator will be allowed to search or scan the program for key information. For example, the control can quickly search to the next occurrence of any programming word. If the programmer used sequence numbers (N words) the operator could use the sequence number of an incorrect command as the word used to search.

Program operation mode

The memory (or auto) mode is used to execute programs from the control's memory. As long as the control's memory is of sufficient size to hold the entire CNC program (as it almost always will be), the memory mode should be used to execute the program.

While there could be several programs stored in the control's memory, only one is active. This is the program that will be run when the cycle start button is pressed. On most controls, the edit mode is used to choose the active program.

With most CNC controls, while a program is being executed from memory, the programmer is allowed to see one full page of the program on the control's display screen. As the program is executed, the cursor will scroll through the program, letting the programmer see the commands that follow the command currently being executed.

Steps for memory operation

1 Press the AUTO mode selection switch.

2 Select a program from the registered programs. To do this, follow the steps below.

- Press PRGRM to display the program screen.
- Press address O.
- Enter a program number using the numeric keys.
- Press the cursor key.

3 Press the cycle start switch on the machine operator's panel. Automatic operation starts and the cycle start lamp goes on. When automatic operation terminates, the cycle start lamp goes off.

4 To stop or cancel memory operation midway through follow the steps below.

  a Stopping memory operations Press the feed hold switch on the machine operator's panel. The feed hold lamp goes on and the cycle start lamp goes off. The machine responds as follows:

    i When the machine was moving, feed operation decelerates and stops.

    ii When dwell was being performed, dwell is stopped.

    iii The current operation, instigated by executing by an M.S or T command, is continued.

  b Terminating memory operation.

Press the RESET key on the CRT/MDI panel.

Automatic operation is terminated and the reset state is entered. When a reset is applied during movement, movement decelerates then stops.

Automatic operations

After memory operation is started, the following are executed.

A one-block command is read from the specified program.

The block command is decoded. The command execution is started.

The command in the next block is read.

Buffering is executed. That is, the command is decoded to allow immediate execution. Immediately after the preceding block is executed, execution of the next block can be started. This is because buffering has been executed.

Hereafter, memory operation can be executed by repeating the steps 4 to 6.

Memory operation can be stopped using one of two methods: Specify a stop command or press a key on the machine operator's panel.

- The stop commands include M00 (program stop), M01 (optional stop) and M02 and M30 (program end).
- There are two key to stop memory operation: The feed hold key and reset key.

Memory operation is stopped after a block containing MOO is executed. When the program is stopped, all existing model information remains unchanged as in single block operation. The memory operation can be restarted by pressing the cycle start button. Operation may vary depending on the machine tool builder. Refer to the manual supplied by the machine tool builder.

Similarly to M00, memory operation is stopped after a block containing M01 is executed. This code is only effective when the Optional Stop switch on the machine operator's panel is set to ON. Operation may vary depending on the machine tool builder. Refer to the manual supplied by the machine tool builder.

When M02 or M30 (specified at the end of the main program) is read, memory operation is terminated and the reset state is entered. In some machines, M30 returns control to the top of the program. For details, refer to the manual supplied by the machine tool builder.

When Feed Hold button on the operator's panel is pressed during memory operation, the tool decelerates to a stop at a time.

Automatic operation. Stopping and terminating by any one of the following:

Program stop (M00)

Optional stop (M01)

Program end (M02, M30)

Feed hold

Reset

Optional block skip.

Editing, entering programs and offsets in machine console

Objective: At the end of this lesson you shall be able to
- **enter program on machine console**
- **edit program on machine console**
- **enter work offset on work offset page**
- **enter tool offset on tool offset page.**

Programs can be created in the edit mode using the program editing functions.

Steps for creating programs

Enter the EDIT mode.

Press the PROG key.

Press address key O and enter the program number.

Press the INSERT key.

Create a program using the program editing functions.

Note the following to enter a comment

Control - in code ")" cannot be registered by itself. Comments entered after the INSERT key is pressed must not begin with a number, space, or address 0.

In an abbreviation for a macro is entered, the abbreviation is converted into a macro word and registered.

Address 0 and subsequent numbers, or a space can be entered but are omitted when registered.

Editing programs

Editing includes the insertion, modification, deletion, and replacement of words. Editing also includes deletion of entire program and automatic insertion of sequence numbers. The extended part program editing function can copy, move, and merge programs. The chapter also describes programs search, sequence number search, word search, and address search.

Heading program

The cursor can be jumped to the top of a program. This function is called heading the program pointer. This section describes the three methods for heading the program pointer.

Procedure for heading a program

Method 1

Press RESET when the program screen is selected in edit mode. When the cursor has returned to the start of the program, the contents of the program are displayed from its start on the screen.

Method 2

Search for the program number.

Press address O when a program screen is selected in the **memory** or **edit** mode

Input a program number.

Press the soft key [o SRH].

Method 3

Select **[Memory]** or **[Edit]** mode

Press **PROG**

Press the **[(OPRT)]** key.

Press the **[REWIND]** key.

Inserting a word

Procedure for inserting a word

Search for or scan the word immediately before a word to be inserted.

Key in an address to be inserted.

Key in data.

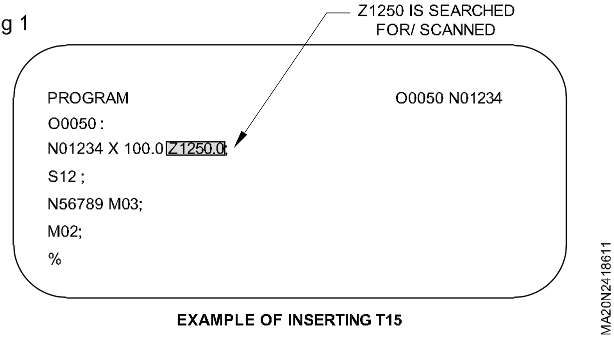
Press the INSERT key.

Example of inserting T15 (Figs 1&2)

Procedure

Search for or scan Z1250.

Fig 1



```
PROGRAM                              O0050 N01234
O0050 :
N01234 X 100.0 Z1250.0;
S12 ;
N56789 M03;
M02;
%
```

EXAMPLE OF INSERTING T15

Key in T 1 5

Press the INSERT key

Altering a word

Procedure for altering a word

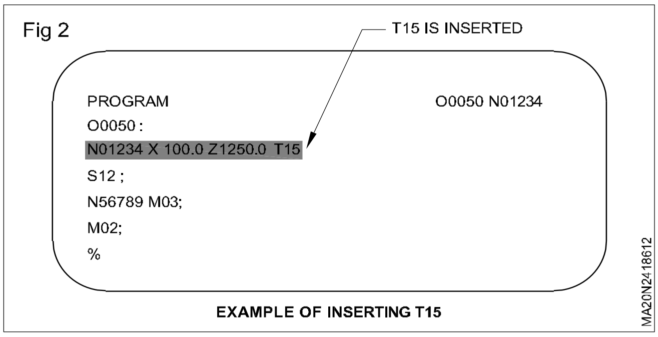
Search for or scan a word to be altered

Key in an address to be inserted

Key in data

Press the ALTER key

Fig 2



T15 IS INSERTED

```
PROGRAM                              O0050 N01234
O0050 :
N01234 X 100.0 Z1250.0  T15
S12 ;
N56789 M03;
M02;
%
```

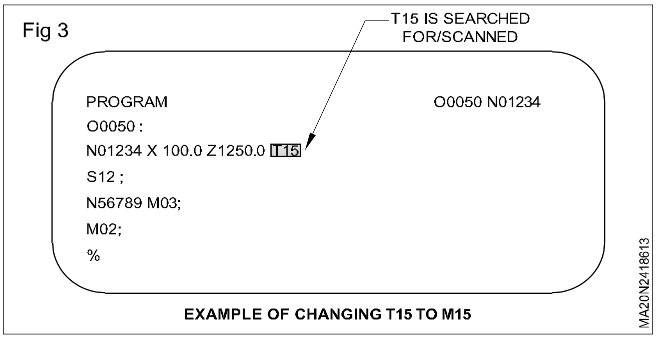
EXAMPLE OF INSERTING T15

Example of changing T15 to M15 (Figs 3 & 4)

Procedure

Search for or scan T15

Fig 3



T15 IS SEARCHED FOR/SCANNED

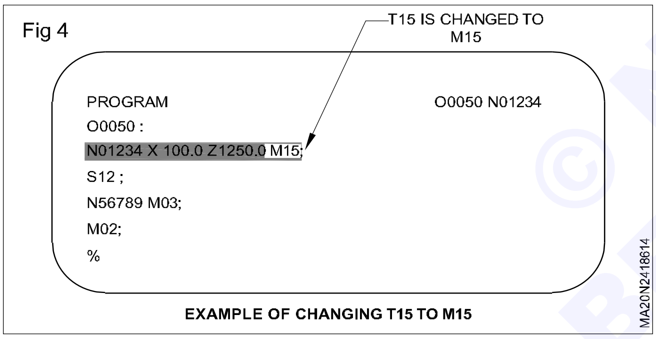
```
PROGRAM                              O0050 N01234
O0050 :
N01234 X 100.0 Z1250.0 T15
S12 ;
N56789 M03;
M02;
%
```

EXAMPLE OF CHANGING T15 TO M15

Key in M 1 5

Press the ALTER key

Fig 4



T15 IS CHANGED TO M15

```
PROGRAM                              O0050 N01234
O0050 :
N01234 X 100.0 Z1250.0 M15;
S12 ;
N56789 M03;
M02;
%
```

EXAMPLE OF CHANGING T15 TO M15

Deleting a word

Procedure for deleting a word

Search for or scan a word to be deleted.

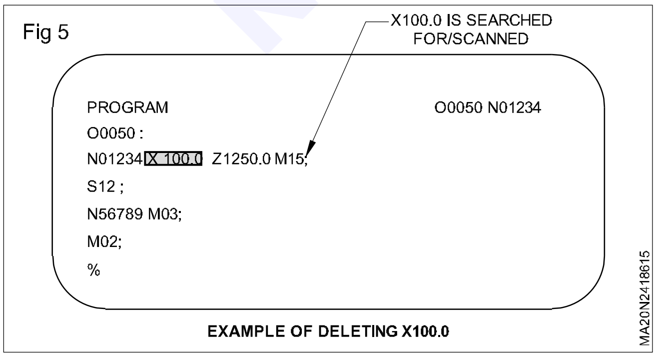
Press the DELETE key.

Example of deleting X100.0 (Figs 5&6)

Procedure

Search for or scan X100.0

Fig 5



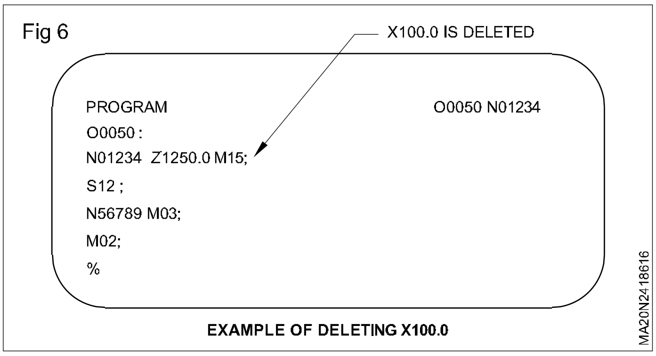
X100.0 IS SEARCHED FOR/SCANNED

```
PROGRAM                              O0050 N01234
O0050 :
N01234 X 100.0  Z1250.0 M15;
S12 ;
N56789 M03;
M02;
%
```

EXAMPLE OF DELETING X100.0

Press the DELETE key.

Fig 6



X100.0 IS DELETED

```
PROGRAM                              O0050 N01234
O0050 :
N01234  Z1250.0 M15;
S12 ;
N56789 M03;
M02;
%
```

EXAMPLE OF DELETING X100.0

Inserting, altering and deleting a word

This selection outlines the procedure for inserting, modifying, and deleting a word in a program registered in memory.

Procedure for inserting, altering and deleting a word

Select edit mode.

Press PROG

Select a program to be edited.

If a program to be edited is selected, perform the operation. If a program to be edited is not selected, search for the program number.

Search for a word to be modified.

- Scan method
- Word search method

Perform an operation such as altering, inserting, or deleting a word.

Explanation

Concept of word and editing unit

A word is an address followed by a number. With a custom macro, the concept of word is ambiguous. So the editing unit is considered here. The editing unit is a unit subject to alteration or deletion in one operation. In one scan operation, the cursor indicates the start of an editing unit. An insertion is made after an editing unit. Definition of editing unit.

i Program portion from an address to immediately before the next address

ii An address is an alphabet, if, while, goto, end, do = or; (EOB). According to this definition, a word is an editing unit. The word "word," when used in the description of editing, means an editing unit according to the precise definition.

Warning :

The user cannot continue program execution after altering, inserting, or deleting data of the program by suspending machining in progress by means of an operation such as a single block stop or feed hold operation during program execution. If such a modification is made, the program may not be executed exactly according to the contents of the program displayed on the screen after machining is resumied. So, when the contents of memory are to be modified by part program editing, be sure to enter the reset state or reset the system upon completion of edition before executing the program.

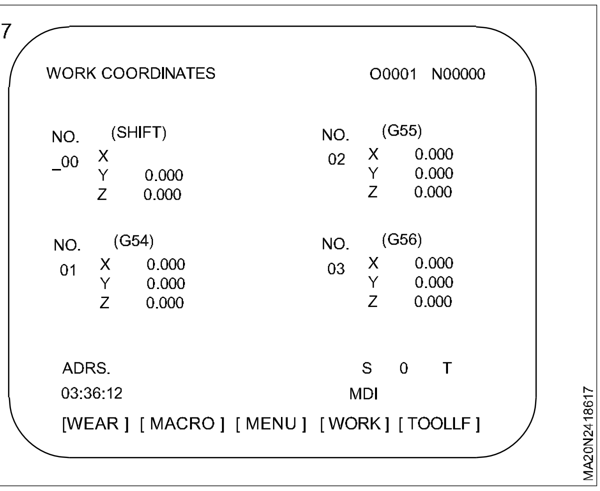
Steps for Displaying and Setting the Workpiece Origin Offset Value

1 Press function key Menu OFFSET

2 Press soft key **[WORK]**

The workpiece coordinate system setting screen is displayed.

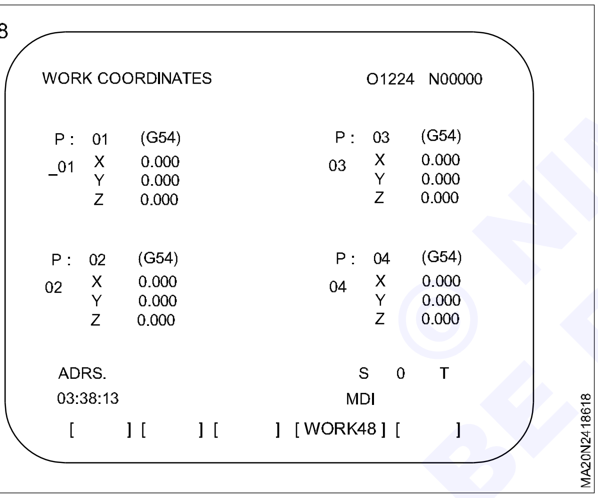
Fig 7



```
WORK COORDINATES                     O0001   N00000

NO.    (SHIFT)                 NO.    (G55)
_00   X                        02    X    0.000
      Y     0.000                    Y    0.000
      Z     0.000                    Z    0.000

NO.    (G54)                   NO.    (G56)
01    X     0.000              03    X    0.000
      Y     0.000                    Y    0.000
      Z     0.000                    Z    0.000

ADRS.                                S    0    T
03:36:12                             MDI
[WEAR ]  [ MACRO ]  [ MENU ]  [ WORK ]  [ TOOLLF ]
```

MA20N2418617

Fig 8



```
WORK COORDINATES                     O1224   N00000

P :  01    (G54)               P :  03    (G54)
_01   X     0.000              03    X    0.000
      Y     0.000                    Y    0.000
      Z     0.000                    Z    0.000

P :  02    (G54)               P :  04    (G54)
02    X     0.000              04    X    0.000
      Y     0.000                    Y    0.000
      Z     0.000                    Z    0.000

ADRS.                                S    0    T
03:38:13                             MDI
[        ] [        ] [        ] [ WORK48 ] [        ]
```

MA20N2418618

3 The screen for displaying the workpiece origin offset values consists of two or more pages. Display a desired page in either of the following two ways:

Press the page up or page down key.

Press NO key and enter the workpiece coordinate system number (0: external workpiece origin offset, 1 to 6: workpiece coordinate systems G54 to G59, 1 to 48: workpiece coordinate systems G54 P1 to G54 P48), and then press IN PUT key

4 Move the cursor to the workpiece origin offset to be changed.

5 Press the address key corresponding to the desired axis.

6 Enter a desired value by pressing numeric keys, then press IN PUT key.

The entered value is specified in the the workpiece origin offset value.

7 Repeat 4 and 6 to change other offset values.

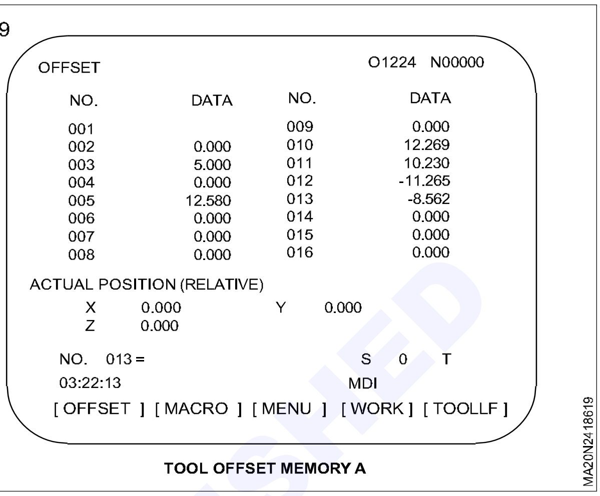
Step for displaying and setting custom macro common variables

1 Press function key

2 Press soft key [OFFSET].

The screen varies according to the type of tool offset memory. (Figs 9 & 10)

Fig 9

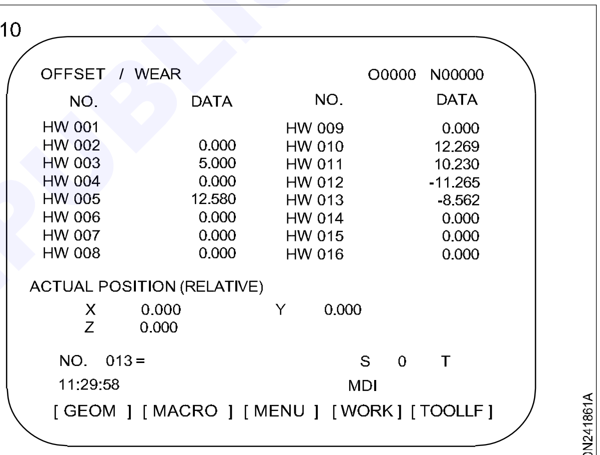


```
OFFSET                                  O1224   N00000
   NO.          DATA          NO.          DATA
   001                        009          0.000
   002          0.000         010         12.269
   003          5.000         011         10.230
   004          0.000         012        -11.265
   005         12.580         013         -8.562
   006          0.000         014          0.000
   007          0.000         015          0.000
   008          0.000         016          0.000
ACTUAL POSITION (RELATIVE)
      X     0.000            Y      0.000
      Z     0.000
   NO.   013 =                       S    0    T
   03:22:13                          MDI
 [ OFFSET ]  [ MACRO ]  [ MENU ]  [ WORK ]  [ TOOLLF ]
```

MA20N2418619

TOOL OFFSET MEMORY A

Fig 10



```
OFFSET  /  WEAR                         O0000   N00000
   NO.          DATA          NO.          DATA
HW 001                        HW 009       0.000
HW 002          0.000         HW 010      12.269
HW 003          5.000         HW 011      10.230
HW 004          0.000         HW 012     -11.265
HW 005         12.580         HW 013      -8.562
HW 006          0.000         HW 014       0.000
HW 007          0.000         HW 015       0.000
HW 008          0.000         HW 016       0.000
ACTUAL POSITION (RELATIVE)
      X     0.000            Y      0.000
      Z     0.000
   NO.   013 =                       S    0    T
   11:29:58                          MDI
 [ GEOM ]  [ MACRO ]  [ MENU ]  [ WORK ]  [ TOOLLF ]
```

MA20N241861A

TOOL OFFSET MEMORY A

3 Move the cursor to the compensation value to be set or changed using No page keys and cursor keys. Press key and enter the compensation number for the compensation value to be set or changed and then press IN PUT key.

4 Enter a compensation value, and key.

Steps for entering programs in MDI mode

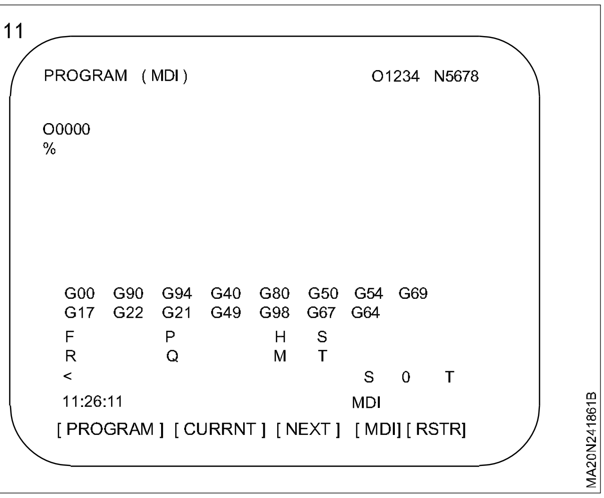
1 Press the MDI mode selection switch

2 Press the PRGM function key on the CRT/MDI panel to select the program screen. The following screen appears: (Fig 11)

Program number O0000 is entered automatically.

3 Prepare a program to be executed by an operation similar to normal program editing. M99 specified in the last block can return control to the beginning of the program after operation ends. Word insertion, modification, deletion, word search, address search, and program search are available for programs created in the MDI mode.

Fig 11



```
PROGRAM  ( MDI )                          O1234   N5678

O0000
%

  G00   G90   G94   G40   G80   G50   G54   G69
  G17   G22   G21   G49   G98   G67   G64
  F           P           H     S
  R           Q           M     T
  <                                   S     0     T
  11:26:11                          MDI
[ PROGRAM ]  [ CURRNT ]  [ NEXT ]   [ MDI] [ RSTR]
```

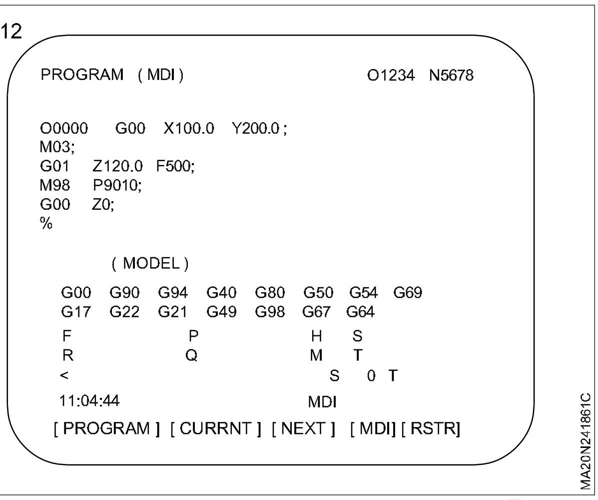
4 To entirely erase a program created in MDI mode,use one of the following methods:

- a Enter address, O then press the key on the MDI panel.
- b Alternatively, press the RESET Key. In case bit 7 of parameter 057 to 1 in advance.

5 To execute a program, set the cursor on the head of the program. (Start from an intermediate point is possible). Push Cycle Start button on the operator's panel. By this action, the prepared program will start. When the program end (M02, M30) or ER (%) is executed, the prepared program will be automatically erased and the operation will end.

By command of M99, control returns to the head of the prepared program. (Fig 12)

Fig 12



```
PROGRAM  ( MDI )                          O1234   N5678

O0000     G00    X100.0    Y200.0 ;
M03;
G01    Z120.0   F500;
M98    P9010;
G00    Z0;
%

          ( MODEL )
  G00   G90   G94   G40   G80   G50   G54   G69
  G17   G22   G21   G49   G98   G67   G64
  F                 P                H     S
  R                 Q                M     T
  <                                     S     0  T
  11:04:44                           MDI
[ PROGRAM ]  [ CURRNT ]  [ NEXT ]   [ MDI] [ RSTR]
```

6 To stop or terminate MDI operation in midway through, follow the steps below.

a Stopping MDI operation

Press the feed hold switch on the machine operator's panel. The feed hold lamp goes on and the cycle start lamp goes off.

When the cycle start switch on the machine operator's panel is pressed, machine operation restarts.

b Terminating MDI operation

Press the RESET key on the CRT/MDI panel.

Automatic operation is terminated and the reset state is entered. When a reset is applied during movement, movement decelerates then stops.

Uses of important switches & buttons and controls

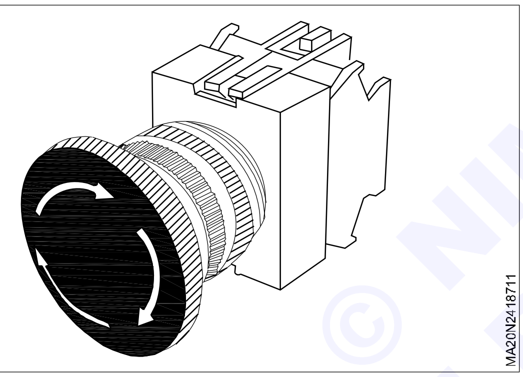
Objectives: At the end of this lesson you shall be able to
- **state the uses of emergency stop**
- **state the purpose of reset button**
- **state the uses of override of feed rate & spindle speed over ride**
- **list the functions of edit, lock and ON/OFF buttons.**

Emergency stop push button

Emergency stop buttons are designed in such a manner in which their role is more physical, such as interrupting a power supply to the machine control system. It is a basic big red pushbuttons fixed on Machine control panel.

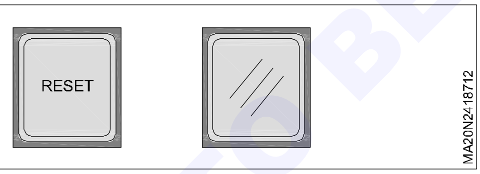
Emergency stop pushbutton that has mechanical plastic or metal tabs and grooves internally such that when you push it (interrupting the circuit), it is held in that position untill you twist it. They are designed to be large, hard to miss, and easy to push, sample is given in Fig 1.

Fig 1



Reset key Fig 2

Fig 2



Press this MDI key board key to reset the CNC.

Press this key to cancel an alarm etc.

Program data override

All CNC units are designed with a number of special rotary switches that share one common feature - they allow the CNC operator to override programmed spindle speed or programmed speed of an axis motion. For example, a 100mm/mm feed rate in the program produces a slight chatter. A knowledgeable operator will know that by increasing the feed rate or decreasing the spindle speed, the chatter may be eliminated. It is possible to change the feed rate or the spindle speed by editing the program, but this method is not very efficient.

A certain 'experimentation' may be necessary during actual cut to find the optimum setting value. Manual override switches come to the rescue, because they can be used by trial during operation. There are four override switches found on most control panels: Rapid feed rate over ride(rapid traverse) (modifies the rapid motion of a machine tool) Spindle speed override (modifies the programmed spindle r/min) Feed rate override(cutting feed rate) (modifies the programmed feed rate) Dry run mode (changes cutting motions to a variable speed)

Override switches can be used individually or together. They are available on the control to make work easier for both the operator and the programmer. Operator does not need to 'experiment' with speeds and feeds by constantly editing the program and the programmer has a certain latitude in setting reasonable values for cutting feed rates and spindle speed. The presence of override switches is not a license to program unreasonable cutting values. Overrides are fine tuning tools only - part program must always reflect the machining conditions of the work. Usage of override switches does not make any program changes, but gives the CNC operator an opportunity to edit the program later to reflect all optimum cutting conditions. Used properly, override switches can save a great amount of valuable programming time as well as setup time at CNC machine.

Rapid Motion Override

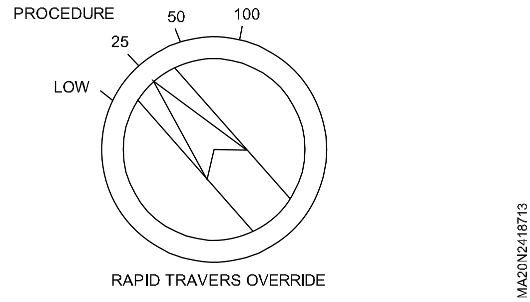
Rapid motions are selected in the CNC program by a preparatory command without a specified feed rate. If a machine is designed to move at 985 in/min (25000 mm/min) in the rapid mode, this rate will never appear in the program. Instead, the rapid motion mode is called by programming a special preparatory command G00. During program processing, all motions in G00 mode will be at the manufacturer's fixed rate. The same program will run faster on a machine with a high rapid motion rating then on a machine with a low rapid motion rating.

During setup, the rapid motion rate may require some control for program proving, when very high rapid rates are uncomfortable to work with. After a program had been proven, rapid rate can be applied at its maximum. CNC machines are equipped with a rapid override switch to allow temporary rapid motion settings. Located on the machine control panel, this switch can be set to one of four settings. Three of them are usually marked as the percentage of the maximum rate, typically as 100%, 50% and 25%. By switching to one of them, the rapid motion rate changes. For example, if the maximum rapid rate is 985 in/min or 25000 mm/min, the actual reduced rates are 493 in/min or 12500 mm/min at the 50% setting and 246 in/min or 6250 mm/min at the 25% setting. Each of the reduced rates is more comfortable to work with during setup.

1 **Steps for rapid traverse override**

- Select one of the four feedrates with the rapid traverse override switch Fig 3 during repid traverse. Refer to the appropriate manual provided by the machine tool builder for rapid traverse override.

Fig 3



- The following types of rapid traverse are available. Rapid traverse override can be applied for each of them.

1 Rapid traverse by G00

2 Rapid traverse during a canned cycle.

3 Rapid traverse in G27, G28 and G30.

4 Manual rapid traverse.

5 Rapid in manual reference position return.

Spindle Speed Override

The same logic used for the application of rapid rate override can be used for the spindle speed over ride. The required change can be established during actual cutting by using the spindle speed override switch, located on the machine control panel. For example, if the programmed spindle speed of 1000 r/min is too high or too low, it may be changed temporarily by the switch. During actual cutting, the CNC operator may experiment with the spindle speed overrides witch to find the optimum speed for given cutting conditions. This method is a much faster than 'experimenting' with the program values.

Spindle speed override switch can be continuous on some controls or selectable in incrementsof10%,typically within the range of 50-120% of the programmed spindle speed. A spindle programmed at 1000 r/min can be overridden during machining to 500, 600, 700, 800, 900, 1000, 1100 and 1200 r/min. This large range allows the CNC operator certain flexibility of optimizing spindle rotation to suit given cutting conditions. There is a catch, however. The optimized spindle speed change may apply to only one tool of them any often used in the program. No CNC operator can be expected to watch for that particular tool and switch the speed up or down when needed. A simple human oversight may ruin the part, the cutting tool or both. The recommended method is to find out the optimum speed for each tool, write it down, then change the program accordingly, so all tools can be used at the 100% spindle override setting for production.

Comparison of the increments on the spindle override switch with the increments on switches for rapid traverse override (described earlier) and the feed rate override (described next), offers much more limited range. The reason for the spindle speed range of 50% to 120% is safety. To illustrate with a rather exaggerated example, no operator would want to mill, drill or cut any material at 0 r/min (no spindle rotation),possibly combined with a heavy feed rate.

In order to change the selected override setting into 100% speed in the program, a new spindle speed has to be calculated. If a programmed spindle speed of 1200r/min for a tool is always set to 80%, it should be edited in the program to 960r min, then used at 100%. The formula is quite simple.

Feed rate Override

Probably the most commonly used override switch is one that changes programmed feed rates. For milling controls, the feed rate is programmed in in/min or m/min. For lathe controls, the feed rate is programmed in in/rev or in mm/rev. Feed rate per minute on lathes is used only in cases when the spindle is not rotating and the feed rate needs to be controlled.

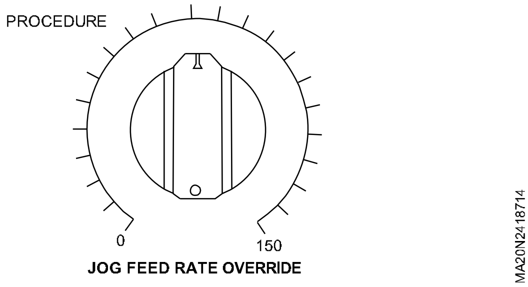
Feed rate can be over ridden with in a large range, typically from 0% to 200% or at least 0% to 150%. When the feed rate override switch is set to 0%, the CNC machine will stop the cutting motion. Some CNC machines do not have the 0% percent setting and start at 10%. This can be change by a system parameter. The maximum of 150% or 200% cutting feed rate will cut 1.5 or 2 faster than the programmed feed rate amount.

There are situations, where the use of a feed rate override would damage the part or the cutting tool - or both. Typical examples are various tapping cycles and single point threading. These operations require spindle rotation synchronized with the feed rate. In such cases, the feed rate override will become ineffective. Feed rate override will be effective, if standard motion commands G00 and G01 are used to program any tapping or thread cutting motions. Single point threading command G32, tapping fixed cycles G74 and G84, as well as lathe threading cycles G92 and G76 have feed rate override cancellation built into the software. Overriding the programmed spindle speed on CNC machines should have only one purpose - to establish spindle speed rotation for the best cutting conditions.

2 **Steps for feedrate override**

- Set the feedrate override dial (Fig 4) to the desired percentage (%) on the machine operator's panel, before or during automatic operation.

Fig 4



- On some machines, the same dial is used for the feedrate override dial and jog feedrate dial. Refer to the appropriate manual provided by the machine tool builder for feedrate override.

Override range

- The override that can be specified ranges from 0 to 150% (10% steps). For individual machines, the range depends on the specifications of the machine tool builder.

Override during thread

- During threading, the override is ignored and the feedrate remains as specified by program.

Machine lock and auxiliary function lock

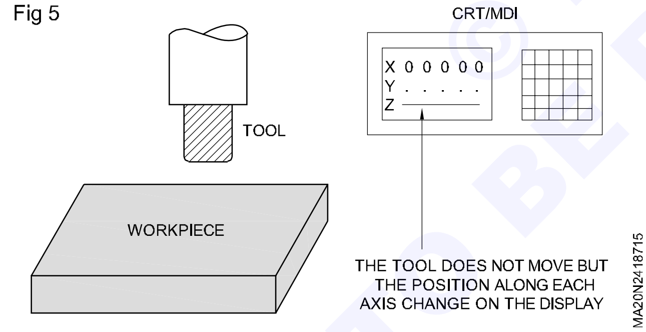
To display the change in the position without moving the tool, use machine lock. There are two types of machine lock: all–axis machine lock, which stops the movement along all axes, and Z–axis machine lock, which stops the movement along Z–axis only. In addition, auxiliary function lock, which disables M, S, and T commands, is available for checking a program together with machine lock.

Steps for Machine Lock and Auxiliary Function Lock

Machine Lock

- Press the machine lock switch on the operator's panel. The tool does not move but the position along each axis changes on the display as if the tool were moving (Fig 5).

Fig 5



- Some machines have a Z–axis machine lock switch. Refer to the appropriate manual provided by the machine tool builder for machine lock.
- The position relationship between the workpiece coordinates and machine coordinates may change after automatic operation by the machine lock function has been executed. If this occurs, reset the workpiece coordinate system by specifying the coordinate system setting command or by making a manual reference position return.

Auxiliary Function Lock

- Press the auxiliary function lock switch on the operator's panel. M, S, and T codes are disabled and not executed. Refer to the appropriate manual provided by the machine tool builder for auxiliary function lock.

M, S, T command by only machine lock

- M, S, and T commands are executed in the machine lock state.

Reference position return under Machine Lock

- When a G27, G28, or G30 command is issued in the machine lock state, the command is accepted but the tool does not move to the reference position and the reference position return LED does not go on.

M codes not locked by auxiliary function lock

- M00, M01, M02, M30, M98, M99, M198, and M199 and commands are executed even in the auxiliary function lock state.

Proving the CNC program

Objectives: At the end of this lesson you shall be able to
- **brief the steps in proving the CNC part program**
- **describe the tool path simulation and its steps**
- **state the dry run and explain the steps**
- **execute the program in single block mode.**

Running the first part

CNC machine operator usually starts a new job by studying the documentation included with the Program mainly drawing, setup sheet and tooling sheet.

The following steps describe a standard setup procedures, that will remain the same for most jobs.

Set the cutting Tool

Use a tooling sheet or tooling information from the part program

Set the cutting tools into their holders and respective tool station. Register all tool numbering into the control memory. Make sure the tools are sharp and mounted properly in the holders cutting tool check list.

- Are the tools are properly mounted in holder
- Are the proper inserts are used
- Are all the tools are right size
- Are the tools are placed in proper magazine station
- Are the offsets set correctly
- Is there an interference between individual tools
- Are all the tools sharp

Set the fixture/work holding device

A fixture that holds or support the part mounted on the machine,aligned and adjusted if necessary, but the part is not mounted at this stage.

Set the part: Locate the part into the fixture and make sure it is safely mounted. Check for possible interferences and obstacles in the setup.

Part checkup check list

- Is the part mounted safely
- Is the part properly oriented on the table
- Is the part aligned squarely
- Are the clearance sufficient
- Are all the clamps away from the cutting path
- Does the tool change takes place in clear area

Set the tool offsets: Depending on the types of machine, get the tool geometry and wear offsets, tool length offsets and cutter radius offset. This setup includes the work offset setting.

Checking the part program

Any unproved program is a potential source of problem. In manual CNC programming, mistakes are common than in CAO/CAM programs

A good way to look at a new program is through the machine operator's eyes. The most important thing is consistency in programming approach. For example.

- Al the tool approach clearances the same way as always
- Basic programming format maintained from one program to another program
- Scans the written program twice once on paper copy.
- Common mistakes such as a missing minus sign or an address, a misplaced decimal points
- Double check the program

Check the program through control graphic display that is tool path simulation. It is fast and accurate, and offers of confidence before actual machining

The tool path simulation shows the outline of finished part and the tool motions . The tool motions are identified by a dashed line(rapid motion) and a solid line(cutting motion).

Dry run operation

The program can be executed much faster than using a feed rate override at the maximum setting. no actual machining takes place when dry run switch is in effect.

During dry run, the work place is not mounted in the machine. If the work piece is mounted it is very much important to provide sufficient clearance by shifting the work off set in Z axis. Then the program is processed dry without actual cutting without a coolant just in air.

Dry run is very efficient setup to prove overall integrity or a CNC program.

Reset the part

On successful completion of all previous steps on successful completion in the fixture, check the tooling once more, also clamps offsets and any other important machine features.

Steps for Dry run a program

Press the dry run switch on the machine operator's panel during automatic operation. The tool moves at the feed rate specified in a parameter the Rapid traverse switch can also be used for changing the feed rate

Single block operation

Pressing the single block switch starts the single block mode. When the cycle start buttons is pressed in the single block mod, The tool steps after a single block in the program executed. Check the program in the single block mode by executing program block by block

Steps for single operation

Press the single block switch on the machine operator's panel. The execution of the program is stopped after the current block is executed

Press the cycle start button to execute the next block. The movement stops after the block is executed.

While execute the program in single block the feed override knob should be set to zero.

The block contain the movement the slide will not make but the distance to go is appeared on the screen.

Read the values of the distance in X,Y,Z axes compare the distance to go and the actual work movement. Judge manually their will not be any collision then release the feed hold or increase in feed rate knob.

Similarly execute the program block by block

Make a Trail cut

Trail cut may be required in order to establish whether the programmed speeds and feeds are reasonable or not and if the various offsets are set properly.

At this point, any necessary adjustments are finalized before production begins.

The ideal way to run a new program is to run it through the control graphic display. It is fast and accurate. Test can be done with a variety of override in effect, for example, dry run single block

Tool offset adjustment on first part

Objectives: At the end of this lesson you shall be able to
- **correct the oversizing of external dimension by adjustung tool offset**
- **correct the undersizing of external dimension by adjustung tool offset.**

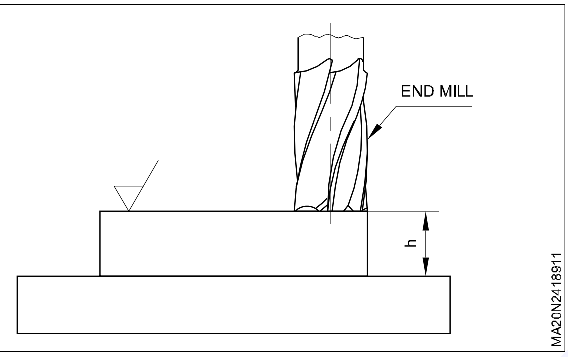
Correcting tool offset

When a programme is executed it is quite possible that the actual dimension achieved may be different from desired value and it may be due to error may be corrected as described below in the following cases.

CASE 1

Height dimension (h) (Fig 1)

Fig 1



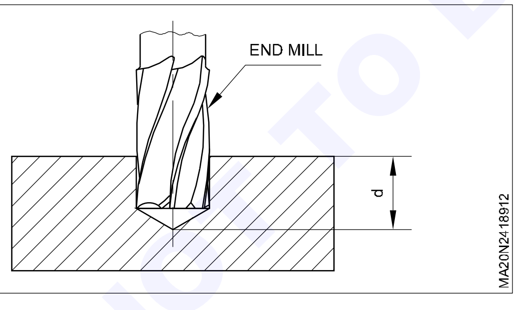
If 'h' is more than the desired value reduce the tool offset the difference of measured value and desired value.

If 'h' is less the desired value increase the tool offset (height) by the difference between the desired value and measured value.

CASE 2

Depth dimension (d) (Fig 2)

Fig 2



If 'd' is more than the desired value increase the tool offset by the difference of measured depth and desired length.

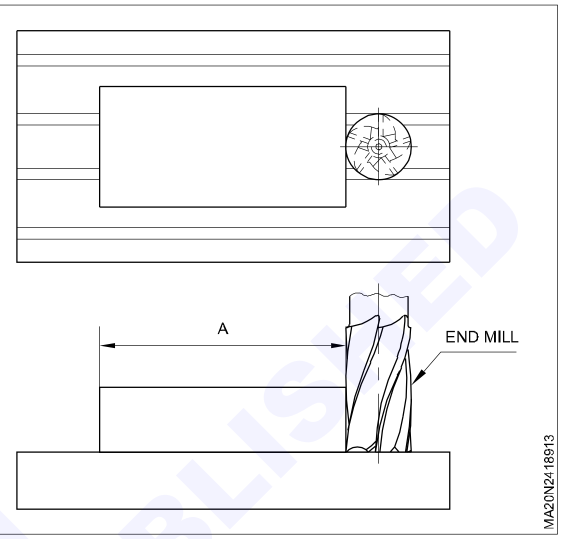
If 'd' is less than the desired value reduced the tool offset by the difference between the desired value and the measured value.

CASE 3

Dimension of external profile

If A (Fig 3) is more than the desired value reduce the tool offset (radius) y half the difference between measured value and desired value.

Fig 3



If A is less than the desired value increase the tool offset (radius) by half the difference between the desired value and measured value.

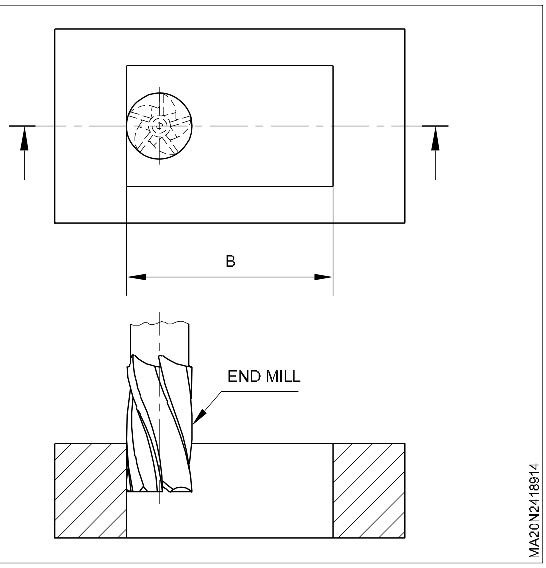
CASE 4

Dimension of internal profile

If B is more than the desired value increase the tool offset by half the difference between the measured value and desired value (Fig 4).

If B is less than the desired value reduce the tool offset (radius) by half the difference between desired value and measured value.

Fig 4



Axes over travel recovering from over travel

Objectives: At the end of this lesson you shall be able to
- **list the two types of over travel alarms**
- **describe the steps involved in recovering from the over travels on both the types.**

There are two kinds of over travel alarms.

1 The alarm generated by the OT switches.

2 The software over travel which only happens on Fanucs that have memory on their axis position .

Recovering alarm generated by the over travel (OT) alarm happens and can only jog that axis in one direction (away from the switch)

Recovering from software over travel

Fanuc control have either absolute encoders on the motors, or encoders with battery backup these control can continue from their position enter if the power is turned OFF. On power up, control of the axes over travel can it a software over travel even before a zero return on that axis.

The Fanuc older controls are without axis position memory while they are until zero return after power up on these control zero return function works like

- Jog at 25 to 50 mm away from the home position
- Turn on the zero return switch and jog towards the home position
- the axis moves in rapid until it hits zero return switch
- The axis slows down and continues to move until the cam switch drops off again

Recovering from battery operated encoders

Depending up on the axis and direction of over travel one can use a simple procedure to disable

- Turn off the control
- Press the (p) key and the (CAN) key simultaneously in the key board.
- Do not release those keys and turn on the control
- Hold the key until the control turns on completely
- Release the keys
- Try to send axis home, be careful since this procedure disables the software limits and you just have the limit switches to stop the machine at the end of it travel
- If it goes home with no problem turn off and then again to set the software limit active again.

Collision due to improper machine setup, operation and effects

Objectives: At the end of this lesson you shall be able to
- **state causes of CNC machine tool crashes**
- **brief how crashes affect 3-axis vertical machining centres**
- **explain how crashes affect 5-axis CNC machine tools**
- **describe the method of recalibrating machine tools**
- **explain the steps involved in repairing CNC machine tools.**

A crash in a CNCMachining Centre.can be disastrous to your manufacturing productivity and very costly to repair. But what can you do when it happens to Machining Center?

Causes of CNC Machine Tool Crashes

Available in many variations and configurations, 5-Axis machining centres have multiple moving parts. However, the source of these crashes is often due to human errors rather than machine errors.

These are some of the common causes of crashes:

- Setting of wrong tool inside the Automatic Tool Changer (ATC) tool pocket
- Input of wrong tool length when they are exchanged or newly set-up
- Setting the wrong feed rate (normally too fast for the job)
- Choosing the wrong tool for the job
- Setting the job wrongly
- Wrong reference point set
- Errors in CNC programming
- Poor maintenance of CNC machine

To minimise these and other common CNC machine problems, you should ensure that your machine operators and programmers are adequately trained. This is also important to avoid any injury to the operators.

Your machine tool operators and programmers should also be alert and experienced in how to troubleshoot crashes when they occur. Thus, careful testing before the start of your production is highly recommended.

3-Axis Vertical Machine Crashes

In a 3-axis vertical machining centre a crash normally happens in the vertical axis or Z-axis. The impact happens straight down and affects mainly the spindle – particularly the first row of spindle bearings which take the main force of such an impact.

If the spindle is not damaged you should check for Z-axis alignment and spindle run-out, especially if there is a higher volume of noise heard when the machine operates at a higher RPM. You may also need to re-align the spindle head or frame if both settings move out of their original tolerance levels.

Regardless of the source of impact – be it vertical or side-wise – you'll need to re-align the spindle head. In the case of vertical impact, the X and Y axes are usually not affected. Should the crash be severe, these axes may run out of their original tolerances.

The Complications in a 5-Axis CNC Machine Tool Crash

If the same impact scenario occurs on a 5-axis configured machining centre, however, the likelihood that more than one axis needs to be adjusted by service is high.

In a machine with a built-in two axis rotary table, a straight down impact in the Z-axis will not only affect the original Z-axis but the two rotational axes C and B. This is especially so if the rotational axis at the time of impact is positioned in a certain angle/degree.

Should the impact be more severe, even the Y and X axes can be affected. This will require a greater amount of rectification.

Machine Recalibration Machining Centre Crash

A re-calibration is often the minimum measure needed to ensure that all axis are in perfect alignment to each other after the incident. This will also reduce the possibility of misalignments, especially at the higher tolerances, when the part is being machined during production.

However, a simple calibration is often not enough after a crash. It is better that your operator checks all the different geometrical dimensions first before calibrating the machine.

Repairing a CNC Machine Tool Crash

Repairs of machining centres tend to take more time. Repair costs are higher too, due to the need to carefully realign each axis and set them properly. Such work requires skilled engineers using highly precise tools

It is strongly advised therefore that such work should be done by a well-trained factory service engineer who knows the intricacies of the machine and how to set it correctly. In our experience, we have seen customers trying to cut cost by hiring inexperienced engineers to do the repair job. Often, the end result is a slipshod repair work which results in more issues. The end result is that they up pay more over the long-term when they try to cut costs.

Recovering from collisions

Objective: At the end of this lesson you shall be able to
- **describe the various methods of retract and recover from collision in Z axis.**

Generally in any CNC machine the Z axis, that is the spindle with or without tool is hit with work piece or machine table thereby spoiling tool, spindle, workpiece and machine alignments. It should be corrected. Before repairing it, the tool and the spindle to be withdrawn from that position.

Tool retract and recover

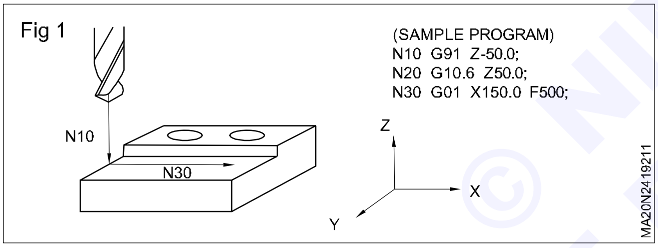
The tool can be retracted from a work piece to replace the tool, if damaged during machining, or to check the status of machining. Then, the tool can be returned to restart machining efficiently.

Steps for tool retract and recover

Step 1: Programming

Specify a retraction axis and distance in command G10.6IP_ beforehand. In the sample program below, the N20 block specifies that the Z-axis is the retraction axis and the retraction distance is to be 50 mm (Fig 1).

Fig 1

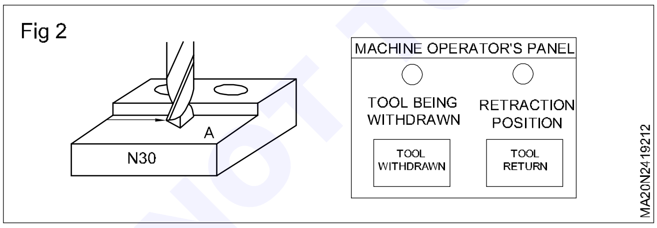


```
(SAMPLE PROGRAM)
N10  G91  Z-50.0;
N20  G10.6  Z50.0;
N30  G01  X150.0  F500;
```

Step 2: Retract

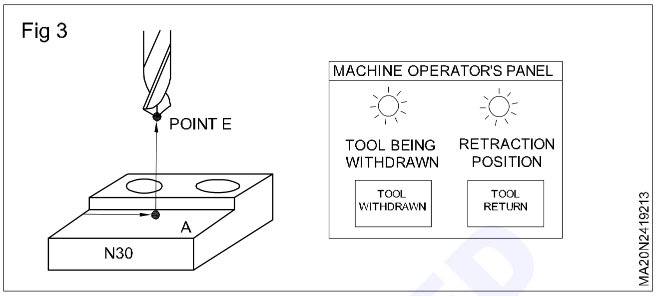
Suppose that the TOOL WITHDRAW switch on the machine operator's panel is turned on when the tool is positioned at point A during execution of the N30 block. (Fig 2)

Fig 2



Next, the tool withdrawal mode is set and the TOOL BEING WITHDRAWN LED goes on. At this time, automatic operation is temporarily halted. The tool is then retracted by the programmed distance. If point A is the end point of the block, retraction is performed after automatic operation is stopped. Retraction is based on linear interpolation. The dry run feedrate is used for retraction. Upon completion of retraction, the RETRACT POSITION LED on the operator's panel goes on. (Fig 3)

Fig 3



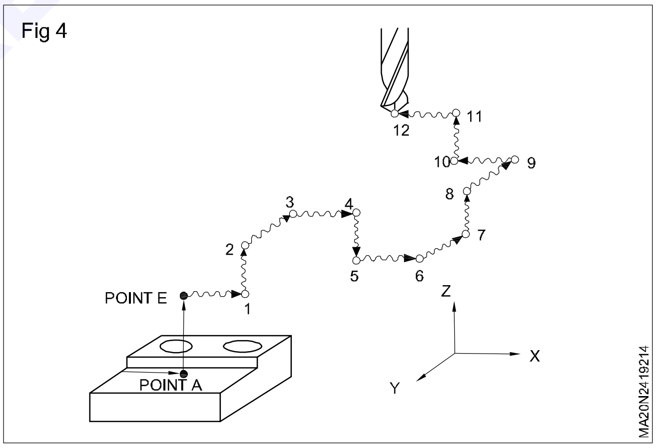
During retraction, the LCD screen displays PTRR and STRT.

- PTRR blinks in the field for indicating states such as the program editing status. - STRT is displayed in the automatic operation status field. - MTN is displayed in the field for indicating status such as movement along an axis.

Step 3: Withdrawal

Set the manual operation mode, then withdraw the tool. For manual operation, either jog feed incremental feed, handle feed, or manual numerical command is possible. (Fig 4)

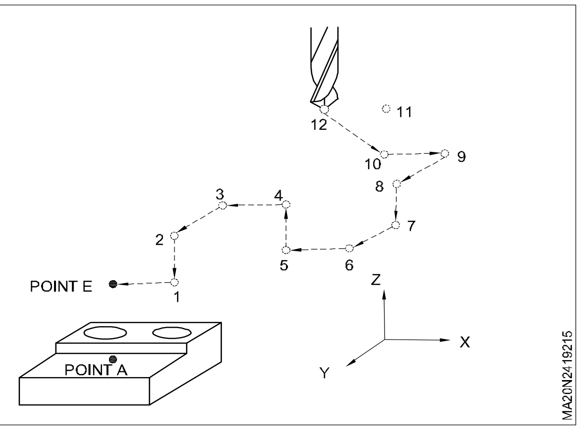
Fig 4



Step 4: Return (Fig 5)

After withdrawing the tool and any additional operation such as replacing the tool, move the tool back to the previous retraction position. To return the tool to the retraction position, return the mode to automatic operation mode, then turn the TOOL RETURN switch on the operator's panel on then off again. The tool returns to the retraction position at the dry run feedrate, regardless of whether the dry run switch is on or off. When the tool has returned to the retraction position, the RETRACTION POSITION LED comes on.

Fig 5



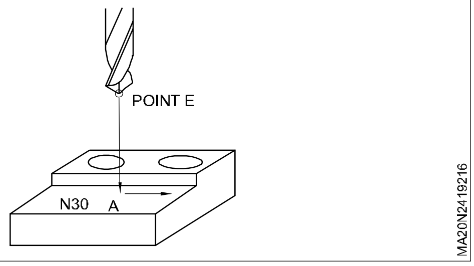
During return operation, the LCD screen displays PTRR and MSTR.

- PTRR blinks in the field for indicating states such as program editing status. - MSTR is displayed in the automatic operation status field. - MTN is displayed in the field for indicating states such as movement along an axis.

Step 5

Repositioning While the tool is at the retraction position (point E in the figure below) and the RETRACTION POSITION LED is on, press the cycle start switch. The tool is then repositioned at the point where retraction was started (i.e. where the TOOL WITHDRAW switch was turned on). Repositioning is based on linear interpolation. The dry run feedrate is used for repositioning. (Fig 6)

Fig 6



Limitation

1 If the origin, presetting, workpiece origin offset value (or external workpiece origin offset value), or workpiece coordinate shift amount (for a lathe system) is changed after the retraction position is specified with G10.6 in the absolute mode, the change is not reflected in the retraction position. After such changes are made or the work piece origin offset value (or external workpiece origin offset value) or work piece coordinate shift amount (for a lathe system) is changed, respectively the retraction position with G10.6.
2 When retracting the tool manually in the tool withdrawal mode, do not use the machine lock, mirror-image, or scaling function.

Warning

The retraction axis and retraction distance specified in G10.6 must be changed in an appropriate block according to the figure being machined. Be very careful when specifying the retraction distance; an incorrect retraction distance may damage the work piece, machine, or tool.

Helical interpolation and thread milling

Objectives: At the end of this lesson you shall be able to
- **state the helical interpolation**
- **brief the importance of thread milling**
- **state the advantage and limitation in CNC machines.**

Helical interpolation

Helical interpolation or helical milling is in a form of circular interpolation. It is a programming technique to machine arcs and circle combined with a linear interpolation in the same block, during the same motion.

Helical interpolation is a simultaneous two-axis circular motion in the work plane, with the linear motion along the remaining axis.

The resulting three axis motion in always synchronized by the control system and the axes reach the target location at the same time.

Plane selection programmed before helical interpolation block determines which axes will be active in the program and what their function will be.

Helical interpolation are programmed using the same principles as in circular interpolation but will be different for each plane. Listed in table 1.

Table 1

Active Plane	Circular Motion	Linear Motion	Arc Vector
G17	X and Y	Z	I and J
G18	X and Z	Y	I and K
G19	Y and Z	X	J and K

Application and usage

Helical interpolation is not most frequently used programming method. It may be the only method available for a special machining applications.

Thread milling, Helical profiling and helical ramping from these three group thread milling is the most common method of helical interpolation applied in industry.

Thread milling

There are two methods of producing a thread on a CNC machine. On machining centers, method of thread generating is tapping using fixed cycles, but most of threads are machined by single point threading method.

In many cases in manufacturing, either tapping or single point threading method is impossible in a given situation. Many of these difficulties can often overcome by thread milling method.

Importanceof thread milling/Advantages of thread milling

Thread milling can be used on programming to achieve special benefits. The benefits are;

- A large thread diameter - virtually any diameter can be thread milled.
- Smoother and more accurate thread generation.
- Combination of thread milling within a single setup eliminates secondary operations.
- Full depth thread can be cut.
- Tap is not available.
- Tapping is impossible in hard materials.
- Part cannot rotated on a CNC lathe.
- Left hand and right hand threading has to be done with one tool.
- External and internal threading has to be done with one tool.
- High surface quality in soft materials.
- Elemination of expensive large taps.
- No need to spindle reversal as in tapping.
- One tool holder can accept inserts for different thread pitch sizes.
- Reduction of over all threading costs.

Thread milling only enhances other threading operations in CNC it uses special multiple condition for thread cutters.

Condition for thread milling

For successful thread milling, three condition must exist before writing a program.

1 Control system must support the operation.

2 Diameter to be threaded must be premachined.

3 Suitable thread milling tool must be selected.

All the above conditions must exist simultaneously.

Thread milling tool

Thread milling cutters are available in two varieties.

1 Made up of solid carbides. (Fig 1)

2 Carbide inter changeable inserts. (Fig 2)

Fig 1

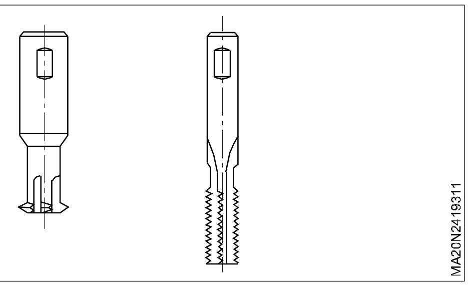
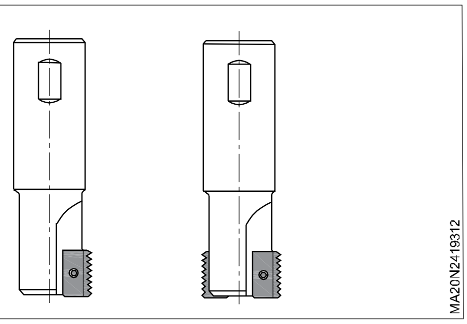
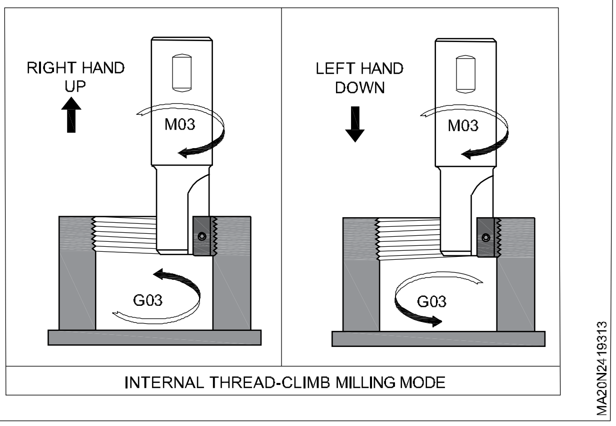


Fig 2



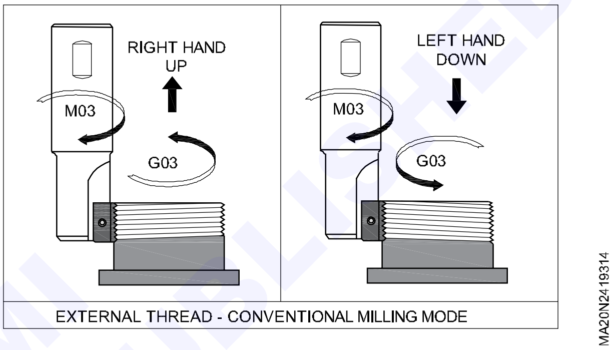
Thread milling tool path for internal and external are shown in Figs 3 & 4.

Fig 3



INTERNAL THREAD-CLIMB MILLING MODE

Fig 4



EXTERNAL THREAD - CONVENTIONAL MILLING MODE

Tool wear and necessity of wear offset

Objectives: At the end of this lesson you shall be able to
- **describe tool wear and affects**
- **explain the necessity of wear offset**
- **brief the step to enter the wear offset in machine control panel.**

Tool wear

Cutting tools are subjected to an extremely severe rubbing process. They are in metal-to-metal contact between the chip and work piece, under high stress and temperature.

Tool wear is generally a gradual process due to regular operation. Tool wear can be compare with the wear of the tip of an ordinary pencil.

Tool wear affects following items

1 Increased cutting forces.

2 Increased cutting temperature.

3 Decreased accuracy of parts.

4 Decreased tool life.

5 Poor surface finish.

6 Economics of cutting operations.

Decreased accuracy of parts mainly depend upon the tool geometry in 'Z' and 'XY' direction, that is tool length offset in 'Z' direction and the cutter radius in 'X' and 'Y' direction.

If you enter the correct tool length offset in geometry, the part size will be equal to the programmed value, that is of the programmer values is say Z - 10.00, the part size also be Z - 10.00 ± that dependent upon the machine accuracy.

If any variation found in the part, that should be adjusted in tool length and tool radius (geometry). This way, the production run will start with the geometry offset set perfectly and with the wear offset set at zero.

As the production run continues and the finish tool show sign of wear, additional adjustments must be made. After so work piece, for example the part dimension by finishing tool (external dimensions) may be larger. Accordingly, adjust the wear in the tool wear offset.

If the dimension are positive, negative is the wear amount.

If the dimension are negative, positive is the wear amount.

Step to enter the wear offset.

In latest control. The tool length offset, redius and wear offsets are provided in geometry offset page. Enter the corresponding wear amount in respective tool number.

Restarting machine from sudden stoppage

Objective: At the end of this lesson you shall be able to
- **describe the effects of sudden stoppage of CNC machine due to power shutdown**
- **explain, methods to restart the machine from sudden power shutdown.**

Effects of sudden stoppage of CNC machine

During the machining of the machinery is interrupted due to power failure, the machine stops immediately. It may be in the air movement or cutting movement.

If it is in air movement, no problem, only the program to be restarted.

If it is in actual cutting movement there are chance of tool breakage, tool mark on the work piece etc.,

When the machining is interrupted due to any reason the following operations are required.

1 Removing the causes, which interrupted the machining. In case of tool breakage replace the tool and take tool offset for power failure wait till the power resume.

2 Locating the interrupted point in the program that be locate the block number in which the program was interrupted.

3 Restoring the machine to the suitable status (including the auxiliary functions such as coolant on, tool call status and spindle status.

4 Moving the tool to the position suitable for restarting the machinery.

5 Resume automatic operation from the interrupted block or before several blocks.

Steps in program restarting.

1 Reset the program at the head you can use REWIND if you want

2 Turn on program restart this will be a switch or a soft key depending on the machine control.

3 Input the N number you wish to start at eg N100.

4 Press Q-Type the machine with jump to this line and the following screen wikk show program restart (Fig 1)

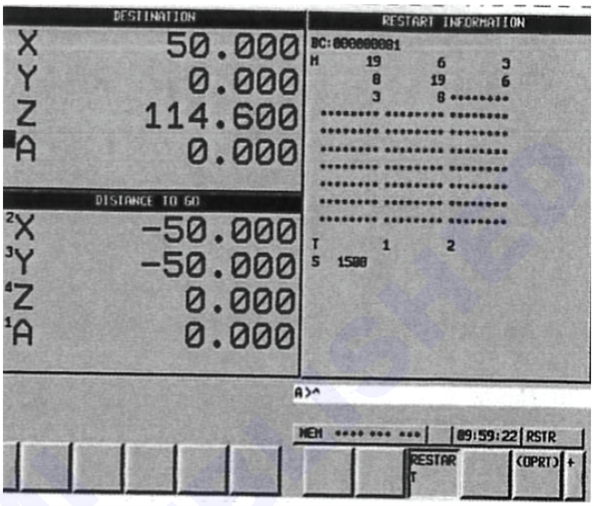
5 Your sequence number is searched for and this screen will show you where your machine is going DESTINATION and the DISTANCE TO GO

6 Turn off program restart. (Distance to go will flash).

7 Go to MDI and get the correct tool then start the spindle at the correct speed issue M8 if you need coolant on. (The tool number and recent M codes are all shown on the screen above).

8 Go back to AUTO or MEMORY mode.

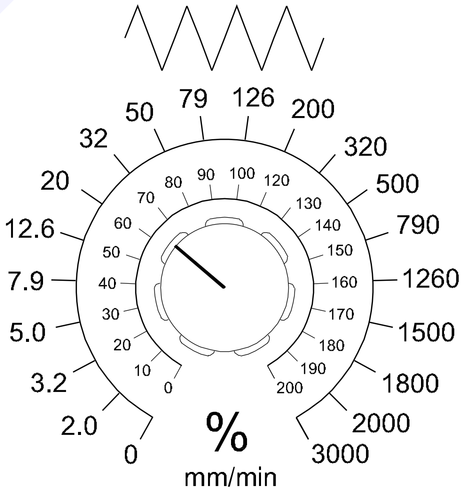
Fig 1



```
DESTINATION                  RESTART INFORMATION
X      50.000                BC:000000001
Y       0.000                M     19       6       3
Z     114.600                       8      19       6
A       0.000                       3       8 ********
                             ******** ******** ********
                             ******** ******** ********
                             ******** ******** ********
                             ******** ******** ********
                             ******** ******** ********
                             ******** ******** ********
DISTANCE TO GO               ******** ******** ********
2X    -50.000                T         1       2
3Y    -50.000                S  1500
4Z      0.000
1A      0.000

                             A>^
                             MEM **** *** ***  | |09:59:22|RSTR
```

9 With the machine in SINGLE BLOCK each time you press cycle start the axis will move to position in the order shown on the screen. Use feed controller to position axis (Fig 2).

Fig 2



10 You are allowed to manually move the axis before continuing if a collision is suspected.

11 Once each axis is in the final position the program runs normally from this point.

Steps Procedure for Quick Program Restart (Auto Mode)

1 Once the part program is interrupted, clear all alarm and move Z axis to reference position

2 Press PROG KEY → Press FOLDER → Press OPRT → Press DEVICE

CHANGE → Select CNC MEM or MEM CARD) → Move cursor to select main program → Press MAIN PROGRM. (Fig 3)

Fig 3

3 In Auto mode, select " in MDI keypad and press RIGHT ARROW

4 Select " and press "

5 Select "SEARCH EXEC" CNC will start to search program restart point and RSTR will blink in bottom right corner After completion of search screen will switch to below indicated screen. (Fig 4)

Fig 4

```
PROGRAM RESTART SETTING                    00002 N04219
RESTART POINT LIST                O0002
                        (1/1)      N4214X-.369Y-4.762-.898;
NO.  PROG.NAME  SEQ.NO.  BLOCK NO. N4215X-.447Y-4.8382-.919;
001  O0002          22        8    N4216Y-4.839;
002  O0002          23        9    N4217X-.55Y-4.9412-.864;
003  O0002          24       10    N4218X-.619Y-5.012-.851;
004  O0002          32       20    N4219X-.662Z-.861;
005  O0002          33       21   ON4220X-.526Y-4.8752-.896;
006  O0002        4220     4208    N4221X-.491Y-4.842-.917;
                                   N4222X-.39Y-4.7392-.893;
                                   N4223X-.269Y-4.6172-.891;
                                   N4224X-.147Y-4.4952-.92;
                                   N4225X-.028Y-4.3772-.976;
RESTART TYPE   : SEARCH
MODAL   :G01   G17   G90   G22   G94   G21   G40   G43   G80   G98   G50
         G67   G97   G54   G64   G69   G15   G40.1 G25   G160  G13.1 G50.1
         G54.2 G80.5 G49.9 G13.3 G54.3 G50.2 G05.5 G54.4 G80.4 G13   G08.9
         G05.7
         F    3000 H       1 D        T          S    7000
         M       8                    B
                                  A>_
                                           FINE  0S120%T00000000
                                  MEM **** *** ***    16:25:43
```

```
PROGRAM RESTART                            00002 N04219
          DESTINATION          RESTART INFORMATION
X           -0.6620       BC:000004193
Y           -5.0100       M      3       8 ********
Z            0.2390        ******** ******** ********
                           ******** ******** ********
                           ******** ******** ********
                           ******** ******** ********
                           ******** ******** ********
                           ******** ******** ********
         DISTANCE TO GO    ******** ******** ********
1X          -0.0305        ******** ******** ********
2Y          -0.0303        ******** ******** ********
3Z           0.0078
                           T  ******** ********
                           S  7000
                           B  *****
                          A>_
                                   FINE  0S120%T00000000
                          MEM **** *** ***    16:26:01 RSTR
```

Don't interrupt/reset during Program Restart operation.

6 Press MDI mode, all previous executed M, S, T codes will be output automatically in the MDI program screen as shown below (Fig 5).

7 Execute the M, S, T codes whichever is needed in MDI mode

8 Select AUTO mode and press CYCLE START] to restart the program.

Please ensure M, S, T, B codes are executed correctly and axis moved to restart point correctly and execute the Cycle.

Fig 5

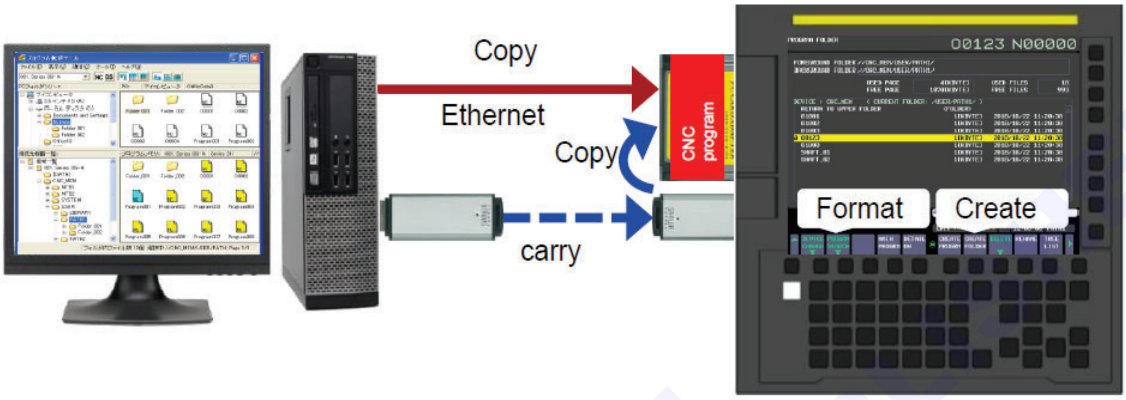
```
PROGRAM                                    00000 N04219
O0000 M3 ;
S7000 ;
M8 ;
%
                          A>_
                                   FINE  0S120%T00000000
                          MDI **** *** ***    16:26:24 RSTR
```

Program transfer through electronic media

Objectives: At the end of this lesson you shall be able to
- **explain the steps is transfer program from PC to machine by transfer tool software.**
- **describe the method to copy/move program from USB memory to memory card**
- **breif the steps to select main program from memory card**
- **brief the steps to safely remove the memory card.**

Program Transfer (Fig 1)

Fig 1



The program file can be transferred to the memory card in the following way

- Copy/Move from USB memory to memory card
- File transfer from PC to Memory Card over Ethernet using FANUCs Program transfer tool Software.

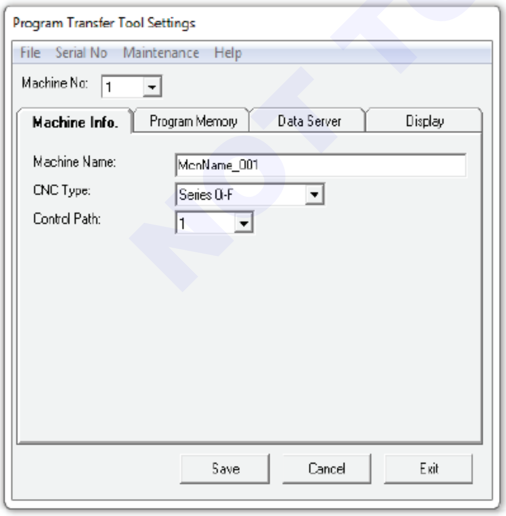
Steps Procedure for Program transfer with FANUC Program transfer tool software

1 Install Program transfer tool software in PC

2 After installing, select Start → Program Transfer tool → Setting dialog

3 In Machine Info tab, set Machine Name, CNC Type and Control path. (Fig 2)

Fig 2



Program Transfer Tool Settings

Machine No: 1

Machine Name: McnName_001

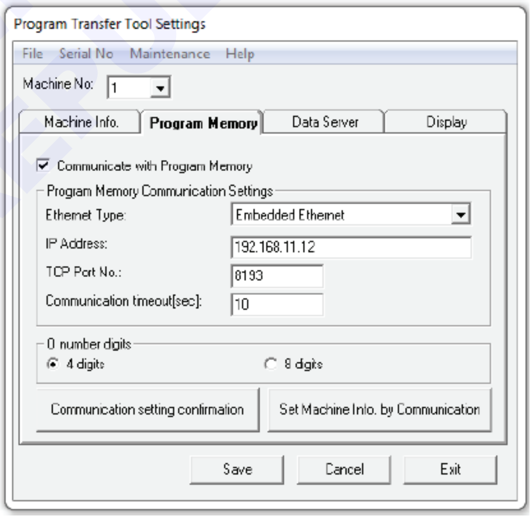
CNC Type: Series 0i-F

Control Path: 1

4 In Program Memory tab Enter IP Address, TCP Port no. (Fig 3)

5 Select SAVE and EXIT.

Fig 3



Program Transfer Tool Settings

Machine No: 1

☑ Communicate with Program Memory

Program Memory Communication Settings

Ethernet Type: Embedded Ethernet

IP Address: 192.168.11.12

TCP Port No.: 8193

Communication timeout[sec]: 10

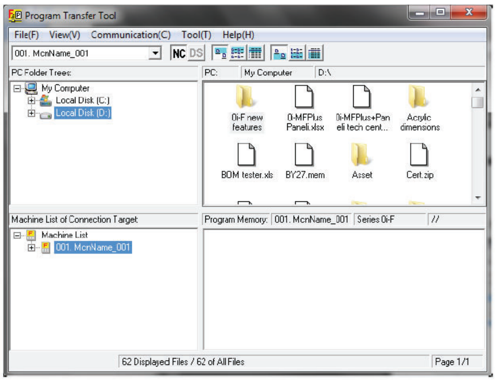
O number digits

● 4 digits ○ 8 digits

6 Select Start „ Program Transfer tool „ Program Transfer tool.

7 Select Machine List of Connection target„ Select Machine. (Fig 4)

Fig 4



001. McnName_001

PC Folder Trees:

My Computer
Local Disk (C:)
Local Disk (D:)

PC: My Computer D:\

Machine List of Connection Target

Machine List
001. McnName_001

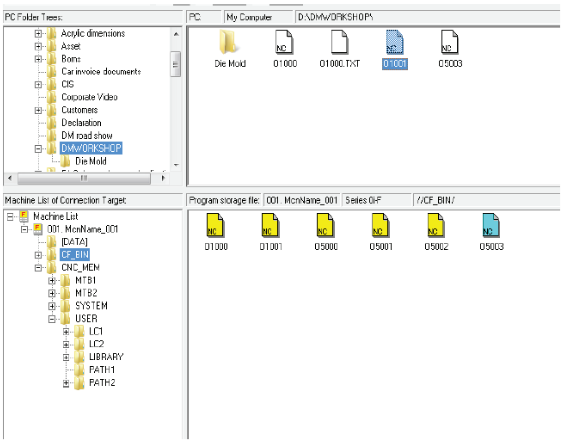
Program Memory: 001. McnName_001 Series 0i-F //

62 Displayed Files / 62 of All Files

Page 1/1

After selecting Machine → Select CF_BIN (Fig 5)

Fig 5



8 To upload / download the program, just drag and drop the program from PC to memory card folder

Set-up for copy/move program from USB to memorycard (Fig 6)

Fig 6

1 Set Parameter 20=4

2 Select EDIT mode press function key [PROG

3 Press the softkey [FOLDER]

4 Press the softkey [(OPRT)]

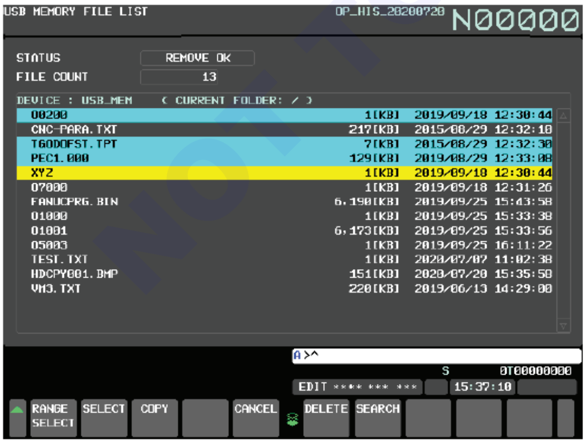
5 Press the softkey [DEVICE CHANGE]

6 Press the softkey [USB MEMORY]

7 Move the cursor [↑] [↓]with the cursor key to select the file.

8 Press the softkey [SELECT]. (Fig 7)

Fig 7



```
USB MEMORY FILE LIST                    OP_HIS_20200728 N00000

STATUS          REMOVE OK
FILE COUNT             13

DEVICE : USB_MEM    ( CURRENT FOLDER: / )
  00200                          1[KB]  2019/09/18 12:30:44
  CNC-PARA.TXT                 217[KB]  2015/08/29 12:32:18
  TG0DOFST.TPT                   7[KB]  2015/08/29 12:32:30
  PEC1.000                     129[KB]  2019/08/29 12:33:08
  XYZ                            1[KB]  2019/09/18 12:30:44
  07000                          1[KB]  2019/09/18 12:31:26
  FANUCPRG.BIN              6,190[KB]  2019/09/25 15:43:58
  01000                          1[KB]  2019/09/25 15:33:38
  01001                     6,173[KB]  2019/09/25 15:33:56
  05003                          1[KB]  2019/09/25 16:11:22
  TEST.TXT                       1[KB]  2020/07/07 11:02:38
  HDCPY001.BMP                 151[KB]  2020/07/20 15:35:58
  VM3.TXT                      220[KB]  2019/06/13 14:29:00

                                   A >^
                                             S     0T00000000
                                   EDIT **** *** ***   15:37:10
```

9 To copy multiple programs press the softkey [RANGE SELECT].

10 Selected file will be highlighted.

11 Press the softkey [COPY].

12 Press soft key [DEVICE CHANGE]

13 Press soft key [MEM CARD]

14 Press soft key [PASTE]

15 "EDIT" is displayed in the lower right of the screen while file is transferring.

Steps to Select "Main Program" from Memory Card (Fig 8)

Fig 8

To display and edit contents of memory card select [MEM CARD] by device changing operation as indicated below

1 Select EDIT model and press function key [PROG]

2 Press soft key [FOLDER]

3 Press soft key [(OPRT)]

4 Press soft key [DEVICE CHANGE]

5 Press soft key [MEM CARD].

6 Move the cursor to file/folder that you want to select the main program with the cursor key

7 Press soft key [MAIN PROGRM] "@" is displayed at the left of the program that has been selected as the main program to be executed.

Steps Safe removal of Memory card

When the memory card are used on the CNC, following procedure to be followed for safe removal of the memory card

1 Press function key [PROG]

2 Press soft key [FOLDER]

3 Press soft key [OPRT]

4 Press soft key [DEVICE CHANGE]

5 Press soft key [UN MOUNT]

6 A message "THE MEMORY CARD CAN BE TAKEN OUT" will be displayed.

Productivity concept

Objectives: At the end of this lesson you shall be able to
- **state the productivity**
- **brief the productivity concept**
- **brief the cycle time, machine downtime and causes of down time**
- **state the inspection,**
- **brief part loading /unloading, chip cleaning**
- **explain the effect of down time and reducing down time.**

Productivity

Productivity refers to the physical relationship between the quantity produced (output) and the quantity of resources used in the course of production (input). "It is the ratio between the output of goods and services and the input of resources consumed in the process of production."

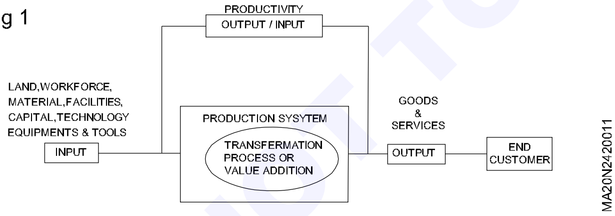
$$\text{Productivity (P)} = \frac{\text{Output (O)}}{\text{Input (I)}}$$

Output implies total production while input means land, labour, capital, management, etc. Productivity measures the efficiency of the production system. The efficiency with which resources are utilized is called productive efficiency. Higher productivity means producing more from a given amount of inputs or producing a given amount with lesser inputs.

Productivity – Concept (Fig 1)

The concept of productivity can be applicable to any economy, small, medium and large business, government and individuals. Productivity aims at the maximum utilization of resources for yielding as many goods and services as possible, desired by consumers at lowest possible cost. Productivity is the ratio of output in a period of time to the input in the same period time.

Fig 1



Productivity can measured with the help of following formula:

$$\text{Productivity} = \frac{\text{Output in a period of time}}{\text{Input in the same period of time}}$$

In simple terms Productivity is the ratio of output to some or all of the resources used to produce the output.

Productivity can thus be measured as:

$$\text{Productivity} = \frac{\text{Quantity of Goods and services produced}}{\text{Amount of resource used}}$$

Mathematically

$$P = \frac{O}{I}$$

Cycle time

The cycle time is the amount of time it takes to complete a specific task from start to finish.

Cycle Time = (Finish Time – Start Time) / Units Produced

What is machine downtime?

Machine downtime is any period of time (planned or unplanned) during which an equipment or machine is not functional or cannot work.

Causes of down time

Unplanned downtime often is a result of human error and inefficiency such as slow changeover and lack of knowledge or ability to complete tasks. Poor processes and inadequate maintenance are also common causes of unplanned downtime. On the other hand, planned machine downtime is frequently attributed to maintenance, cleaning, and upgrading equipment. Because it is a known and controlled type of downtime, planned downtime won't be part of our focus.

unplanned machine downtime

Most production managers can tell you their downtime in hours, but not all think through how much it actually costs. Getting buy-in from key stakeholders to make your recommended changes often requires informing them what is being lost in terms of dollars and not overlooking any costs. To consider the true cost of downtime, you should consider a number of factors such as lost revenue from not producing actual goods, loss in labor productivity, costs devoted to rescheduling runs and repairing equipment, and intangible costs like public relations (time spent setting things right with customers and managing your reputation). Taking these items into account, your formula for downtime would be:

Lost Revenue + Lost Productivity + Recovery Cost + Intangible Cost = Cost of Downtime

Machine break down

Manufacturing environments are dynamic in nature and are subject to various disruptions, referred to as real-time events, which can change system status and affect its performance .

These disruptions are machine breakdown, rush orders, and order cancellations etc. Machine breakdown is often an important factor in the throughput of manufacturing systems

If a machine breakdown occurs in a production system, throughput, which is one of the performance measures, might change, therefore, production planners must take precautions against this happening.

Machine breakdown can cause delays in the production system. Due to these delays, work activities are completed later than the time planned. The delays in workstations affect the whole production system and lead to deviations

Inspection

This standard defines inspection as “examination of a product, process, service, or installation or their design and determination of its conformity with specific requirements or, on the basis of professional judgment, with general requirements”.

An inspection is, most generally, an organized examination or formal evaluation exercise. In engineering activities inspection involves the measurements, tests, and gauges applied to certain characteristics in regard to an object or activity. The results are usually compared to specified requirements and standards for determining whether the item or activity is in line with these targets, often with a Standard Inspection Procedure in place to ensure consistent checking. Inspections are usually non-destructive.

Inspections may be a visual inspection or involve sensing technologies such as ultrasonic testing, accomplished with a direct physical presence or remotely such as a remote visual inspection, and manually or automatically such as an automated optical inspection. Non-contact optical measurement and photogrammetric have become common NDT methods for inspection of manufactured components and design optimization.

Part loading /unloading

Overhead XZ gantries need to perform many movements in the course of a day, reacting to any machine that requires parts to be placed or collected; reliability therefore has to be taken for granted.

Gantries can be any length within reason, but typically 20 meters is a common requirement. Using a gantry to feed in-line CNC machines often requires one or more Z axes, travelling 1.5 to 2.0 meters into the machine to pick or place components.

If you have been thinking about automating the loading/ unloading of your CNC lathes to reduce the labor cost on some of your longer running jobs, you may want to consider the parts loaders/unloaders made especially for such machines .

The part loaders offer 30 to 50 percent faster loading times than manual loading, enable the lathes to operate without operator attention for long periods of time and combine more consistent production with more accurate scheduling.

The loaders/unloaders can be retrofitted to machines from all or most of the major builders. According to the manufacturer, they are substantially less expensive than a gantry or a robot, and they typically pay for themselves in 12 months or less.

Chip cleaning

At least twice a year, all metal-working workshops needed maintenance and a complete cleaning of the CNC machinery to ensure their maximum efficiency.

The CNC machinery are machines designed to produce in series a large quantity of pieces, all equal to each other, removing, with a specific tools installed in the turret, large quantities of material in the form of chips or swarfs.

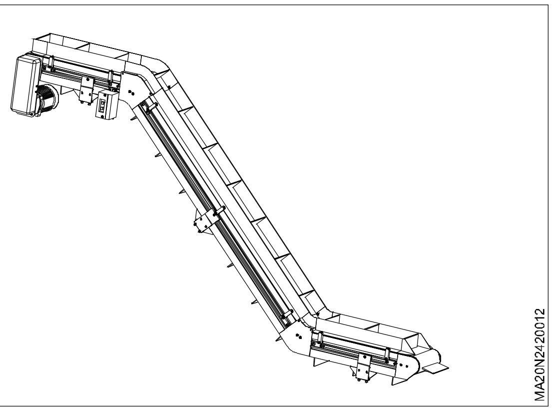
A chip conveyor system is very common on CNC machine tools where it can be part of the machine or it can be offered as an accessory. The main function of a chip conveyor system is to remove the cutting excess material from the machine tool.

Chip conveyors, including drag conveyors, allow you to extend the period between tank cleanouts from daily (when using chip trays) to as long as quarterly, reducing maintenance costs and machine downtime.

Drag conveyors (Fig 2)

Drag Conveyor scraper conveyor well suited for cast iron and aluminum.

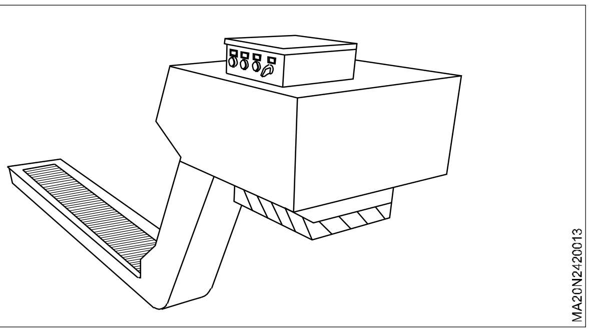
Fig 2



Clean Sweep RM (Fig 3)

Best midrange general purpose conveyor.

Fig 3



Effect of down time

Repeated downtime events can result in **unhappy customers, which can quickly translate into bad customer reviews and tarnished brand image**. Data Loss: Downtime affects not only your business but your clients as well. Downtime due to cyber attacks, server or network outage can result in corrupt, damaged or stolen data.

Reducing down time

Downtime is a serious threat to any manufacturing business. One of the most dreaded scenarios within a manufacturing plant is that if one area backs up, it can have a snowball effect on other areas waiting for completion of prior processes to deliver parts to them. The old saying "time is money" is very true in these manufacturing plant situations. The longer amount of time it takes to complete processes and finish products, the more money it costs to get those products out the door. Reducing the amount of time that people or machines idle will be more efficient for the business's bottom line. Take a look at the nine tips below on how to reduce downtime in manufacturing.

Motivating and inspiring your employees is a big first step to reducing downtime in your manufacturing plant. It is important to inspect and repair all machinery, and even somtimes make upgrades so that your employees have the correct atmosphere to achieve production goals. With a combined effort, correct equipment, and good atmosphere you can significantly reduce downtime in your manufacturing plants.

1 Increase and Improve Staff Communication

Staff communications between manager and employees is proven to be essential to improving efficiency between departments. For an employee to feel part of the team, the supervisor needs to explain the relationship between downtime and business profits—this makes the employee feel part of important decisions and operation strategies. Keeping your employees involved is key to increasing productivity. Happy and motivated staff members do a better job of contributing to the ultimate goal to reduce downtime in manufacturing settings.

2 Hold Regular Staff Evaluations

The majority of companies conduct regular performance reviews, but you can make yours more powerful by including a discussion of manufacturing downtime. Regular employee evaluations are standard, but the key is *how* they are administered. Evaluations need to be honest and straightforward. Evaluations should focus on what management views as areas of excellence and areas needing improvement. Armed with the proper feedback, employees stay motivated to increase their areas of strength, but also fight to reduce manufacturing downtime.

3 Set Specific Manufacturing Plant Goals

Set both daily and quarterly incentive goals to reduce errors and improve productivity both of these can help reduce downtime in manufacturing dramatically.

With motivated and inspired staff, downtime is minimized. Employees who understand the role they play in the manufacturing process provide more valuable feedback and work at more efficient levels. Using employee suggestions on how to decrease downtime, increase morale, and produce more goods is also important. Listen to your employees on the production line because they're the ones involved hands-on with production, they can offer valid ideas on how to decrease downtime.

Listening to your production line employees' suggestions also makes them feel part of the decision-making process. Educating employees about each part of the manufacturing process may help them better understand their specific role and job. Managers who make sure that all employees have incentives to excel and consider their suggestions are also rewarded with higher productivity. This is true across any industry.

4 Conduct Regular Manufacturing Equipment Maintenance

Machinery malfunctions need to be prevented to minimize manufacturing downtime. Businesses that monitor, inspect, update, and repair their machinery create the right atmosphere for meeting goals to help eliminate downtime. Machines that continuously jam or break down have a huge negative impact on downtime outcomes. Regular maintenance is critical to keeping machines operating at their best for the longest stretches of time possible.

5 Upgrade Your Manufacturing Equipment

Replacing obsolete equipment to improve performance is an essential piece of reducing manufacturing downtime. Outdated machines slow down the manufacturing process. Installing new equipment should allow your plant floor to stay up-to-date on the latest technologies, which allows managers to suggest updates to improve productivity.

The Overall Equipment and Effectiveness (OEE) measurement is frequently used to gauge a plant's effectiveness. There are several common causes of efficiency loss:

- Breakdowns
- Set up Losses
- Minor Stoppages
- Reduced Speed
- Start-up Issues
- Rework to Resolve Quality Issues

Ethernet communications network in your plant will also allow you to harvest actionable energy management data from connected drives, energy meters and process controllers. This will all help eliminate energy waste.

6 Track Manufacturing Downtime Carefully

Knowing when, where, and how downtime occurs is essential to knowing how to prevent it. An early step toward reducing unexpected production backups or outright downtime can be achieved by carefully and accurately tracking when and where downtime occurs. Automatic trackers detect downtime immediately and feed this information out to the factory floor for real-time viewing. Alerts are visible and show the exact location issues have occurred. This allows you to figure out why each instance happened. When you know that user errors are causing the majority of your noted stalls, you know what changes to make and what to prioritize in your efforts.

7 Properly Train Production Line Employees

User error is one of the top causes of downtime on a production line. Some reports say as much as 70% of downtime can be attributed to user error. There are numerous ways that operators can cause a machine break down, so proper training shouldn't be taken lightly. Proper education is imperative for machine operators to produce their most efficient results. They need to know how to use tools effectively. Teaching team members so they're capable of operating machines correctly will result in less operator errors that ultimately cause slowdowns and downtime.

8 Replace the CPU and Software

Instantly gain benefits of a new platform without changing your hardware. This enables you to preserve your investment in application design and embedded process knowledge all while extending the life of your existing control system incrementally. The new operational capability you provide helps eliminate previous downtime areas that were out of reach of the old CPU. New software provides the basis for any technology upgrades you will want to do in the future. New software can also provide a smooth transition for operating personnel to the new technology.

9 Create a New Technology Plan for Your Manufacturing Plant

Develop a strategic enrollment plan that allows you to integrate new technologies into your plant with a modernization program. Implementing a modernization program is a good way to reduce the amount of unplanned downtime. A step-by-step plan is a good approach that will not only increase uptime, but also provide a range of benefits for your processing facility. In modernizing the technology in your plant you may choose to install an industrial Ethernet plant network, standardize on single PLC software, introduce networked Human Machine Interface (HMI's). Budgets can be more easily maintained with smaller upgrades that are planned over time. They can possibly be financed by a multiple departmental budgets, which can ease the upgrade processes.

Taking the steps to reduce downtime in manufacturing settings can seem overwhelming, but the money saved, and additional profits earned from making adjustments to reduce downtime are worthwhile.

Machine hour rate

Objectives: At the end of this lesson you shall be able to
- **state the meaning of machine Hour Rate**
- **describe the components of machine hour rate**
- **list the points in computing machine hour rate**
- **explain how to calculate the machine hour rate**
- **explain the down time in manufacturing**
- **determine the true cost of down time.**

Meaning of Machine Hour Rate

Machine hour rate refers to the amount of factory overheads incurred for running a machine for an hour. In short, machine hour rate means the factory expenses incurred in running a machine for an hour. The machine hour rate is calculated by dividing the total amount of factory overheads incurred in running a machine during a particular period by the total number of working hours of that machine during that period.

Types of Machine Hour Rates

There are two types of machine hour rate, viz.:

1 Simple or Ordinary Machine Hour Rate.

2 Composite or Comprehensive Machine Hour Rate.

Simple Machine Hour Rate

Simple machine hour rate, generally, means only the total machine expenses or the total variable expenses like power, fuel, repairs and depreciation of the machine which are directly connected with the operation of the machine per hour.

Composite Machine Hour Rate

Composite machine hour rate means the total variable expenses per hour plus the total fixed, constant or standing charges per hour not directly connected with the operation of the machine, but are the general factory overheads of the department. In other words, it is the machine hour rate which includes the total machine expenses per hour plus the total fixed charges per hour plus the direct wages of machine operators per hour.

Computation of Machine Hour Rate

1 The factory overheads are first apportioned to production departments under allocation and apportionment.

2 Overheads of the departments are further apportioned to different machines or group of machines. For this purpose each machine or a group of machines is treated as a cost centre or a small department.

3 Specific overheads like power, depreciation, etc., should be directly allocated to the machine.

4 The working hours of a machine are estimated for the period.

5 Overheads pertaining to the machine are totaled and divided by the number of effective machine hours. The resultant figures will be machine hour rate. The time required for getting the machine should be deducted from the total working hours to arrive at effective hours.

6 Where comprehensive machine hour rate is desired include direct wages along with other expenses incurred for operating the machine.

7 Those overheads which can be allocated directly to the machine, such as depreciation, power repairs and maintenance are to charged, specifically to the machine.

8 Along with above, include cost of stand by equipment to calculate machine hour rate.

9 Make necessary adjustments for calculating the effective machine hours by deducting cleaning and warming up time.

Departmental overheads can be classified into two broad heads:

1 Standing or Fixed Charges

2 Variable expenses.

Standing or Fixed Charges

Standing or fixed charges refer to those charges which remain constant or fixed irrespective to the use of running of the machine. Example of standing charges are rent, rates and taxes, lighting and heating, insurance charges of the machine, insurance charges of the building, indirect wages, supervisory charges, general expenses, labour welfare expenses, etc.

In this context, it may be noted that, generally, all expenses other than depreciation of the machine, repairs and maintenance charges of the machine, and power, steam and water are considered as fixed charges. All these expenses are allocated to all the machines of the department on a logical basis.

The main bases of the allocation of fixed expenses are as follows

S. No.	Fixed Expenses	Basis of Allocation
1	Rent, Rates and Taxes	Floor area or space occupied by each machine
2	Lighting	Number of light points of each machine of floor area occupied by each machine or number of operators engaged on each machine.
3	Heating	Floor area occupied by each machine or on the basis of technical estimates.
4	Supervisory Charges	Time devoted by the supervisory staff
5	Canteen Expenses and Welfare Expenses	Number of workers exgaged on each machine
6	Insurance schemes (Various)	Number of workers engaged on each machine
7	General charges	Generally these are apportioned equally among all the machines in the department
8	Insurance of machines	Insurance charges are apportioned on the basis of the capital value of cost of each machine
9	Insurance of building	On the basis floor area
10	Consumable materials, stores, lubricants, etc.	Separately or are apportioned on the basis of the nature of the machine.
11	Sundry expenses	On any justified basis

Depreciation on the Machine

As regards the depreciation on the machine, certain portion of depreciation is attributable to the use of the machine. That means a part of the depreciation is a fixed charged and a part is a variable charge. But it is difficult to ascertain that part of depreciation which is fixed and that part of depreciation which is variable. So, generally, the entire depreciation on the machine is treated as variable charges.

Another point to be noted in the context of depreciation on the machine is that, while calculating the depreciation on the machine, the replacement cost, even if it is given, should not be taken into account. Only its original cost and scrap value should be taken into account.

These can be expressed in the form of formula as follows:

$$\text{HourlyDepreciation} = \frac{\text{Cost of Machine} - \text{Scrap value}}{\text{Total Working Hours of Machine}}$$

Where,

1 Cost of Machine = This includes purchase price of the machine, carriage inward and installation charges.

2 Scrap Value = Value of machine remaining after its exhaustive use.

3 Total Working Hours of Machine = It means total working hours during the life time of the machine.

Repairs and Maintenance Charges: If the problem does not state whether the given repairs and maintenance charges are for the life time of the machine or for one year, one can assume that the given repairs and maintenance charges are for the life time of the machine. Repairs and maintenance charges are divided by the working hours for the concerned period to find out repairs and maintenance charges per hour.

Power Expenses

Those expenses which are concerned with the operation of the machine are called power expenses.

This can be calculated separately per hour as follows:

$$\text{(a)}\frac{\text{Power Charges per Year}}{\text{No.of working hours per Year}}$$

Or

(b) Number of Units of power consumed per hour x Rate of Power per Unit

Other Points to be Borne in Mind While Computing Machine Hour Rate

Base Period

In every problem in machine hour rate certain period is taken as the base. Such a period is called base period. The base period may be a year, half year, a quarter, a month, a week or a day. Though any of the above periods can be taken as the base period.

It is preferable to take that period for which most of the fixed charges are given per annum, it is better to take one year as the base period. If most fixed expenses are given for one month, then, it is better to take one month as the base period. If most of the fixed charges are given for a week, it is better to take week as the base period.

Effective or Productive Working Hours

The effective working hours of the machine during the base period is necessary for the calculation of the total fixed charges per hour and each of the variable charges per hour. So, it has to be calculated.

The effective working hours of the machine can be calculated as follows:

From the total normal working hours of the machine, the hours required for maintenance and repairs, setting up, cleaning of machine, warming up, staggering and the time cost due to other normal idle time of the machine must be deducted. The resulting balance is the effective working hours of the machine.

It may be noted that the time required for the setting up of the machine should be deducted from the normal working hours of the machine only if it is considered as unproductive time. On the other hand, if the setting up time is considered as productive time, it should not be deducted.

It should also be noted that the abnormal idle time resulting from break-down of machine, power failure, etc., should not be deducted from the normal working hours. Only the normal idle time due to maintenance and repair, setting up, and staggering should be deducted.

The idea behind deducting the normal idle time from the normal working hours is that when we take only the effective working hours for the calculation of the machine hour rate, the cost of normal idle time, which is one of the factory overheads, is also absorbed in the machine hour rate.

Machine Operator's Wages

Machine operator's wages refer to wages of workers, such as turners and fitters who operate the machine. Strictly speaking, wages of machine operators are direct wages and so, it should not be taken into account as factory overhead in the computation of machine hour rate. That is why, some of the authors do not include machine operator's wages in the computation of machine hour rate.

Consumable Stores

Consumable stores refer to lubricating oil, cotton waste, rages, etc. used for cleaning the machines. There is a different of opinions among the authors as regards the treatment of these expenses. Some authors argue that these expenses vary with the use or running of the machine, and as such, they should be treated as variable expenses. But there are other authors who take them as fixed or standing charges.

Hire Purchase Installment

Sometimes, a machine might have been purchase on hire purchase system. In such a case, the price of the machine is payable in instalments. Each instalment payable is called hire purchase installment. In each hire purchase instalment, some amount is payable towards the cash price of the machine and some amount is payable towards the interest for the outstanding cash price of the machine.

The hire purchase instalment amount has to be treated as follows:

The amount payable in each instalment towards the cash price of the machine should not be included in the computation of the machine hour rate, as it is capital expenditure, and not revenue expenditure.

However, as regards the amount payable in each instalment towards the interest, there is difference of opinion among authors. Some authors consider the hire purchase interest as one of the fixed charge, and as such, include it in the calculation of the machine hour rate.

Format for computing machine-hour rate

Computation of machine-hour rate

Base Period..........

Working hours............

Particulars	**Per Day or Per Week or Per Month or Per Year (Base Period)**	**Per Hour**
Standing or fixed expenses:	**Rs.**	**Rs.**
1 Rent, Rates and Taxes (According to the areas used by each machine)	....	
2 Lighting and Heating (In the ratio of light points or bulbe or floor area)	....	
3 Supervision Expenses (Time devoted by supervisory staff to each machine)	....	
4 Labour welfare expenses (In the ratio of number of worker)	....	
5 Insurance (In the ratio of number of worlers)	....	
6 Manager's Salary	....	
7 Foreman's Salary	....	
8 Canteen Expenses	....	
9 Consumable stores--Lubricants, cotton waste, etc. (On the basis of give information	....	
10 Other expenses (on justified basis)	....	
Total Standing Charges		
Standing Charges p.h = $\frac{\text{Total Standing CHarges}}{\text{Working hour in base period}}$		....
Machine Expenses or Variable Expenses		
1 Depreciation= $\frac{\text{Cost of Machine - Scrap Value}}{\text{Total Working life of machine in hours}}$		....
2 Repairs and Maintenance- $\frac{\text{Repaire and Maintenance}}{\text{Working hours for given amount of Repairs and Maintenance}}$		....
3 Power = Units Consumed p.h x Unit rate		....
4 Steam and Water (As per instruction given)		....
	Machine-hour Rate	....

The following particulars relates to new machine:

Purchase price	4, 00,000	
Installation Expenses	1, 00,000	
Rent per quarter	3,750	
General Lighting for the total area	30,000	per month
Forman's salary	3,000	per month
Insurance Premium for the machine	5,000	per month
Departmental Overheads for the machine	4,000	per month

Consumable Stores

Power -2units per hour at 50 paisa per unit.

The estimated life of the machine is 10years and scrap value at the end of 10th years is Rs.1, 00,000/

The machine is exempted to run 20,000 hours in its life time. The machine occupies 25% of total area. The foreman devoted 1/6th of his time for the machine.

You are required to

a Work out the machine hour rate

b Work out the rate for quoting to the outside party for utilising the ideal capacity in the machine shop assuming a profit of 20% above variable cost.

Solution

a Computation of Machine Hour Rate

Standing charges

Rent, Rates and Taxes	₹ 5,000.00	
Supervision	₹ 5,400.00	
Operator's wages	₹ 1,950.00	
Total standing charges	₹ 12,350.00	
Effective hours (46x13)-598		₹ 20.65
Fixed costs per hour (12,350/598)		
Variable costs per hour		
Power: $\frac{15x44x0.4}{46}$	₹ 5.74	
Repairs and maintenance 4,000/598	6.69	
Consumable stores 7,500/598	12.54	
Depreciation: 21/4 x 600/598= $\frac{21,600}{598 \times 4}$	9.03	
		34.00
		54.65

b Quotation for outside parties

Variable cost per hour	34.00
Add: 20% profit	6.80
Minimum to be quoted	₹ 40.80

The machinists divided the machine's price by the number of hours ot will operate in a determine their machine shop rates (also known as the machinig cost per hour). Some clients also use machining cost estimator apps to estimate the cost for their projects.

Downtime in Manufacturing

Downtime in manufacturing is defined as any period of time when a machine is not in production. The total amount of downtime a factory experiences includes any stops during production that cause into two different categories; planned and unplanned. Planned downtimes are schedulated and budgeted stops during production downtime in manufacturing occurs when equipment that is scheduled to be in operation has an unexpected event such.

Planned downtime in manufacturing

Scheduled machine maintenance is an example of planned downtime in manufacturing. Daily maintenance programs that can be corrected before they become a major problem that can shut down a production line. Product changeover is another example of planned downtime. Producing one product to another during a planned downtime. A typical changeover consists of setup and adjustments such as ramping up and down both at the beginning and end of a run.

Planned downtime still has a cost to revenue, so the priority is to expedite machine maintenance and product changeover to remain within the budgeted timeframe. This is not always the case and delays have an impact on the overall equipment effectiveness (OEE) as well as contribution margins.

Unplanned downtime in manufacturing

Unplanned downtime are any unexpected stops that occur during production. The stops occur without notice and can last any length of time and can create massive backups along the production line.

These unplanned halts during production eat into the maximized hours in a work day and inevitably diminish optimized revenue.

The true downtime cost

The true costs of downtime (also known as TDC), consists of analyzing all cost factors associated with downtime. The two variables of TDC are tangible costs and intangible costs. This information is then used for management decisions and for the justification of costs.

Importance of Technical term used in Industry

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

- **state the meaning of different terms used in industry.**

Engineering terminology

Broach (v) to finish the inside of a hole to a shape other than round, as in a keyway (n) The tool for the process, which has serrated edges and is pushed or pulled through the hole to produce the required shape.

Bushing (n) a smooth walled bearing (AKA a plain bearing). Also a tool guide in a jig.

Cam (n) A mechanical device consisting of an eccentric or multiply curved wheel mounted on a rotating shaft, used to produce variable or reciprocating motion in another engaged for contacted part (Cam follower). Also Camshaft

Casting (n) any object made by pouring molten metal into a mold.

Chamfer (n) Angular surface made by cutting off the edge or corner of a object (bevel) (v) the process of creating a chamfer.

Clevis (n) A U-shaped piece with holes into which a link is inserted and through which a pin or bolt is run. It is used as a fastening device which allows rotational motion.

Collar (n) A Cylindrical feature on a part fitted on a shaft used to prevent sliding (axial) movement.

Collet (n) a cone-shaped sleeve used for holding circular or rod like pieces in a lathe or other machine.

Core (v) to form the hollow part of a casting, using a solid form placed in the mould (n) the solid form used in the coring process, often made of wood, sand, or metal. Mould with core is final cast manifold.

Counter bore (n) a cylindrical flat-bottomed hole, which enlarges the diameter of an existing pilot hole. (v) The process used to create that feature.

Countersink (n) a conical depression added to an existing hole to accommodate and the conic head of a fastener recessing it below the surface of a face. (v) The process used to create that feature.

Coupling (n) A device used to connect two shafts together at their ends for the purpose of transmitting power. May be used to account for minor misalignment or for mitigating shock loads.

Die (n) one of a pair of hardened metal plates or impressing or forming desired shape. Also, a tool for cutting external threads.

Face (v) to machine a flat surface perpendicular to the axis of rotation of a piece.

Fillet (n) A rounded surface filling the internal angle between two intersection surfaces. Also Rounds

Fit (n) The class of contact between two machined surfaces, based upon their respective specified size tolerances (Clearances, transitional, interference)

Fixture (n) A device used to hold and support a work piece while manufacturing operations are performed upon that work piece.

Flange (See bushing example) (n) a projecting rim or edge for fastening, stiffening or positioning.

Gauge (n) A device used for determining the accuracy of specified manufactured parts by direct comparison.

Gear Hobbing (v) A special form of manufacturing that cuts gear tooth geometrices. It is the major industrial process for cutting involute form spur gears.

Idler (n) A mechanism used to regulate the tension in belt or chain. Or, a gear used between a driver and follower gear to maintain the direction of rotation.

Jig (n) A special device used to guide a cutting tool (drill jig) and hold workpiece in the correct position for drilling, reaming etc.

Journal (n) The journal of a shaft is the part of it that is in contact with (or) enclosed by a beaning

Keyseat (n) A slot or groove cut in a shaft to fit a key. A key rests in a keyseat.

Keyway (n) A slot cut into a hub to fit a key. A key slides in a keyway.

Knurl (v) One of a series of small ridges or beads on a metal surface to aid in gripping.

Pinion (n) A plain gear, often the smallest gear in a gearset, often the driving gear, May be used in conjuction with a gear rack (rack and pinion).

Planetary Gears (n) A gear set characterized by one or more planet gear (s) rotating around a sun gear. Epicyclic gearing systems include an outer ring gear (known as an planetary system)

Rack (w/pinion gear) (n)A toothed bar acting on (or acted upon) by a gear (pinion)

Ratchet (n) A space mechanical device used to permit motion in one direction only.

shims (n) a thin strip of metal inserted between two surfaces to adjust for fit. (v) The process of inserted shims

Spline (n) A cylindrical pattern of keyways. May be external (L) or Internal (R)

Tap (v) To cut internal machine threads in a hole, (n) the tool used to create that feature.

Undercut (n) It is a special type of recessed surface in a diameter.

Definitions of technical terms- CNC SYSTEM

Absolute co - ordinates: The distance (or dimensions of the current position from the origin (or zero point) of a coordinate system (or measuring system) measured parallel to each axis of the system.

Address: A name (or label, or number) identifying a storage area in a control system or computer memory.

Analog: The use of physical quantity (like voltage) whose amplitude represents that of another quantity (like distance)

APT: Automatically programmed tools.

ASCII: American standard code for information interchange.

Auxiliary function: Another name for miscellaneous function.

Axis: When associated with machine tools, an axis is a direction in which a machine tool table or head can move.

Backlash: The maximum movement at one end of a mechanical system (such as geartrain) which does not cause the other end to move.

Base number: A base number (or radix) is an implied number used when expressing a value numerically in the normal decimal system, the base is 10 and 87 is a short way to representing $8x10^1+7x10^o=87$ in binary notation 1011 represents $1x2^3+0x2^2+1x2^1+1x2^0 = 8+2+1 = 11$ in decimals

Binary coded decimal number (BCD): The representation of a number by groups of four binary digits for each decimal digit in a number.

Binary digits: The characters 0 or 1 used in the binary system to express any value.

Bit: A binary digit or its representation.

Block: A collection of words in some agreed form. Usually on a control tape, and separated from succeeding blocks by an End of -Block code.

BIS - Bureau of Indian Standards.

Buffer: A temporary storage area where information is held until it is moved into an operating area.

CCW: Counter clock wise.

Characters: The set of letters, decimal digits, signs (such as + - :% etc)

Circular interpolation: A contour control system with circular interpolation cuts an arc of a circle from one block on a control tape.

Co -ordinate system: A series of intersecting planes, or planes and cylinders (usually three) which form a reference system.

CPU: Central processing unit. The controlling unit in a digital computer.

Cutter diameter compensation: Provision in the control system to modify the cutter offset by entering a numerical correction to the cutting tool diameter.

Cutter offset: Position of reference point on the tool.

Cutting speed: The velocity of the cutting edge of the tool relative to the work piece.

Cycle: A sequence of operations which frequently repeated.

Depth of cut: The amount of metal (in mm or inch) removed perpendicularly to the direction of feed in one pass of the cutter over the work piece.

Digit: a character used in a numbering system.

Downtime: Time during which equipment or machine is out of action because of faults.

Dwell: A pause of programmed duration, usually to ensure that a cutting action has time to be completed.

End-of-block mode: An agreed code which indicates the completion of a block of input information on punched tape.

Feed: The movement of a cutting tool into a work piece.

Feed rate: The rate, in mm/min or in. /min, at which the cutting tool is advanced into the work piece.

Format: An agreed order in which the various types of words will appear within a block,

Hardware: Equipment e.g., Machine tool, or Control, or computer.

Incremental co-ordinates: The distance of current position from the preceding position, measured in terms of axial movements in the co-ordinate system.

Interpolation: The process of supplying the positions of a set of more closely spaced points between more widely spaced points such as change points.

IPM: Inches per minute (in. /min).

IPR: Inches per revolution (in. /rev) of a cutter or work piece.

ISO: International Organization for Standardization.

Machine language or machine instruction: The instructions and data for computers are based on patterns of bits.

Machine zero: The machine zero point is at the origin of the coordinate measuring system of the machine.

Manual data input (MDI): A means, on the control panel, of inserting numerical information into the control system.

Manuscript: Handwritten program.

Miscellaneous function: A control tape term for codes such as M03 used to control machine tool functions such as 'rotate spindle clockwise'.

NC: Numerical control.

Preparatory function word: A word near the beginning of a control tape block which calls for a change in mode.

Preset tools: The setting of tools in special holders away from the machine tool.

Program: A systematic arrangement of instructions or information to suit a piece of equipment.

RPM: Revolutions per minute (rev/min), a measure of spindle speed.

Sequence number: The number allocated to a block or group of blocks to identify them. Commonly takes a form like N 278.

Servomechanism: A closed-loop positioning-control system.

Software: Programs, sequences of instructions, etc. Not equipment (see Hardware).

Spindle speed: the rotational speed in RPM of the spindle or shaft which supplies the cutting power.

Subroutine: A sequence of computer-programming statements or instructions which perform an operation frequently required.

Word: An agreed arrangement of characters and digits (usually less than 10) which conveys one instruction, piece of information, or idea.

Technical forms used in industries

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

- **state the documents used in the production shop**
- **read and implement the process chart**
- **analyse the production drawing**
- **explain the process chart, flow chart**
- **estimate the time for each operation**
- **fill the work activity form**
- **record report for the inspection, batch production record and production cycle time**
- **select the relevant drawing and prepare the production related documents.**

Production activities

Maintain various documentation of formats according to, their manufacturing capacity. The production unit has to maintain certain records to streamline the production activities, to control, guide, monitor, status.

The following documents are required in production units.

1 Product drawing

2 Process chart

3 Flow chart

4 Job card

5 Bill of materials

6 Estimation sheet

7 Work related activity

8 Inspection report

9 Batch production record

10 Production cycle time

11 Machines and equipment record

12 Preventive maintenance log.

Apart from the above, some organizations maintain daily production record, stage inspection report according to their needs.

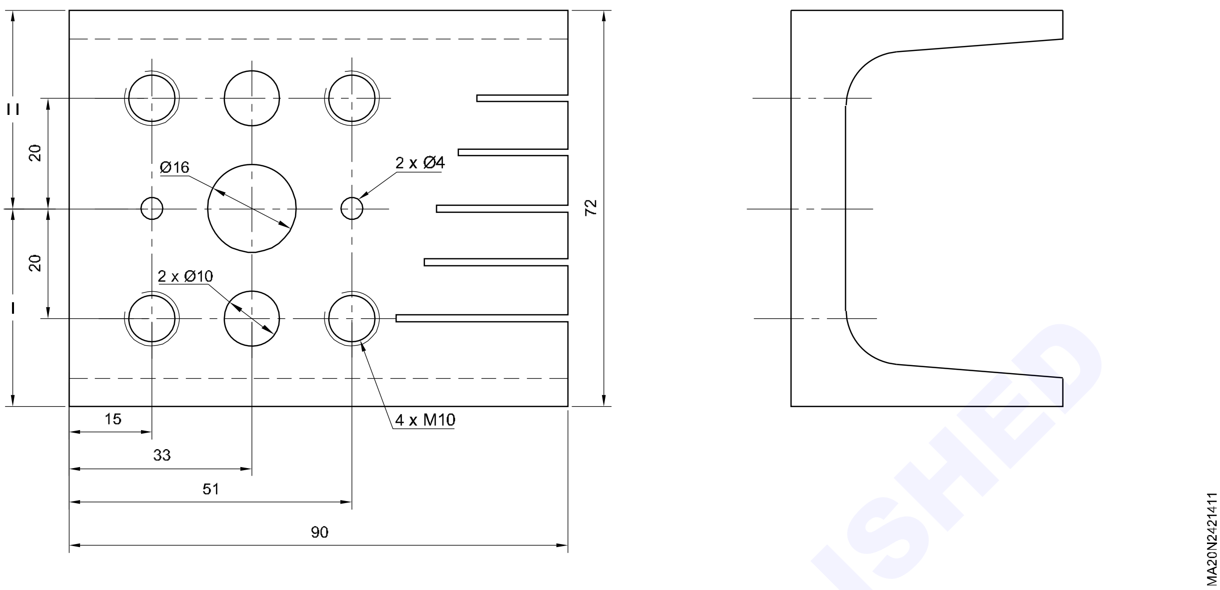
Product drawing

This document contains complete details assembly/ drawing parts raw material dimensions, technical parameters, test conditions, inspection details, preservation methods, packaging etc.

Process chart (Fig 1)

The process chart is prepared from raw materials to packing stage. It indicates the step by step operation to be performed, machine, equipment, tools, instruments, safety equipment/ instructions, treatment/ test procedure, etc. These are recorded in a regular format.

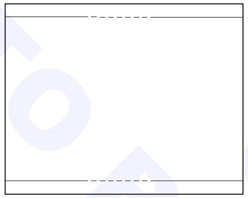
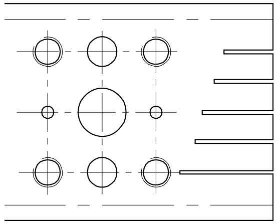
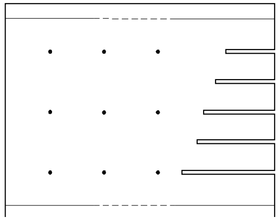
Fig 1

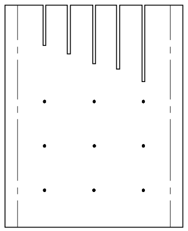
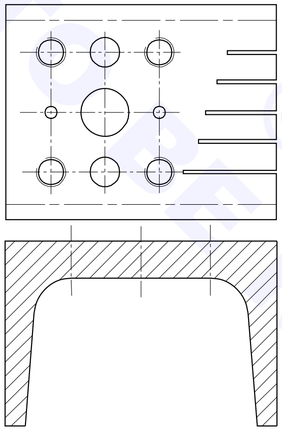
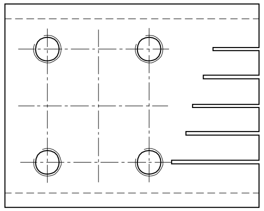


Process Chart

Product Detail Name Fitting Excercise DRG No: 013 Qty/Assy W.O.NO: Warrant No:

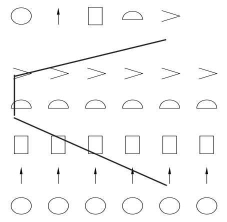
RAW Material Specification: FE1-05, ISMC 4.4x75x95L.F No:xx/xx/xxxx Total Order Quantity: 1

OPN No. Centre	**Work Description**	**Operation Safety**	**Operation Sketch**	**Operation**	**Tools**	**Instrument**	**Precaution**	**Remarks**
000	Stores	Collection of Material						
005	Fitting	Maintaning Overall size of Channel 90x72		Filing	5” Bench vice 300mm Bastard File, 300mm Second Cur File	Steel Rule Vernier caliper		
010	Work Bench	Apply Marking Media		Applying Marking Media	Cellulose Lacquer	-		
015	Marking Table	Mark Hole Centre and sawing location as per drawing		Marking	Vernier Height Gauge, Angle plage		Ensure zero setting of vernier height gauge before marking	
020	Work Bench	Punching hole Center and sawing lines		Punching	Anvil center punch prick punch, ball peen Hammer		Keep the punch at 90° with respect to job face while punching	

OPN No. Centre	Work Description	Operation Safety	Operation Sketch	Operation	Tools	Instrument	Precaution	Remarks
025	Work Bench	Sawing slot to the required depth		Sawing	5” Bench vice, Hand Hacksaw Frame with blade	Steel Rule		
030	Drilling Machine	Drilling ϕ8.5, ϕ4, ϕ10, ϕ16		Drilling	Drills - ϕ4, ϕ8.5 ϕ10, ϕ16 5” Machine vice drill chuck with key drift	Vernier caliper 150mm		
035	Fitting	Tapping 4 holes to M10		Tapping	5” Vice M10, tap and tap wrench 6” Try square	M10 Standard Bolt		Check the tap hole by M10 bolt
040		Deburr		Deburring	File 150mm smooth			
045	Inspection table	Check all dimension as per drawing		Inspection	Try squre	Vernier caliper 150 mm		
050	Stores	Apply oil		Preserving	oil			To protect from rusting

Flow chart

Induction, monitoring and stages of production activity will be recorded graphically in the flow. It also shows quantity and time at each stage. This helps to position the stages when the product is under process.

Detail		Qty	Time (in minutes)
Raw material from stores to cutting			
Machining			
Filing			
Inspection of finished job			
To stores			

The following symbol set derived from Gilbreth's signal work as the standard for flow chart

Symbol	Letter	Description
○	O	Operation
→	M	Transport
□	I	Inspection
D	D	Delay
∇	S	Storage

Job card

The job card is the document that records the work to be performed. It contains about order numbers, starting date, line, start, end and total time.

JOB CARD						
DOC. NO.			Rev No.		Date	
W.O. NO.				Order starting date		
Details						
Sl.No.	Date	Production line	Time in minutes			Remarks
			Start	End	Total	
		Stores				
		Surface table				
		Work bench				
		Fitting				
		Drilling machine				
		Inspection				

Bill of material

This is a document prepared from the assembly drawing and consists of part drawing, standard parts, raw materials specifications and following size. The details included are shown in the following format.

Bill of material						
Sl.No.	**Item No**	**Description**	**Quantity**	**Ref. Drawing No**	**Material size as per standard**	**Remarks**

Normally, this information is available in the assembly drawing along with the title block column.

1 **Estimation Sheet:** This document is very essential to ascertain and establish the manufacturing cost.

2 **Estimation sheet is prepared:** Very critically taking into account the operations performed during manufacturing. A sample format shown below.

Estimation Sheet

Part Name: ____________________ Part No: ______________________ Part Drawing

Assembly: ____________________ Material: ____________________

Assembly No.: ____________________ Stock size: ____________________

Operation No.	Operation	Machine	Estimated time	Rate / piece per	Tool

Prepared ______________________________ Date ________________

Approved by ____________________________

Note: Trainer will workout the required details for each operation

Work activity

In this document, the performance of the operation is recorded at one hour time intervals. It gives effective use of the operation. A work activity log gives the required information.

Work Activity

Organisation Name:
Department:
Section:
Employee Name:
Supervisor Name:
Date:

Start/ stop	Operations performed	Equipment / Machinery / Instruments used	Remarks
8.00 to 9.00 a.m.			
9.00 to 10.00 a.m.			
10.00 to 11.00 a.m.			
11.00 to 12.00 noon			
12.00 to 1.00 p.m.			
1.00 to 2.00 p.m.			
2.00 to 3.00 p.m.			
3.00 to 4.00 p.m.			
4.00 to 5.00 p.m.			
5.00 to 6.00 p.m.			

Inspection report

The inspection report document authenticates the correctness of the part manufactured as per dimensions and technical parameters. However, some products require various stages of inspection according to the operation before it reaches the final stage. The format with minimum requirement is as given.

Inspection Report							
Part Name:			W.O.No.			Ordered	
DRG No.:			Wt. No.			Qty:	
SI.No.	Operation	Submitted Quantiy	Rejected Quantiy	Accepted Quantiy	Rework Quantity		Remark
					Deviations	Quantity	

Batch production record

After inspection the manufactured component is grouped according to the deviation remarks. Deviation details are useful while assembling with the mating parts. The format is given below

Batch Production Record

Batch Production Record in accordance with batch processing record

Manufacturing Organisation Name:________________________________

Description of job:____

Name of part:____

Batch No.:____

The following deviations have appeared (continued).

No. of step in process	**Name of processing step**	**Documented page no.**	**Short description of deviation**
1	Raw material preparation: Operation 1 ____________________ Operation 2 ____________________ Operation 3 ____________________ Operation 4 ____________________		1 __________________ 2 __________________ 3 __________________ 4 __________________
2	Sizing of material: Operation 1 ____________________ Operation 2 ____________________		1 __________________ 2 __________________

Production cycle time

It is the total time starting from the beginning upto the end of the process. This document is interrelated with the estimation sheet. Job card is used to check the production activity to know the bottleneck in the production process. Production cycle time format comprises details about the organisation, product, operator / operation.

<table>
<tr><th colspan="15">Production Cycle Time</th></tr>
<tr><td colspan="15">Organisation Name: Process: Line Incharge: Date/Time:</td></tr>
<tr><td colspan="15">Department / Section:</td></tr>
<tr><td colspan="15">Operator:</td></tr>
<tr><th>Operation Sequence</th><th colspan="11">Observed Times</th><th>Lowest Repeatable</th><th>Machine Cycle Time</th><th>Notes</th></tr>
<tr><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td></tr>
<tr><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td></tr>
<tr><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td></tr>
<tr><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td></tr>
<tr><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td></tr>
<tr><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td></tr>
<tr><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td></tr>
<tr><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td></tr>
<tr><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td></tr>
<tr><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td></tr>
<tr><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td></tr>
<tr><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td></tr>
<tr><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td></tr>
<tr><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td></tr>
<tr><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td></tr>
<tr><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td></tr>
<tr><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td></tr>
<tr><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td></tr>
<tr><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td></tr>
<tr><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td></tr>
<tr><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td></tr>
<tr><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td></tr>
<tr><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td></tr>
</table>

Machinery and equipment record

The production unit should maintain the list of machines and equipment details like supplier, model, spares,

Inbricants, etc. Each item detail should be entered in the format given.

<table>
<tr><th colspan="2">Machinery and equipment record</th></tr>
<tr><td>Organisation Name:</td><td></td></tr>
<tr><td>Department</td><td></td></tr>
<tr><td>Section:</td><td></td></tr>
<tr><th colspan="2">History sheet of Machinery & Equipment</th></tr>
<tr><td>Description of equipment/Machinery</td><td></td></tr>
<tr><td>Manufacturer’s address</td><td></td></tr>
<tr><td>Supplier’s address</td><td></td></tr>
<tr><td>Order No. and date</td><td></td></tr>
<tr><td>Date on which received</td><td></td></tr>
</table>

Date on which installed and placed	
Date of commissioning	
Size: Length x Width X Height	
Weight	
Cost	
Motor particulars	Watts/H.P./ r.p.m: Phase: Volts:
Bearings/ spares/ record	
Belt specification	
Lubrication details	
Major repairs and overhauls carried out with dates	

Preventive maintenance log

The machines and equipment should be periodically monitored and should be subject to routine maintance schedule to avoid production bottleneck due to breakdown. The format enables to program the frequency of maintenance schedules so as to minimise frequent breakdown of machinery / equipment.

Maintenance Log

Organisation Name:

Department:

Section:

Name of the machine:

Sl.No.	Date	Nature of fault	Details of rectification done	Signature of in-charge

Lubricants and lubrications

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 lubricants.**

With the movement of two mating parts of the machine, heat is generated. If it is not controlled the temperature may rise resulting in total damage of the mating parts. Therefore a film of cooling medium with high viscocity 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 minimises 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.

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 the classification of lubricants as per standard.**

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-phosporous 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.

Annexure - I - Industrial lubricating oils

Product	Kinematic viscosity Cst at 40°C.	VI	Flash point COC°C	Description/Application
General Purpose Machinery Oils				
Lubrex 57	54.60	..	160	Lubrex oils are low viscosity index straight mineral lubricants having good inherent oxidation stability; they protect machine elements from excessive wear and provide economical lubrication. These oils are recommended for lubrication of bearings, open gears, lightly loaded slides and guideways of machine tools.
Lubrex 68	64.72	..	160	
Flushing Oil				
Lubrex Flush 22	19.22	..	150	Lubrex Flush 22 is a light coloured, low viscosity, straight mineral oil specially developed for flushing of automotive and industrial equipment. The characteristics of Lubrex Flush 22 make it possible to easily clean all inacessible internal surfaces of various equipments.
Circulating and Hydraulics Oils (Anti-wear Type)				
Servosystem 32	29.33	95	196	Servosystem oils are blended from highly refined base stocks and carefully selected anti-oxidant, anti-wear, anti-rust and anti-foam additives.These oils have long service life, and are recommended for hydraulic systems and a wide of circulation systems of industrial and automotive equipment. These oils are also used for compressor crank. case lubrication, but are not recommended for lubrication of turbines and equipment having silver coated components.
Servosystem 57	55.60	95	210	
Servosystem 68	64.72	95	210	
Servosystem 81	78-86	90	210	
Servosystem 100	95-105	90	210	
Servosystem 150	145-155	90	230	
Spindle Oils				
Servospin 2	2.0-2.4	..	70	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.
Servospin 5	4.5-5.0	..	70	
Servospin 12	11-14	90	144	
Machinery Oils				
Servoline 32	29.33	..	152	Servoline oils provide good oiliness for general lubrication even under boundary lubrication conditions,protect parts against rust and corrosion and maintain thin film strength and anti-rust additives. Servoline oils are general purpose lubricants for all lubrication systems of textile mills, paper mills, machine tools.
Servoline 46	42.50	..	164	
Servoline 68	64-72	..	176	
Gear Oils				
Servomesh 68	64-72	90	204	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. Servomesh oils are recommended for lubrication of industrial gears, plain and anti-friction bearings subjected to shock and heavy loads and should be used in systems were operating temperature does not exceed 90°C. These oils are not recommended for use in food processing units.
Servomesh 150	145-155	90	204	
Servomesh 257	250-280	90	232	

Lubricating machines

Objective: At the end of this lesson you shall be able to
- **state the methods of applying a lubricant.**

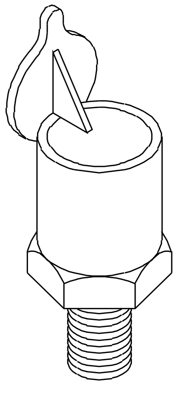
There are 3 systems of lubrication.

- Gravity feed system
- Force feed system
- Splash feed system

Gravity feed

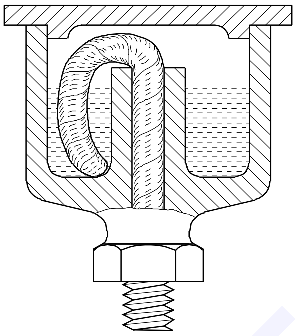
The gravity feed principle is employed in oil holes, oil cups and wick feed lubricators provided on the machines. (Figs 1 & 2)

Fig 1



OIL CUP

Fig 2



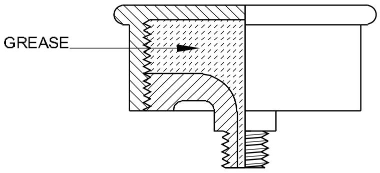
WICK FEED

Force feed/pressure feed

Oil, grease gun and grease cups

The oil hole or grease point leading to each bearing is fitted with a nipple, and by pressing the nose of the gun against this, the lubricant is forced to the bearing. Greases are also force fed using grease cup. (Fig 3)

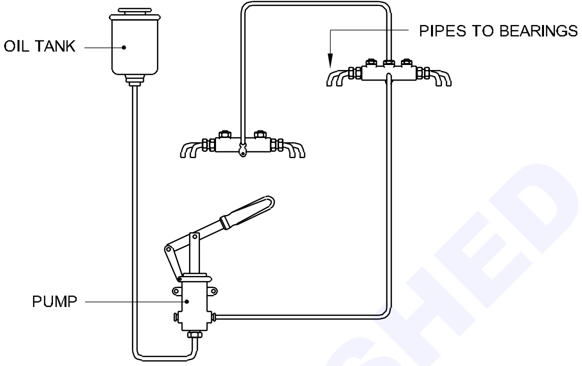
Fig 3



MANUAL SCREW DOWN GREASER

Oil is also pressure fed by hand pump and a charge of oil is delivered to each bearing at intervals once or twice a day by operating a lever provided with some machines. (Fig 4) This is also known as shot lubricator.

Fig 4



HAND PRESSURE FEED (ONE SHOT)

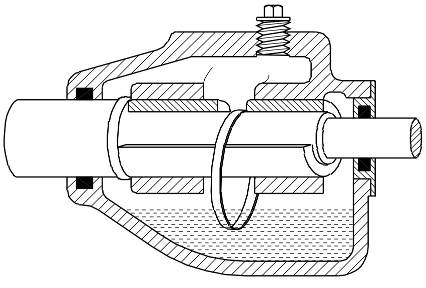
Oil pump method

In this method an oil pump driven by the machine delivers oil to the bearings continuously, and the oil afterwards drains from the bearings to a sump from which it is drawn by the pump again for lubrication.

Splash lubrication

In this method a ring oiler is attached to the shaft and it dips into the oil and a stream of lubricant continuously splashes around the parts, as the shaft rotates. The rotation of the shaft causes the ring to turn and the oil adhering to it is brought up and fed into the bearing, and the oil is then led back into the reservoir. (Fig 5) This is also known as ring oiling.

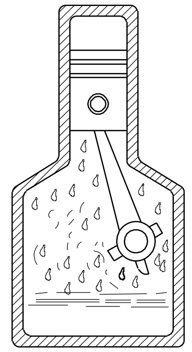
Fig 5



RING OILING

In other systems one of the rotating elements comes in contact with that of the oil level and splash the whole system with lubricating oil while working. (Fig 6) Such systems can be found in the headstock of a lathe machine and oil engine cylinder.

Fig 6



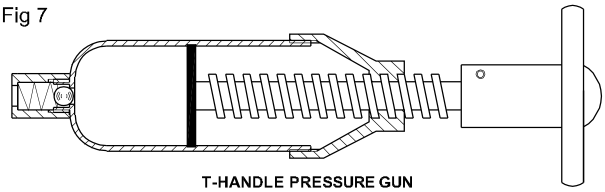
SPLASH-LUBRICATING SYSTEM

Types of grease guns

The following types of grease guns are used for lubricating machines.

- **'T' handle pressure gun** (Fig 7)

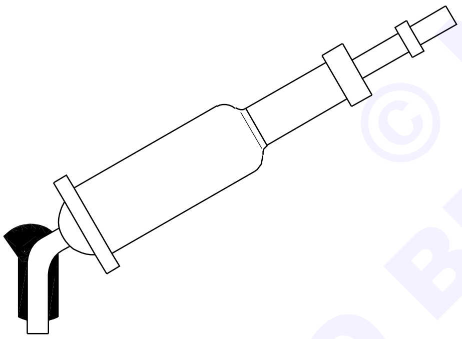
Fig 7



T-HANDLE PRESSURE GUN

- **Automatic and hydraulic type pressure gun** (Fig 8)

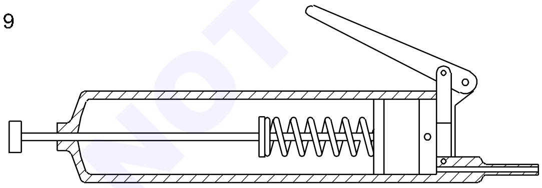
Fig 8



AUTOMATIC HYDRAULIC-TYPE PRESSURE GUN

- **Lever-type pressure gun** (Fig 9)

Fig 9



PRESSURE GREASE GUN

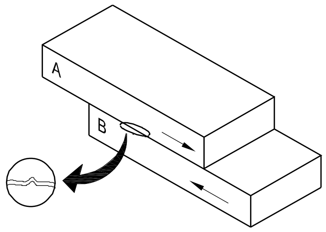
Lubrication to exposed slideways

The moving parts experience some kind of resistance even when the surface of the parts seems to be very smooth.

The resistance is caused by irregulartities which cannot be detected by the naked eyes.

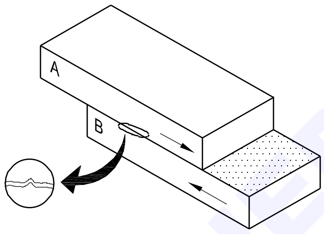
Without a lubricant the irregulartities grip each other as shown in the diagram. (Fig 10)

Fig 10



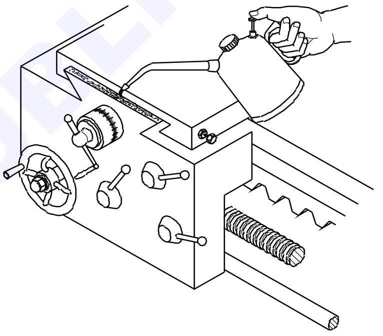
With a lubricant the gap between the irregularities fills up and a film of lubricant is formed in between the mating components which eases the movement. (Fig 11)

Fig 11



The slideways are lubricated frequently by an oilcan. (Fig 12)

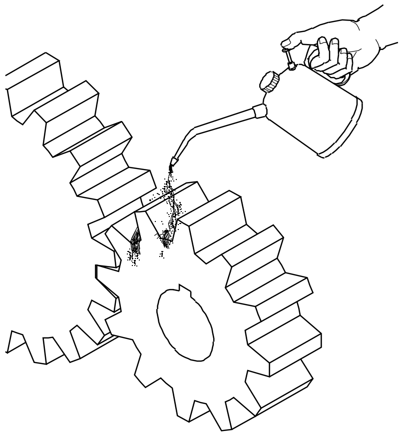
Fig 12



OIL THE SLIDES FREQUENTLY

After cleaning the open gears, oil them and repeat lubrication regularly. (Fig 13)

Fig 13

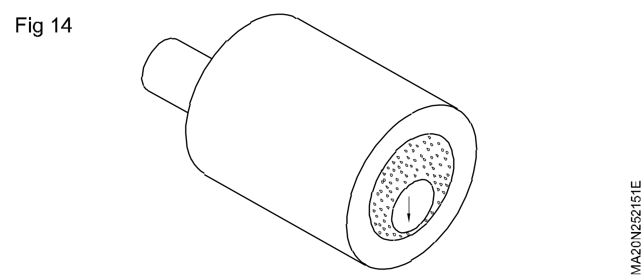


Lubricate bearings

A shaft moving in a bearing is also subjected to frictional resistance. The shaft rotates in a bush bearing or in ball/ roller bearing, experiencing friction.

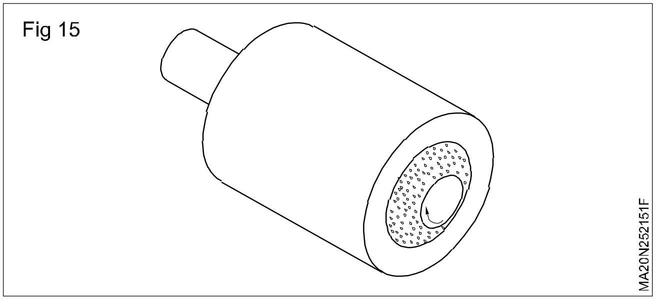
When the shaft is at rest on the bottom of the bush bearing, there is hardly any lubricant between the shaft and the bush. (Fig 14)

Fig 14



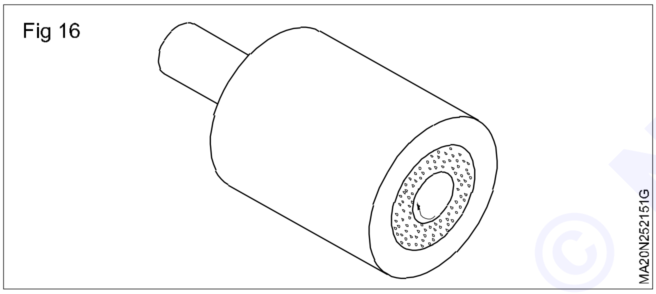
When the shaft starts rotation the lubricant maintains a film between the shaft and the bush and an uneven ring of lubricant builds up. (Fig 15)

Fig 15



When the shaft is rotating at full speed a full ring of lubricating film surrounds the shaft (Fig 16) which is known as hydro dynamic lubrication.

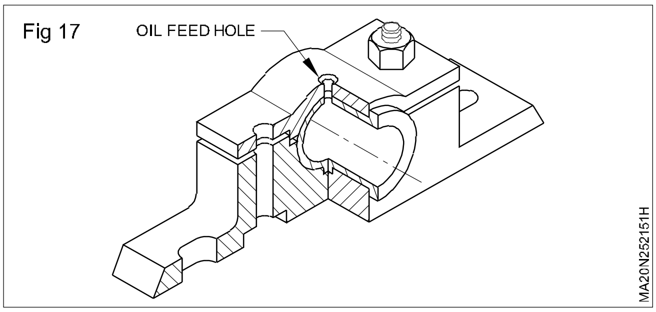
Fig 16



This lubrication ring decreases the frictional resistance very much and at the same time protects the mating members against wear and changes.

Some bush bearings have oil feeding holes over which the oil or grease cup is mounted and the lubricant is fed through the holes into the bearing by gravity feed system.(Fig 17)

Fig 17



Hints for lurbicating machines:

- identify the oiling and greasing points
- select the right lubricants and lubricating devices
- apply the lubricants.

The manufacturer’s manual contains all the necessary details for lubrication of parts in machine tools. Lubricants are to be applied daily, weekly, monthly or at regular intervals at different points or parts as stipulated in the manufacturer’s manual.

These places are indicated in the maintenance manuals with symbols.

The best guarantee for good maintenance is to follow the manufacturer’s directives for the use of lubricants and greases. Refer to the Indian Oil Corporation chart for guidance. The commonly used oils in the workshop is given in Annexure I.

The lubricant containers should be clearly labelled. The label must indicate the type of oil or grease and the code number and other details. Oil containers must be kept in the horizontal position while the grease container should be in the vertical position.

Automatic lubricating system

Objectives: At the end of this lesson you shall be able to
- **state the automatic lubricating system**
- **list the different types of automatic lubricating system**
- **brief the various types of automatic lubricating system.**

An Automatic Lubrication System, sometimes referred to as a Centralized Lubrication · System, is a system that delivers controlled amounts of lubricant to multiple locations on a machine while the machine is operating

There are several different types of automatic lubrication systems including:

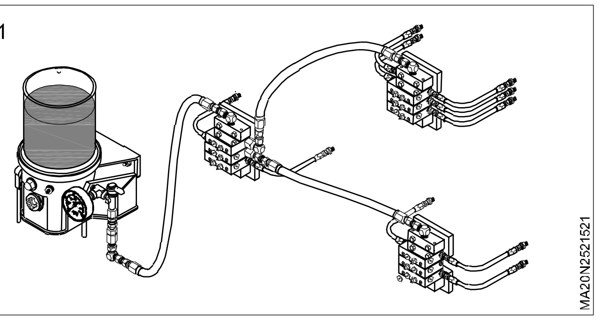
- Single Line Parallel systems .
- Dual Line Parallel systems.
- Single Point Automatics.
- Single Line Progressive systems (or Series Progressive)
- Single Line Resistance.
- Oil Mist and Air-Oilsystems.
- Oil re-circulating.
- Chain lube systems.

The 4 most commonly used Automatic Lubrication System types are:

- Single Line Parallel,
- Dual Line Parallel and
- Single Line Progressive.
- Multi port direct lubricators

Single line progressive (Fig 1)

Fig 1

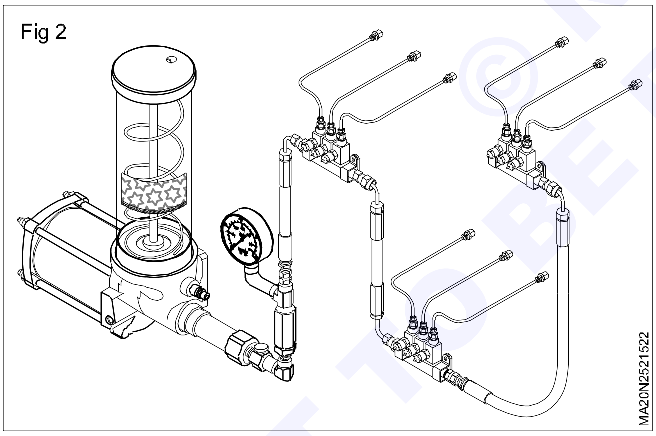


A single line progressive system uses lubricant flow to cycle individual metering valves and valve assemblies. The valves consist of dispensing pistons moving back and forth in a specific bore. Each piston depends on flow from the previous piston to shift and displace lubricant. If one piston doesn't shift, none of the following pistons will shift valve output is not adjustable.

Operation begins when the controller/timer sends a signal to the pump to start the lube event. The pump then feeds lubricant into the supply line which connects to the primary metering valve, for either a preprogrammed amount of time or number of times as monitored through a designated piston cycle switch. Lubricant is fed to the multiple lubrication points one after another via secondary progressive metering valves sized for each series of lubrication points, and then directly to each point via the feed lines.

Single line parallel (Fig 2)

Fig 2



A single line parallel system can service a single machine, different zones on a single machine or even several separate machines and is ideal when the volume of lubricant varies for each point. In this type of system, a central pump station automatically delivers lubricant through a single supply line to multiple branches of injectors. Each injector serves a single lubrication point, operates independently and may be individually adjusted to deliver the desired amount of lubricant.

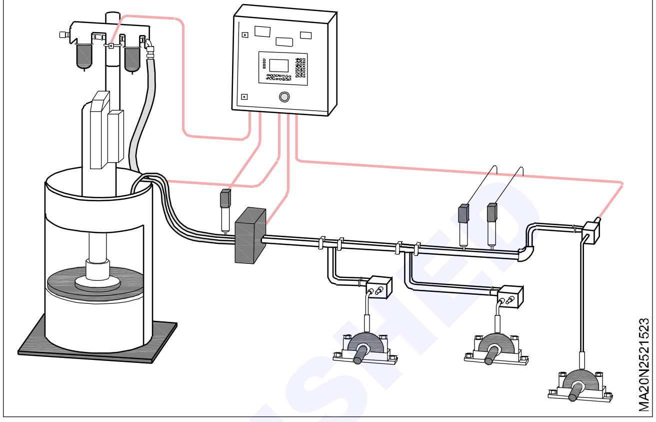
Operation begins when the controller/timer sends a signal to the pump starting the lube cycle. The pump begins pumping lubricant to build up pressure in the supply line connecting the pump to the injectors. Once the required pressure is reached, the lube injectors dispense a predetermined amount of lubricant to the lubrication points via feed lines.

Once the entire system reaches the required pressure, a pressure switch sends a signal to the controller indicating that grease has cycled through to all the distribution points.

The pump shuts off. Pressure is vented out of the system and grease in the line is redirected back to the pump reservoir, until the normal system pressure level is restored.

Dual line parallel (Fig 3)

Fig 3



A dual line parallel system is similar to the single line parallel system in that it uses hydraulic pressure to cycle adjustable valves to dispense measured shots of lubricant. It has 2 main supply lines which are alternatively used as pressure I vent lines.The advantage of a two-line system is that it can handle hundreds of lubrication points from a single pump station over several thousand feet using significantly smaller tubing or pipe.

Operation begins when the controller/timer sends a signal to the pump to start the lubrication cycle. The pump begins pumping lubricant to build up pressure in the first (the pressure) supply line while simultaneously venting the second (vent) return line. Once the required pressure is reached, a predetermined amount of lubricant is dispensed by the metering devices to half of the lubrication points via feed lines.

Once the pressure switch monitoring main supply line pressure indicates a preset pressure in the line has been reached, the system is hydraulically closed. The controller shuts off the pump and signals a changeover valve to redirect lubricant to the second main supply line.

The next time the controller activates the system, the second main line now becomes the pressure line while the first line becomes the vent line.The second line is pressurized and the entire process is repeated lubricating the remaining lube points.

Multi point direct lubricator

When the controller in the pump or external controller activates the drive motor, a set of cams turns and activates individual injectors or pump elements to dispense a fixed amount of lubricant to each individual lubrication point. Systems are easy to design, direct pump to lube point without added accessories and easy to troubleshoot.

Maintenance and its types

Objectives: At the end of this lesson you shall be able to
- **state the types of maintenance**
- **list the types of maintenance**
- **brief the each types of maintenance.**

Maintenance

Maintenance is a combination of any actions carried out to retain an item in, or restore it to an acceptable condition.

Types of maintenance

Breakdown maintenance

In breakdown maintenance the equipment is usually attended to only after it breaks down. Despite its numerous disadvantages this type of system may be suitable in certain conditions such as the equipment is non critical and standbys are available or the plant capacity exceeds market demand.

Preventive maintenance

Organising maintenance before the needs being developed would minimize the possibility of anticipated break downs.

Scheduled maintenance

Analysis of routine maintenance like cleaning greasing etc. which will keep equipments running efficiently and in a state of reliable operational readiness.

Reconditioning

At periodic intervals, depending on type and nature of the machine or equipment and its condition, it needs to be overhauled or reconditioned. It is the process of bringing the machine back to its new conditions.

Corrective maintenance

A study of failure of equipment in service may warrant a change in design; materials or working conditions of the equipment and corrective steps should be taken thereafter.

Maintenance prevention

While designing and developing the equipment objective is set to provide no maintenance or higher maintainability which would reduce the maintenance effort in the life time.

Predictive maintenance

While the equipment is in actual operating condition a study of performance of the equipment would reveal whether unexpected deterioration is taking place in it and the range of frequency of scheduled maintenance to reduce such deterioration.

Condition based maintenance

In this system of maintenance, the machine is continually monitored to see the health of machine derived from different electrical instrument, in the form of mechanical vibration, noise, sound, thermal emissions, change in chemical composition, small pressure and relative displacement and so on. From this result the severity of faults can be evaluated for decision making.

Pro- active maintenance

The most recent innovation in maintenance is called Pro-active Maintenance.

It includes a technique called "root causes failure analysis" in which the primary cause of the machine failure is sought and corrected.

The root causes of the majority of machine faults are imbalance and misalignment. Both of these condition place undue force on bearing, shortening their service life. Rather than continual · replacing worn bearing, a far better policy is, to perform precision balance and alignment on the machine and to verify the result by vibration analysis.

Total productive maintenance (TPM)

To improve the company, the attitudes and skills of all the personnel's from top management down to shop floor workers , the idea that, "I am the operator and you're the maintenance man" was deep rooted and the operator had no interest in maintenance.

Changing this idea on the part of all the employees and so the operator autonomously maintain the equipment he use by himself is the first step in TPM.

TPM takes on the challenge of

ZERO LOSSES

ZERO FAILURES

ZERO DEFECTS

System of symbol and colour coding for lubricants

Objectives: At the end of this lesson you shall be able to
- **state the necessity of the colour coding system for lubricants**
- **list the points that helps the management by colour coding**
- **brief the colour and shapes in colour coding.**

A color code is a system for displaying information by using different colors. The earliest examples of color codes in use are for long distance communication by use of flags, as in semaphore communication. ... On forms and signage, the use of colour can distract from black and white text.

The Ease and Simplicity of Color-Coded Lubrication Management

The idea behind color-coded lubrication management is simple: Everything that comes into contact with lubricant during the maintenance process is marked with a color and symbol that correspond to a particular lubricant. This means all the drums, dispensers, transfer containers, and fill points in a color-coded system have colored tags·or labels of some kind. Workers can easily see which items are to be used with which lubricant and where each lubricant should be applied. Following the path of any given lubricant from storage to application is as easy as connecting the dots -or colored symbols

Color-coded lubrication management helps you do the following

1 Prevent Lubricant Misapplication

2 Eliminate Cross-Contamination

3 Foster Workplace Safety

4 Safeguard Your Workflow

The colour of lubricating oils can range from transparent to opaque. The color is based on the crude from which it is made, its viscosity, method and degree of treatment during refining, and the amount and types of additives included.

In addition, each viscosity grade has its·own colour that can be standardized throughout the plant. For oils, the LIS tag is square-shaped, while for greases the tag is round.

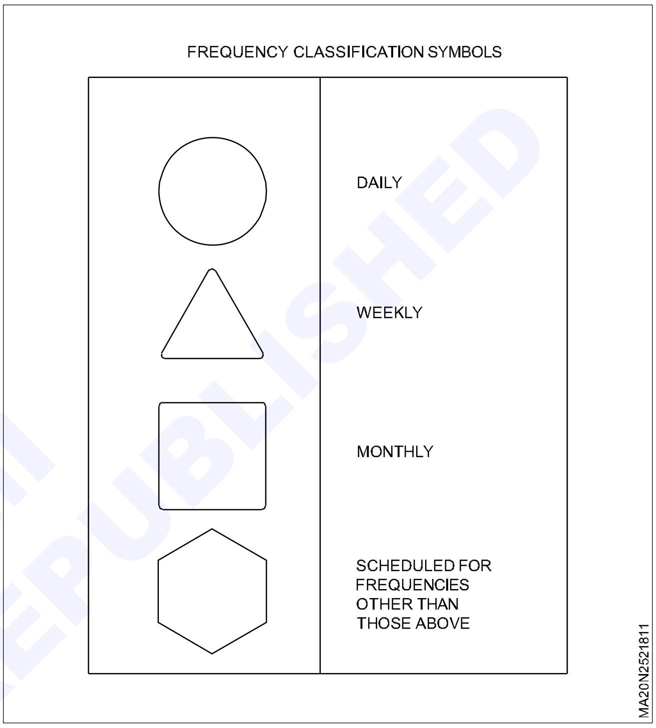
Example of the colour coding system is shown in Table 1

However

A circle to represent the need for daily lubrication, A triangle for weekly lubrication, and

A square to represent monthly intervals between lubrication activities.

Table 1



For activities conducted on a quarterly basis (or over longer periods), the square was to again be used, but this time with a number painted inside the square to highlight the number of interval months.

Lubricant storage and transfer systems, though, reflect just one area where colorization pays off for a site.Another important use of color identification involves a condition-based approach to filling oil reservoirs.

Fig 1 is a good example of this Hi-Lo technique. Itinvolves using red, amber (yellow), and green lines taped on the side of an automated-lubrication-system reservoir. Tills arrangement is known as a RAG (red/amber/green), or the traffic-light indicator system:

- The green line indicates the upper fill level.
- The amber (yellow) line indicates a level at whlch the operator is to contact the maintenance department with a first request to fill the reservoir.
- The red line alerts the operator to call ina priority request to fill the reservoir

Fig 1

General possible causes and remedies of equipment in industries

Objectives: At the end of this lesson you shall be able to
- **list out the causes for failures in equipments and machineries**
- **list out the remedies to rectify the failures.**

Regardless of the industry all equipment falls into three broad categories:

Powered Equipment: These are machines that rely on fuel or electric motors to operate. Also known as "heavy equipment," they include bulldozers, cranes, and forklifts.

Mechanical Equipment: Mechanical equipment usually includes moving components that work together to perform a specific task. Refrigeration units, condensers, extruders, conveyor belts, and air compressors are examples of mechanical equipment.

Non-Mechanical Equipment: Non-mechanical equipment is generally small and doesn't have any moving parts. Most hand-held tools fall into this category, including plumbing, gas lines, electric panels, and HVAC system tools.

Most frequent causes of machine breakdowns are

Aging Equipment

Assets that consistently run year after year require more frequent repairs over time. Unfortunately, this natural deterioration translates to more money spent on parts, shipment fees, and production interruptions. It also requires technicians to shift from practicing preventive maintenance (PM) to reactive maintenance more frequently.

Additionally, the more outdated a model becomes, the more difficult it becomes to procure replacement parts. Since manufacturers generally produce lower quantities of antiquated parts, managers of older equipment often have trouble securing what they need when they need it.

For these reasons, plants should allow pieces of equipment that have passed their useful life to run to failure. Though purchasing new machinery is undoubtedly expensive, most organizations will save money in the long run in the form of less frequent repair expenses, increased production, and more efficient parts management.

Operator Error

Another common cause of equipment failure is operator error. Both fortunately and unfortunately, human beings aren't machines! Consequently, we sometimes make mistakes due to fatigue or forgetfulness.

Most plants prepare machine operators to properly run complex pieces of equipment with educational training, accessible standard operating procedures (SOPs), and clear communication channels. However, it's not unheard of for workers to occasionally work on unfamiliar machines when filling in for others.

For example, maybe the machine operator who typically runs the machine had to take care of an emergency. As a result, someone asked an untrained worker to temporarily step in.

Not only could the worker's lack of specialized knowledge result in an equipment breakdown, but it could also cause an accident.

Lack of Preventive Maintenance

There's a reason why world-class maintenance programs predominantly practice preventive maintenance: it works to decrease downtime! Studies suggest PM programs can reduce equipment failure by as much as 45 percent.

However, many managers still operate under the adage, "If it ain't broke, don't fix it." While run-to-failure maintenance is often the ideal strategy for non-critical equipment, a lack of planned inspections can lead to missing subtle signs of impending failure and depreciating performance.

Run-to-Failure Maintenance vs. Preventive Maintenance

Run-to-failure maintenance is a maintenance strategy in which maintenance activities are performed after a failure occurs. It's a reactive strategy that allows an asset to break down before deciding whether to repair or replace it.

Also referred to as "breakdown maintenance," this strategy is best suited for non-critical assets, inexpensive assets, and those with shorter life spans. It's also sometimes used for equipment such as satellites that are difficult to perform regular maintenance on. However, run-to-failure maintenance should never be used for critical assets that impact production and safety.

Alternatively, preventive maintenance (PM) involves scheduling and performing recommended upkeep according to time or usage-based intervals. It's a proactive strategy best suited for expensive assets, critical assets, and components that are essential to employee safety.

Ultimately, organizations that solely practice run-to-failure miss out on solving minor issues before they become big problems. Additionally, frequently missed or delayed tasks, like periodically lubricating equipment parts, can reduce useful life spans over time. The result? Needing to frequently purchase new pieces of equipment earlier than necessary.

Over-Maintenance

On the other hand, performing too much maintenance can also be detrimental. Though less common, over-maintenance can speed up an asset's depreciation.

Constantly taking apart and reconfiguring equipment components can disrupt even the most stable systems, causing them to become less effective.

Additionally, maintenance technicians may be more likely to "go through the motions," viewing frequent tasks as yet another item to be checked off. Lastly, over-maintenance steals time away from busy technicians that could be put to better use elsewhere!

Not enough PMs	Just enough PMs	Too much PMs
Signs of early failure are missed	Costs are controlled	Overspending increases
Maintenance schedules suffer Reduced equipment efficiency	Just-in-time delivery is achieved Downtime is cut	Wear and tear on asset soars Technician time is wasted
Asset lifespan shrinks Safety improves	Technician efficiency is boosted Unnecessary inventory is	Breakdowns and downtime rise used
Costs skyrocket	Productivity of critical assets is optimized	Inaccurate information is collected

The Solution to Equipment Failure

As previously mentioned, equipment failure can be extremely costly depending on the industry. Here are a few tips to prevent or minimize equipment failure:

Provide Adequate Operator Training: Ensuring equipment operators receive adequate training can help minimize failures that stem from human error. Managers should go above and beyond to train workers how to use machines they don't normally operate. This way, someone will always be available who has been properly trained according to OSHA standards. Likewise, management must strictly enforce policies that prevent untrained employees from operating machinery.

Develop an Effective PM Strategy: Regular PM activities help prevent failures and extend asset life spans. The secret to running an effective PM program lies in inventory control, data management, and effective scheduling. A mobile-friendly computerized maintenance management system (CMMS) like MaintainX is the easiest way to begin.

Perform Regular Inspections: Performing regular asset inspections remains one of the most effective ways to identify problems early on. Inspections should be thorough and not done just for the sake of checking off your checklists.

Embrace Condition-Based Maintenance (CBM): CBM is the key to dealing with over-maintenance. Maintenance is performed only when it's needed, just before failure occurs. Since this maintenance strategy requires organizations to monitor asset activity such as vibrations with real-time sensor technology, it's best suited for maintenance departments with large budgets.

Invest in Maintenance Management Software: Simplify, and track all organizational maintenance activities and data points from the convenience of their smart phones.

Bevel gear - Elements, types, application and calculation

Objectives : At the end of this lesson you shall be able to
- **state the elements of a bevel gear**
- **name the elements of a bevel gear in the drawing**
- **define the element of a bevel gear.**

Bevel gear

A bevel gear is a gear in which the teeth are cut on a conical surface ie. the teeth are inclined towards its axis. The teeth are radiating from the apex. Bevel gears are cut on conical blanks. The tooth size decreases from the back end towards the apex. The elements such as addendum, dedendum, whole depth, pitch diameter and outside dia. are measured with respect to the large end. All these proportions except the outside dia. are the same as for the spur gear. The definitions of all the above elements are also the same as for the spur gear. (Fig 1 & 2)

Fig 1

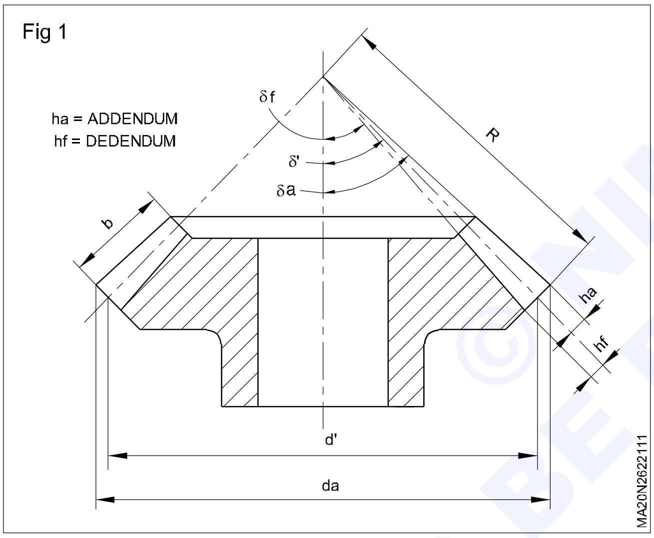
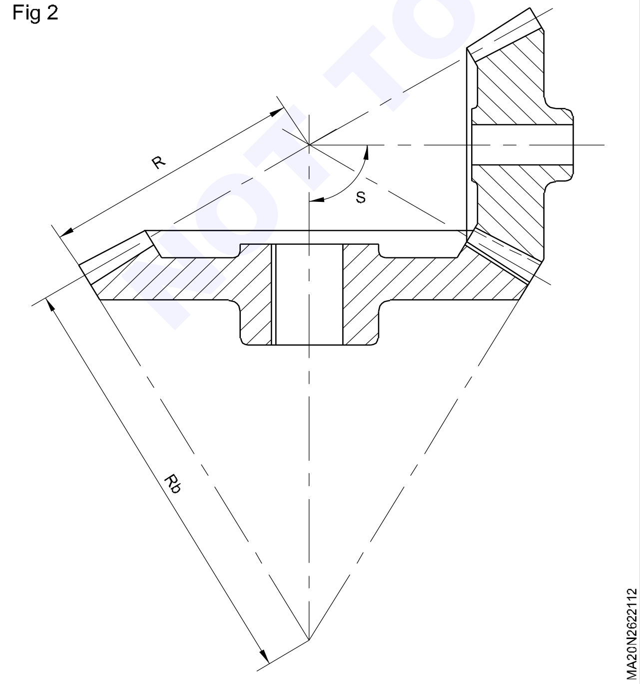


Fig 2



The following are the other elements with respect to bevel gears.

Pitch cones

These are the cones which roll without slipping when they are in peripheral contact. These are same like pitch cylinders of spur gears.

Pitch cone angle (δ')

This is the angle subtended at the apex by the axis and the pitch cone generator.

The cone distance (R)

This is the length of pitch cone generator from the pitch circle to the apex. It is generally denoted by ‘R’.

Back cone (Rb)

This is a cone generated by a line which is perpendicular to the pitch cone generator at the pitch point on the pitch circle. It is denoted by ‘Rb’.

Face angle (δa)

This is the angle between the axis and the top surface of the tooth. This angle is equal to the sum of the pitch cone angle and the addendum angle. It is denoted by ‘da’. This is the angle to which the blank has to be turned.

The shaft angle (S)

The angle between the intersecting axes of mating gears is the shaft angle. It is normally denoted by ‘S’. It is the sum of the pitch the cone angles of the mating gears.

Root angle(δf)

This is the angle between the axis and the root surface of a tooth space. It is equal to the pitch cone angle minus the dedendum angle. It is denoted by ‘df’. For this angle the index head is set to cut the teeth on the bevel gear.

Face width (b)

This is the length of the tooth measured along the pitch cone generator. It is denoted by ‘b’. Generally b = 1/3 R to 1/4 R.

Addendum angle (θa)

It is the angle between the pitch cone generator and the tip surface of the tooth. It is denoted by ‘ha’.

Dedendum angle(θf)

It is the angle between the pitch cone generator and the root surface of a tooth space. It is denoted by ‘hf’.

Bevel gear tooth (straight) proportions and calculation

Objectives : At the end of this lesson you shall be able to
- **list the elements of a bevel gear in the drawing**
- **calculate the proportions of bevel gear elements.**

BEVEL GEAR PROPORTIONS AND CALCULATIONS **(20° pressure angle)**

Description	**Symbol**	**Proportions in terms of module 'm'**	**Proportions in terms of diametral pitch 'DP'**
Pitch diameter	d'	Zm	Z/DP
Outside (or) Tip circle diameter	da	m (Z + 2Cos δ')	$\frac{Z + 2\cos\delta'}{DP}$
Addendum	ha	m	$\frac{1}{DP}$
Dedendum	hf	1.25m	$\frac{1.157}{DP}$
Working depth	2ha	2m	$\frac{2}{DP}$
Total depth	ha+hf	2.25m	$\frac{2.157}{DP}$
Clearance	hf–ha	0.25m	$\frac{0.157}{DP}$
Cone distance	R	$\frac{d^1}{2} \div \sin\delta^1$	
Face width	b		b max = $\frac{R}{3}$ min = $\frac{R}{4}$
Addendum (Small end)	–	$\frac{R-b}{R} \times ha$	
Dedendum (Small end)	–	$\frac{R-b}{R} \times hf$	
Pitch cone angle	δ^1	$\tan\delta^1 g = \frac{Zg}{Zr}$	
Addendum angle	θa	$\frac{R}{M} = \cot\theta_a$	
Dedendum angle	θf	$\frac{R}{1.25m} = \cot\theta_1$	
Face angle	δa	$\delta^1 + \theta a$	
Root angle	δf	$\delta^1 - \theta f$	
Back cone distance	Rb	$R \tan\delta'$	
Virtual number of teeth (For cutter selection)	Z'	$Z/\cos\delta^1$	

Offset for the first flank = $\frac{R - b}{R} \text{x} \operatorname{Sin}\frac{90}{Z}$

First angular movement of the blank = $\left(\frac{90^\circ}{Z}\right)$

Offset for the second flank = $\frac{R - b}{R} \text{x} \operatorname{Sin}\frac{(180^\circ)}{Z}$

Second angular movement of the blank (from first offset position) = $\left(\frac{180^\circ}{Z}\right)$

Calculation of proportions of bevel gear

Example

Calculate all the elements with their std. proportions required for milling a bevel gear, given module = 3, number of teeth = 24 and pitch cone angle = 45°.

Solution

Pitch diameter

= d' = Zm

= 24 x 3 = 72 mm

Outside/tip circle diameter

= da = m(Z+2Cosd^1)

= 3(24 + 2 x Cos45°)

= 3(24 + 2 x 0.7071)

= 3(24 + 1.4142)

= 76.2426 mm - 76.24 mm

Addendum (large end)

= ha = m = 3 mm

Dedendum (large end)

= hf = 1.25 x 3 = 3.75 mm

Working depth

= 2ha = 2 m = 2 x 3 = 6mm

Total depth

= ha + hf = 3.00 + 3.75 = 6.75 mm

Clearance

= hf – ha = 3.75 – 3.00 = 0.75 mm

Cone distance

$$(R) = \frac{d'}{2\operatorname{Sin}\delta} = \frac{72}{2 \times \operatorname{Sin}45} = \frac{72}{1.4142}$$

= 50.9121 mm = 51 mm

Face width

$$= b = \frac{R}{3} = 16.9707 \text{ mm} = 17 \text{ mm (maximum)}$$

Addendum (small end)

$$= \frac{R - b}{R} \text{x} \, ha = \frac{51 - 17}{51} \text{x} \, 3 = \frac{34 \text{x} 3}{51} = \frac{102}{51} = 2\text{mm}$$

Dedendum (small end)

= Addendum angle

$$= \operatorname{Tan} \theta a = \frac{M}{R}$$

$$= \operatorname{Tan} \text{x} \frac{3}{51} = 0.0588 = 3^\circ 22'$$

Dedendum angle

$$= \theta f = \tan \text{x} \frac{hf}{R} = \tan \text{x} \frac{3.75}{51}$$

= tan x 0.0735 = 4°12'

Face angle

= $\delta a = \delta^1 + \theta a$ = 45° + 3°22' = 48°22'

Root angle (Setting angle)

= $\delta f = \delta^1 - \theta f$ = 45° – 4°12' = 40°48'

Back cone distance

= Rb = R x tan δ^1 = 51 x tan 45°

= 51 x 1 = 51 mm

Virtual number of teeth(for cutter selection)

$$= Z^1 = \frac{Z}{\operatorname{Cos}\delta^1} = \frac{24}{\operatorname{Cos}45^\circ} = \frac{24}{.7071}$$

= 33.94 or 34

Therefore, cutter No.3 is to be selected.

Offset for the first flank

$$= \frac{R-b}{R} x \sin\frac{90°}{2}$$

$$= \frac{51-1}{R} x \sin\frac{90°}{24} = \frac{34}{51} x \sin 3° 45'$$

$$= \frac{34}{51} x\ 0.0654$$

= 0.0436 mm

First angular movement of the blank

$$= \frac{(90°)}{Z} = \frac{90}{24} = 3°45'$$

Offset for the second flank (from first off set position)

$$= \frac{R-b}{R} x \operatorname{Sin}\frac{(180)}{Z}$$

$$= \frac{51-17}{51} x \operatorname{Sin}\frac{(180)}{24}$$

$$= \frac{34}{51} x \operatorname{Sin} 7°30' = \frac{34}{51} x\ 0.1305$$

= 0.087 mm

Second angular movement of the blank (from first offset position

$$= \frac{(180)}{(24)} = \frac{180}{24} = 7°30'$$

Bevel gears types and applications

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

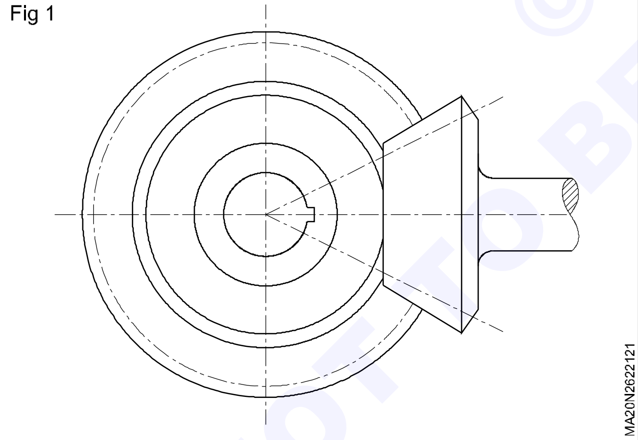
- **state the types of bevel gears**
- **state the applications of bevel gears.**

Types and applications

According to the tooth surface and the use, the bevel gears are classified as follows.

Straight tooth bevel gears (Fig 1)

Fig 1



Straight bevel gears have straight teeth. Straight bevel gears are used to connect the shafts at an angle and the centres intersecting. If the angle of inclination is 90° and the gears are of the same size, the gears are called “Mitre” gears. (Fig 2) If the angle of inclination is more or less than 90°, the gears are called angular bevel gears. (Fig 3) Straight bevel gears are used in drilling, milling, shaping and lathe machines to transmit power.

Fig 2

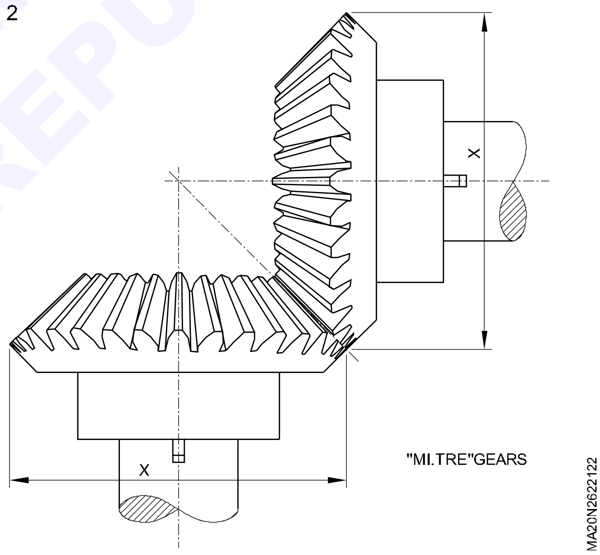
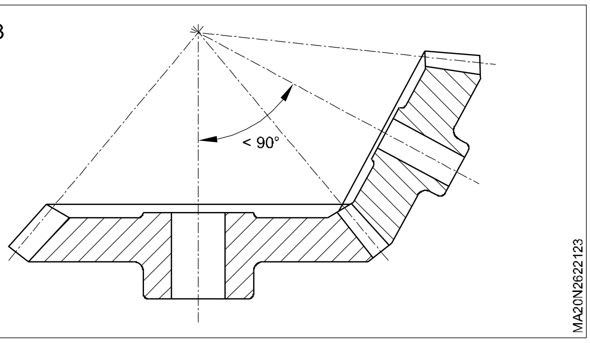


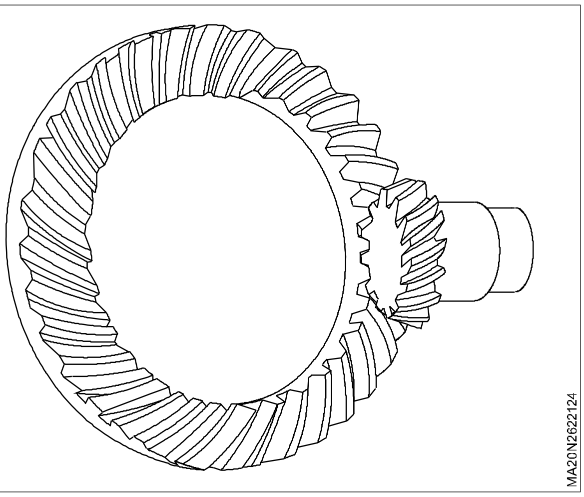
Fig 3



Spiral bevel gears (Fig 4)

It is a bevel gear with the teeth curved longitudinally along its length. It is also used to connect the shafts whose axes are intersecting. The advantage of this type over straight tooth bevel gear are a smooth operation, high speed range, more torque transmission.

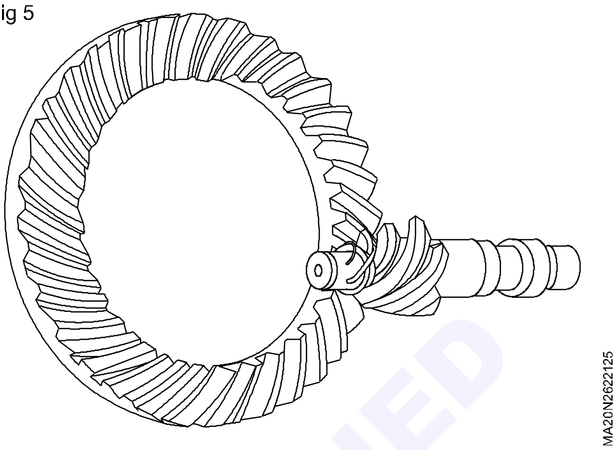
Fig 4



Hypoid gears (Fig 5)

This type of gears is used to connect shafts which are at an angle but not intersecting. Hypoid gears are mainly used in automobile drives.

Fig 5



Variation

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

- **define the variation**
- **state the different types of variation**
- **state the different types of errors.**

Variation

Variation seems inevitable in nature. Manufacturing process arc no expectation to thin. Variability this appears to be an in her pends on various characteristic of production process such as machine, the material the operation, The purpose of statistical analyse is to examine this variability and to detect when and to what extent external factor enter into the process to after the variability pattern.

Types of variations on gear

Composite testing of gears consists in measuring the variation in centre distance when a gear is rolled in tight mesh (double flank contact) with a specified or master gear. In composite gear checking two types of checking are made:

1 Total composite variation,

2 Tooth to Tooth composite variation.

Total composite variation is the centre distance variation in one complete revolution of the gear being inspected; whereas tooth to tooth composite variation is the centre distance variation as the gear is rotated through any increment of 3607 JV.

A uniform tooth to tooth variation shows profile variation whereas a sudden jump indicates the pitch variations. Composite type of checking takes care of all the errors in the gears. It is specially very much suited for large gears as it also ensures control over the tooth spacing. The composite method of checking is very much suitable for checking worn gears.

Cause of variation

Random causes that cannot be identified.

An important task in quality control is to find out the range of natural random variation in a process. For example, if the average bottle of a soft drink called cocoa fizz contains 16 ounces of liquid, we may determine that the amount of natural variation in between 15.8 and 16.2 ounces. If this were the case, we would monitor the production process to make sure that the amount stays within this range. If production goes out of this range-bottles are found to contain on average 15.6 ounces-this would lead us to believe that there is a problem with the process because the variation is greater than the natural random variation.

Assignable causes of variation

Causes that can be identified and eliminated

The second type of variation that can be observed involves variations where the causes can be precisely identified and eliminated. These are called assignable causes of variation. Examples of this type of variation are poor quality in raw materials, an employee who need more training, or a machine in need of repair. In each of these examples the problem can be identified and corrected. Also, if the problem is allowed to persist, it will continue to create a problem in the quality of the product. In the example of the soft drink bottling operation, bottles filled with 15.6 ounces of liquid would signal a problem. The machine may need to be readjusted. This would be an assignable cause of variation. We can assign the variation to a particular cause

Class or Grade of Gear	Total composite Error in Microns	Tooth or Tooth Composite Errors
		in Microns
1	4+0.32 F	2+0.16 F
2	6+0.30 F	3+0.224 F
3	10+0.08 F	4+0.32 F
4	16+1.25 F	6+0.45 F
5	25+2.0 F	9+0.56 F
6	40+3.2 F	12+0.90 F
7	56+4.5 F	16+1.25 F
8	71+5.6 F	22+1.8 F
9	90+7.1 F	28+2.24 F
10	112+9.0 F	36+2.8 F
11	140+11.2 F	45+3.55 F
12	140+11.2 F	45+3.55 F

(machine needs to be readjusted) and we can correct problem. (readjust the machine)

Tolerance for composite errors

The following table gives the tolerance on total composite errors and tooth to tooth composite error.

Here factor F = M + 0.25$\sqrt{D}$

Master gears

Master gears are made with sufficient accuracy capable of being used as the basis for comparing the accuracy of other gears. These are mostly used in composite errors determination in which the master gears are rotated in close mesh (double flank) or in single contact with the gears under test. These can also be used for calibration of gear checking instruments used in shop-floor. Master gears are generally of two types i.e. Master gears type A used for checking precision gears of accuracy class up to 7 and type B master gears used for checking gears of grades from 8 to 12 master gear are made from chromium manganese steel or good quality gauge steel and are hardened to 62 HRC. These are properly stabilised to relieve internal stresses. The master gears should preferably have lower module values because with coarse pitches the master gear would have either a very few teeth or else it will be quite big making it difficult to handle besides high-production cost.

Gear errors. Various possible types of error on spur, helical, bevel and worm gears are described below:

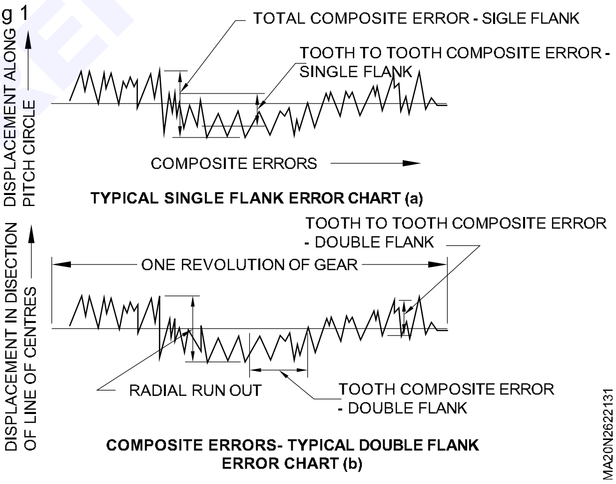
i) Adjust pitch error Actual pitch-design pitch.

ii) Cumulative pitch error Actual length between corresponding flanks of teeth not adjacent to each other-design length. (Hoi) Profile error The maximum distance of any point on the tooth profile form and normal to the design profile when the two coincide at the reference circle.

iv) The tooth to tooth com- The range of difference between the displacement at the posite error- single flanks pitch circle of a gear and that of a master gear meshed with it at fixed centre when moved through a distance (Fig 1). Corresponding to one pitch with only the driving and driven flanks in contact.

v) The total composite The range of difference between the displacement at the errors-single flank pitch circle of a gear and that of a master gear meshed with it at fixed centre distance when moved through one revolution with only the driving and driven flanks in contact.(Fig 1)

Fig 1



TYPICAL SINGLE FLANK ERROR CHART (a)

COMPOSITE ERRORS- TYPICAL DOUBLE FLANK ERROR CHART (b)

1	The tooth to tooth Composite error- double Flank	The range of variation in the minimum centre distance Between a gear and a master gear when rotated through A distance corresponding to the pitch of the teeth (Fig 1a)
2	The total composite Error- double flank	The range of variation in the minimum centre distance Between a gear and master gear when the gear is rotated through one revolution (Fig 1b)
3	The tooth thickness error	Actual tooth thickness measured along the surface of the reference cylinder-design tooth thickness.
4	Cyclic error	An error occurring during each revolution of the element under consideration.

5	Periodic error	An error occurring at regular intervals not necessarily corresponding to one revolution of the component.
6	Run out	It is the total range of reading of a fixed indicator with the contact point applied to a surface rotated, without axial movement about a fixed axis.
7	Radial run out	It is the run-out measured along a perpendicular to the axis of rotation.
8	Eccentricity	It is half the radial run-out
9	Axial run-out (wobble)	It is the run-out measured parallel to the axis of rotation, at a specified distance from the axis.
10	Undulation	A periodical departure of the actual tooth surface form the design surface (Fig 1b)
11	Undulation height	The normal distance between two surfaces that contain respectively crests and the troughs of the tooth undulation
12	Wavelength of an undulation	The distance between two adjacent crests of an undulation
13	Tooth alignment error	The distance of any point on a tooth trace from the design tooth trace passing through a selected reference point on that tooth

Gear measurement & Errors

Objectives : At the end of this lesson you shall be able to
- **state the methods of manufacturing gears**
- **list the different types of gear errors**
- **brief the uses of a flange micrometer.**

Methods in manufacturing gears and errors

1 Reproducing method - the cutting tool is a formed involute cutter, which forms the gear teeth profiles by reproducing the shape of the cutter itself each tooth space is cut independently of the other tooth spaces.

Gear errors

Due to: i) incorrect profile of the cutting tool, ii) incorrect positioning of the tool wrt the work, iii) incorrect indexing of the blank.

2 Generating method - the cutting tool (hob) forms the profiles of several teeth simultaneously during constant relative motion of the tool and blank.

Gear errors

Due to: i) errors in the manufacture of the cutting tool. ii) error in the relative motion during the generating.

Gear measurement

1 Teeth concentricity

Checked by:

- Mounting the gear between the bench center, placing a standard roller in each tooth space and using dial indicator.
- Using projector-, matching the tooth image with a line on the screen.
- If teeth are not concentric - flutuating velocity

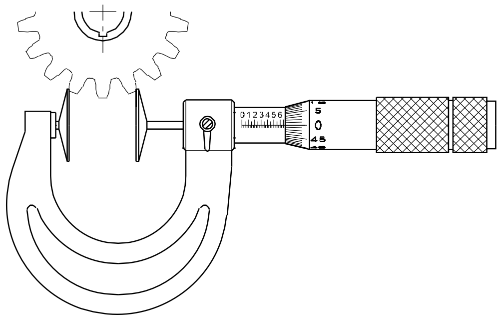
2 Good alignment of each tooth

- 1st method = checked by placing a standard rooler in the tooth space and checking for parallelism off a surface plate.
- 2nd method = teeth on one gear are lighly marked with prussion blue and mounted in a testing machine having a master gear.

3 Flange micrometer

A flange micrometer is similar to a regular micrometer and is equipped with two flanges in the place of the anvil and spindle. This is used to measure chordal thickmess of the gear teeth and the thickness of the fins of an engine and the collar thickness of the job.

Fig 1



FLANGE MICROMETER

Cams - types - application and elements

Objectives : At the end of this lesson you shall be able to
- **state the function of cams**
- **explain the types of cams and their applications**
- **define the elements of a cam.**

Cams

A cam is a machine element with a surface or groove formed to produce special or irregular motion to another part called a follower.

Types of cams

Cams are classified according to:

- the direction of motion of the follower with respect to the axis of the cam
  - movement of the follower in the direction perpendicular to the cam axis, known as Discardial cam
  - movement of the follower parallel to cam axis known as cylindrical cam.
- the method by which the follower is moving

Examples

- plate or radial cam (Fig 1)

Fig 1

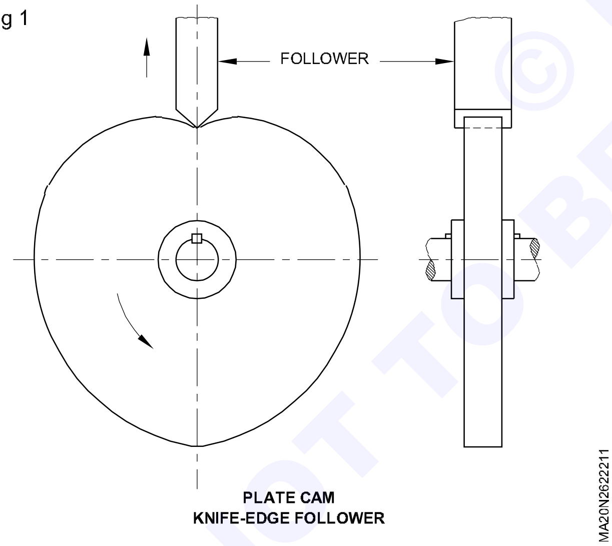
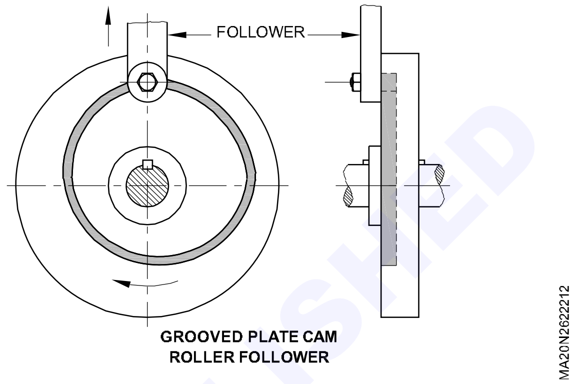


PLATE CAM
KNIFE-EDGE FOLLOWER

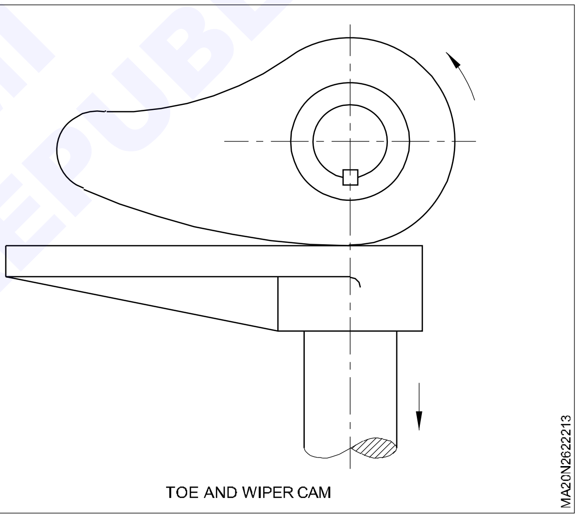
- grooved plate cam (Fig 2)
- Toe and Wiper cam (Fig 3)
- Cylindrical or drum cam (Fig 4)
- End or crown cam (Fig 5)

Fig 2



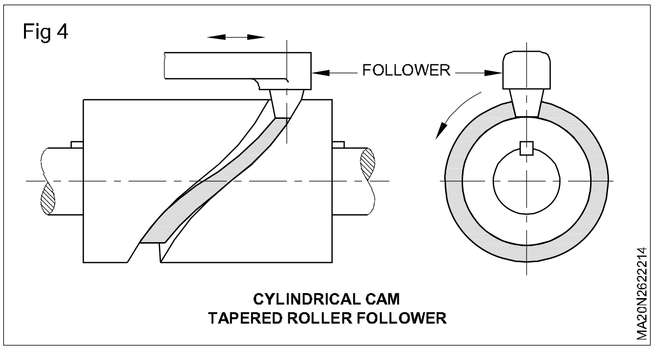
GROOVED PLATE CAM
ROLLER FOLLOWER

Fig 3



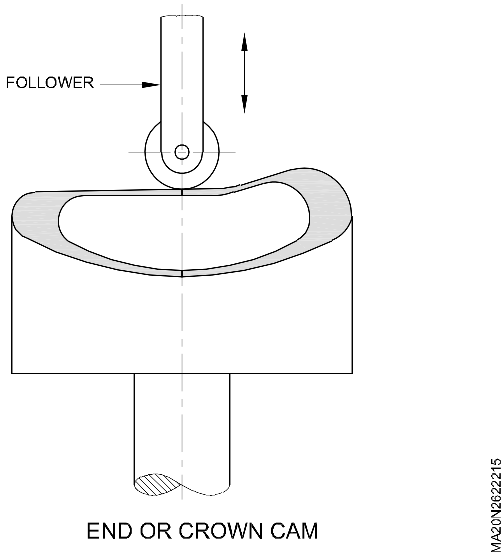
TOE AND WIPER CAM

Fig 4



CYLINDRICAL CAM
TAPERED ROLLER FOLLOWER

Fig 5



END OR CROWN CAM

The follower remains engaged in the groove on the face or periphery of the cam. These are generally called positive cams.

- Cylindrical or drum cam
- Grooved plate cam

Cam pushes the follower depending on some external forces. Such as gravity or springs, to keep the follower bearing against cam surface.

These are generally called non-positive cams.

- Plate cam
- Toe and wiper cam
- End or Crown cam

Elements of Cam

The following terms/elements used in milling radial cams.

Lobe

A lobe is a projecting part of the cam which imparts a reciprocating motion to the follower. Cams may have one or several lobes, depending on the application to the machine.

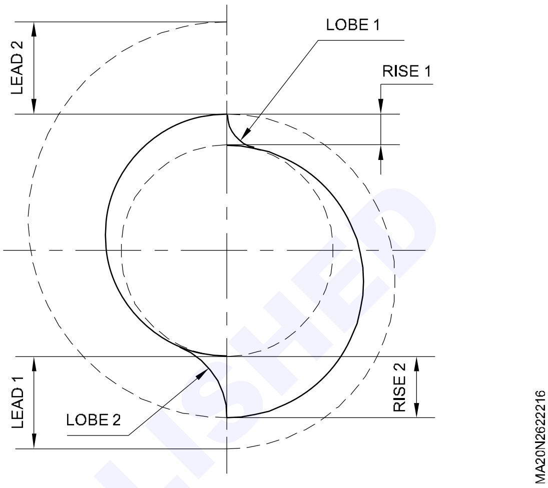
Rise

Rise is the distance one lobe will raise or lower the follower as the cam revolves. (Fig 6)

Lead

Lead is the total travel which would be imparted to the follower in one revolution of a uniform-rise cam, having only one lobe in 360°. (Fig 6)

Fig 6



$$\text{Lead} = \frac{\text{rise of lobe x } 360^{o}}{\text{number of degress of circumstances occupied by the lobe}}$$

The lead for a double lobe cam is twice the lead of a single lobe cam having the same rise.

It is the lead of the cam and not the rise that controls the gear selection in cam milling.

Uniform rise

Uniform rise is the rise generated on a cam which moves inward at an even rate around the cam assuming the shape of an Archimedes spiral. This is caused by uniform feed and rotation of the work when a cam is being machined.

Plate cams - Manufacturing and calculation

Objectives: At the end of this lesson you shall be able to
- **state the purpose of a plate cam**
- **state the methods of producing a plate cam**
- **calculate the crank movement of the indexing head.**

Cams are components used in all kinds of machinery to impart a reciprocating motion to cam ‘followers’ in accordance with the requirements of the individual application.

To satisfy these conditions, cams are made in a variety of shapes and contours.

- They may be made for mounting on a shaft to transform rotary cam motion into a reciprocating motion of the follower.
- They may be developed linearly for transmitting linear cam motion into a reciprocating motion of the follower.

The cam contour may be produced on the periphery of a disc or a cylinder, the face of a disc or the edge of a plate. These are known as radial, drum, face, and plate or bar cams, respectively.

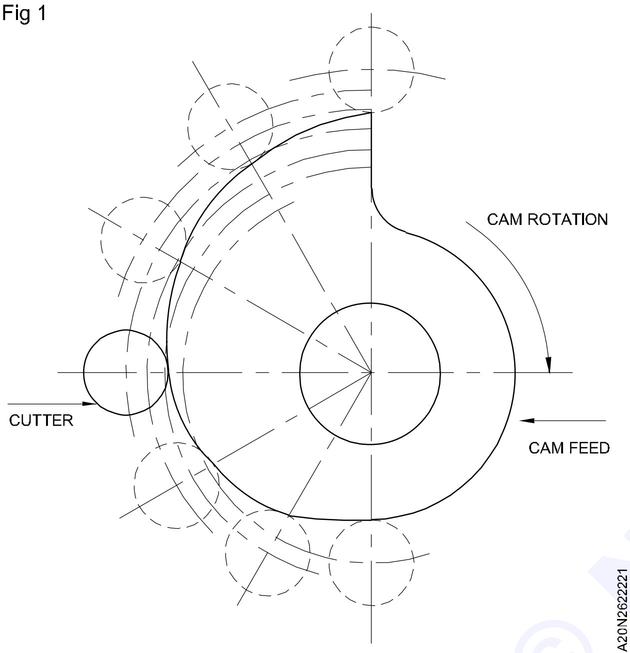
Plate or bar cams are also referred to as templates. When the contour is on an outer edge, the plate is known as an external template; if the contour is on the inside of the plate, it is called an internal template.

Templates having a contour cut to the outline of a part are employed for gauging purposes, or for guiding the path of a milling cutter, as in the case of rise and fall and tracer controlled milling machines.

Manufacturing of cam

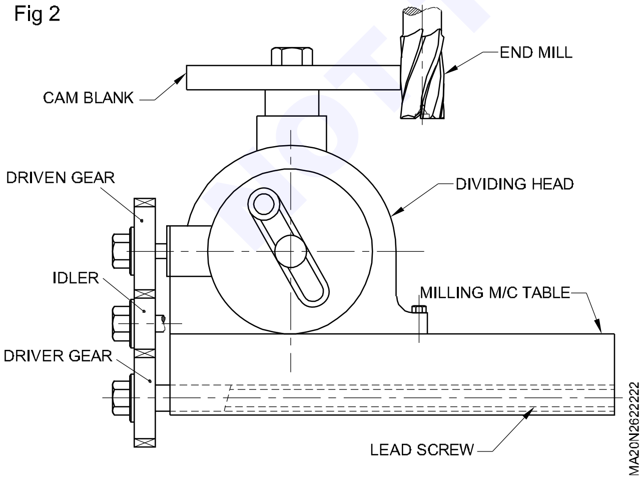
Plate cams (Fig 1) can be milled using a universal dividing head geared to the table lead screw of a vertical milling machine. The milling procedure is more or less same as for spiral milling. As the table feeds the cam blank into the cutter (end mill) the dividing head rotates the blank and the cam is generated. The gear ratio fixes the lead.

Fig 1



The cam axis is set perpendicular to the table. The arrangement shown in Fig 2 can be used for milling cams. In this set up, as the table advances and the cam blank is rotated, the centre distance between the dividing head spindle axis and the end mill axis is gradually reduced. This makes the radius of the cam to be reduced, and it produces a spiral lobe with a lead according to the ratio at which gears are connected.

Fig 2



$$\text{Gear ratio} = \frac{\text{driver}}{\text{driven}} = \frac{\text{lead of machine}}{\text{rise per revolution of cam}}$$

Example 1

Calculate the gear ratio to cut a cam that has a rise of 15 mm in 90° rotation if the table lead screw has a pitch of 6 mm.

Lead of machine = 40 x p = 40 x 6 = 240 mm

Rise of cam per revolution = 15 x 360°/90° = 60 mm

$$\text{Gear ratio} = \frac{\text{driver}}{\text{driven}} = \frac{\text{lead of machine}}{\text{rise per revolution of cam}}$$

$$= \frac{240}{60} = \frac{4}{1}.$$

The ratio of rise to lead rarely works out so conveniently like example 1 in practice. In practice it is difficult to get a clear or convenient ratio of rise to lead. Some means has to be used to obtain intermediate values from the standard gears supplied.

The rise generated on the cam is maximum for any given gear ratio in the perpendicular set up shown in Fig 2. However, with the dividing head and cutter set horizontally (Fig 3), only a cylindrical surface will be produced and the rise will be zero. Therefore, an intermediate set up is required to get the required rise. The intermediate arrangement is the angular setting of both the dividing head and the machine spindle. (Fig 4) The gear ratio then selected gives a rise larger than that required, but the spindle head and the dividing head are inclined to reduce to the actual rise required.

Fig 3

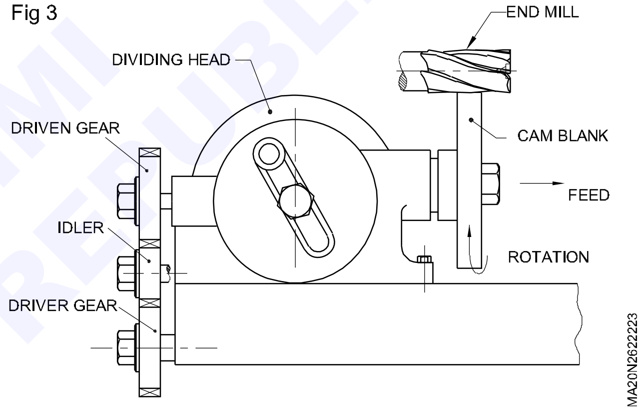
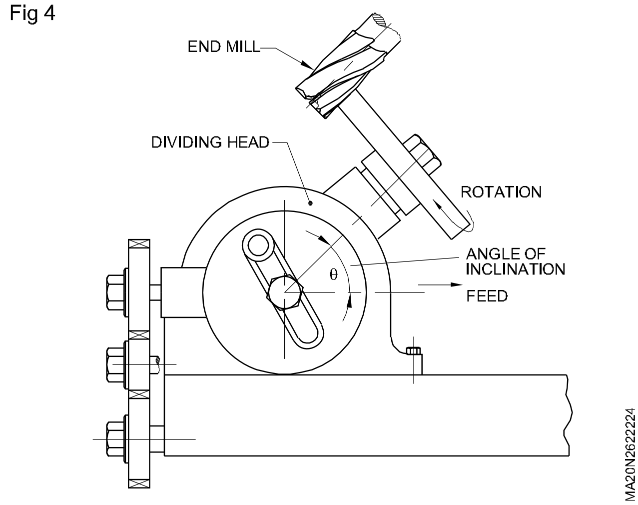


Fig 4



Angle of inclination = θ

$$\text{Sin } \theta = \frac{\text{Cam rise or lift per rev(x)}}{\text{Table movement per revolution of cam(L)}}$$

$$= \frac{\text{Rise of the cam}}{\text{Lead of the table}}$$

$$\frac{\text{Driver}}{\text{Driven}} = \frac{\text{Lead of machine}}{\text{Table movement per revolution of cam(L)}}$$

Example 2

A cam whose rise is 23.5 mm in 83° is to be cut using a vertical milling machine.

Calculate the

(i) gear ratio

(ii) inclination of the spindle.

$$\text{Lift of cam per revolution} = \frac{23.5 \times 360^\circ}{83^\circ}$$

$$= 101.9 \text{ mm/rev.}$$

The nearest gear ratio greater than this rise would be:

$$\frac{\text{Driver}}{\text{Driven}} = \frac{\text{Lead of machine}}{\text{Max.rise per revolution}}$$

$$= \frac{240}{105 \text{(nearest to 101.6)}}$$

$$= \frac{48}{21}.$$

The angle of inclination (θ)

$$\text{Sin } \theta = \frac{\text{Rise of the cam}}{\text{Lead of Table}}$$

$$= \frac{101.9}{105} = 0.9705$$

$$\theta = 76^\circ 3'.$$

The vertical attachment should be set to

$$90^\circ - 76^\circ 3' = 13^\circ 57'.$$

Drum cam - calculation & advantages

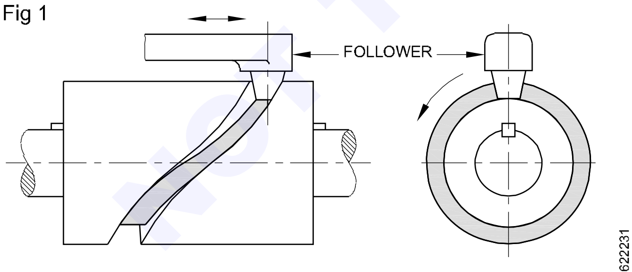
Objectives: At the end of this lesson you shall be able to

- **state the method of producing a drum cam**
- **calculate the gear ratio for milling a drum cam**
- **advantage & disadvantage of cam.**

Drum / Cylindrical Cam

Drum/cylindrical cams are milling on vertical milling machine with end mill cutter. The index head centres should be in same line as that of the machine spindle. The table remains paralled to the machine column and the angle of the helix is obtained by gearing. The gearing connection are same as of the helical milling. The cut is done by the end face of the end mill cutter. At the starting point of the helical flute pilot holes are drilled. (Fig 1)

Fig 1



CYLINDRICAL CAM
TAPERED ROLLER FOLLOWER

Calculation

Gear ratio = Lead of the machine / lead of the lobe =Dr/Dn= on lead screw/On the head plate shaft.

Advantages of cam

- High load carrying capacity.
- Low shock and acceleration - virtual elimination of impact.
- Very short life.
- High reliability
- Quiet operation

Disadvantage of cam

- Its required only rotary input.
- A cam must be manufactured with great precision.
- Cam shapes are complex.
- Good cams are costly.

Cam - Application

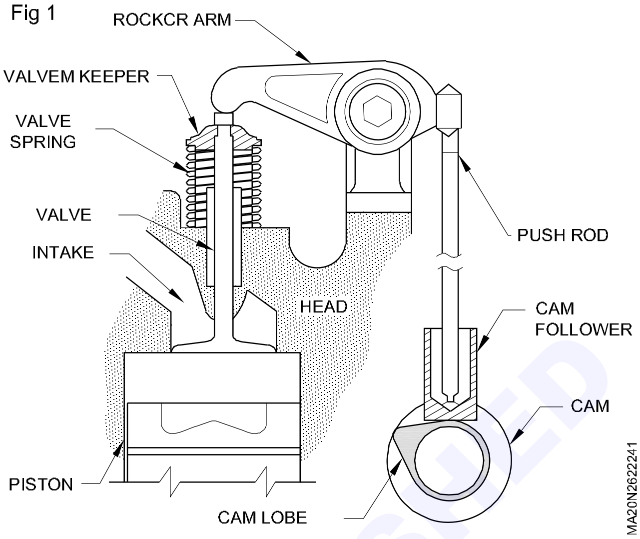
Objective: At the end of this lesson you shall be able to
- **explain the application of cam in modern machine tools.**

Application of cams in modern machine tools

Cam is a machine component that either rotates or moves back and forth (reciprocates) to create a prescribed motion in a contacting element known as a follower. The shape of the contacting surface of the cam is determined by the prescribed motion and the profile of the follower; the latter is usually flat or circular .

Cam follower mechanisms are particularly useful when a simple motion of one part of a machine is to be convertible to a more complicated prescribed motion of another part, one that must be accurately timed with respect to the simple motion and may include periods of rest(dwells). The motion of the camshaft in an automobile engine, for example is a simple rotation that bears a fixed ratio to the crankshaft speed, whereas the valve motion produced by the cams is accurately timed relative to the crankshaft rotation and includes dwells during which the valves remain closed. Cams are essential elements in automatic machine tools, textile machinery, sewing machines, printing mahines, and many others. (Fig 1)

Fig 1



Follower - types

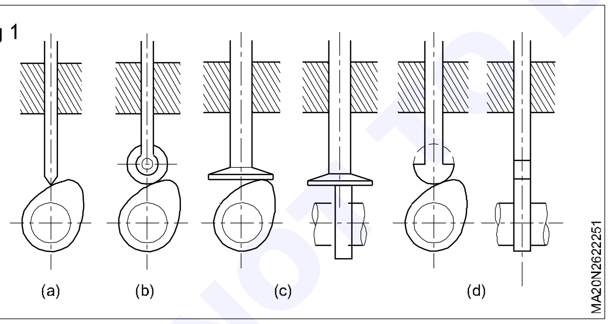
Objectives: At the end of this lesson you shall be able to
- **state the types of followers**
- **state the uses of followers.**

Followers

Followers can be divided according to the shape of that part which is in contact with the cam.

The following diagram shows some of the more common types. (Fig 1)

Fig 1



a **Knife edged**. These are not often used due to the rapid rate wear of the knife edge. This design produces a considerable side thrust between the follower and the guide.

b **Roller follower.** The roller follower has the advantage that the sliding motion between cam and follower is largely replaced by a rolling motion. Note that sliding is not entirely eliminated since the inertia of the roller prevents it from responding instantaneously to the change of angular velocity required by the varying peripheral speed of the cam. This type of follower also produces a considerable side thrust.

c **Flat or mushroom follower.** These have the advantage that the only side thrust is that due to friction between the contact surface of cam and follower. The relative motion is one of sliding but it may be possible to reduce this by off setting the axis of the follower as shown in the diagram. This results in the follower revolving under the influence of the cam.

d **Flat faced follower.** These are really an example of the mushroom follower and are used where space is limited. The most obvious example being automobile engines.

Limits imposed on the shape of the cam working surface by the choice of follower type.

The knife follower does not, theoretically, impose any limit on the space of the cam.

The roller follower demands that any concave portion of the working surface must have a radius at least equal to the radius of the roller.

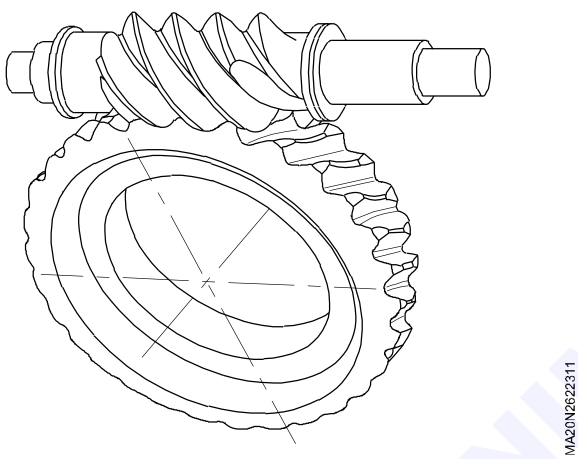
The flat follower requires that everywhere the surface of the cam is convex.

Worm wheel: Applications, elements and calculation

Objectives: At the end of this lesson you shall be able to
- **state the application of worm and worm wheel**
- **name and define the terms of a worm and worm wheel**
- **state the methods of manufacturing worm and worm wheel.**

A worm and worm wheel drive (Fig 1) is used to connect two non-parallel, non-intersecting shafts which are at right angle to each other, mainly used for large speed reduction and torque transmission. In this type of drive the drive is always from the worm and not form the worm wheel.

Fig 1

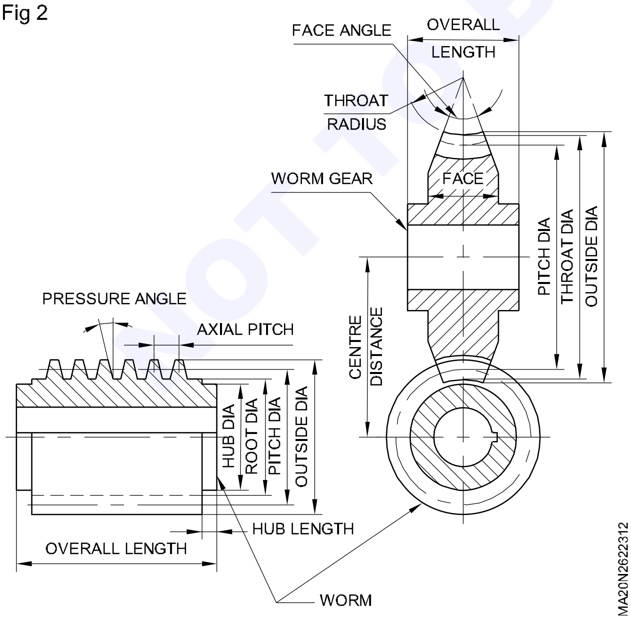


Always the worm and worm wheel have the same hand of helix. Otherwise they will not match.

The speed ratio depends on the number of teeth in the worm wheel and number of starts of worm shaft.

Worm gearing terms (Fig 2)

Fig 2



Throat

The concave surface of a worm gear is called throat.

Worm and worm wheel are used in indexing heads, feed gearboxes of machine tools, automobiles, toys and for speed reductions mechanism, crane, pulley block, etc.

Throat radius

The radius of the concave surface of the throat of a worm gear is known as throat radius.

Throat diameter

The dia. of the worm gear measured at the centre of the throat is known as the throat diameter.

Face angle

It is the angle to which the face of the worm gear is machined.

Normal pitch of worm

It is the distance between the centre of one tooth (thread) and the centre of the adjacent tooth measured perpendicular to the teeth on pitch circle.

Normal pitch

Axial pitch X cosine of helix angle

Axial pitch of worm

It is the distance between the centre of one tooth (thread) and the centre of the adjacent tooth measured along the axis of the worm on pitch line.

Speed ratio

It is the ratio of number of turns of the worm to 1 turn of the worm wheel.

$$\text{Speed ratio} = \frac{Z2}{Z1}$$

where Z2 = Number of teeth on the worm wheel.

Z1 = Number of starts on the worm.

Methods of machining worm

- On a centre lathe
- On a worm milling machine
- On a gear hobbing machine

Methods of machining a wormwheel

- On a milling machine
- On a hobbing machine

Worm wheel : Proportions and calculations

Objectives: At the end of this lesson you shall be able to
- **state the formula for worm-wheel proportions**
- **calculate the elements on worm wheel using std. proportions.**

Worm wheel proportions (suffix 2 indicates worm wheel proportions)

Pitch diameter = d_2 = mZ_2

h_a (addendum) = $m = ha_2$

Throat diameter = $d_2 + 2ha_2$

Outside diameter (Flat top type) = $d2 + 2ha_2 + m$ = Throat dia + m

Root diameter = Pitch dia – $2hf_2$ (dedendum)

Throat radius = 'r' = $\frac{d_1}{2} - m$ (d_1 pd of worm)

Face width of wheel $b_2 = 0.8d_1$

For proper meshing of worm and worm wheel

a. Lead angle of worm = Helix angle of worm wheel

b. Axial pitch of worm = Circular pitch of wheel or pd

c. Centre distance = $\frac{d_1 + d_2}{2}$

Worm wheel calculations

Calculate the dimensions of a worm wheel driven by a worm having 2 modules 60 mm PD, single start, speed ratio 40 : 1 (Flat top type)

Pitch dia = $d_2 = mZ_2$

Therefore, Pitch dia. = 2 x 40 = 80 mm

Throat dia = $(d_2 + 2ha_2)$ = 80 + 2 x 2 = 84 mm

Outside dia = Throat dia. + m = 84 + 2 = 86 mm

Root dia. = Pitch dia. – $2hf_2$ = 80 – 2 x 1.25 x 2 = 75 mm.

Face width of wheel b2 = 0.81d1 = 0.81 x 60 = 48.6 mm.

Throat radius = 'r' = $\frac{d_1}{2} - m$

$= \frac{60}{2} - 2 = 28\text{mm}$

For proper meshing of worm and worm wheel

a Lead angle of worm = Helix angle a of worm wheel

$\tan\alpha = \frac{\text{Lead}}{\pi d_1} = 1°55'$

b Axial pitch of worm = circular pitch of worm wheel = πm = 3.1416 x 2 = 6.2832 say 6.3mm

c Centre distance $\frac{d_1 + d_2}{2} = \frac{60 + 80}{2} = 70$ mm.mm

Worm - calculation

Objectives: At the end of this lesson you shall be able to
- **state the formulae for worm proportions**
- **calculate the dimensions of a worm.**

Worm calculations

Worm proportions (Suffix 1 for worm, 2 for worm wheel)

Pitch diameter = d_1

Outside diameter = $d_1 + 2ha_1$

Float diameter = $d_1 + 2hf_1$

Tooth (thread) height = $ha_1 + hf_1$

where $ha_1 = m$ and $hf_1 = 1.25$ m.

$\tan r = \frac{\text{Lead of worm}}{\pi d_1}$

Lead of worm = πmZ_1

Tooth thickness = $\frac{\pi m}{2}$ Cos r

Worm calculation

Example

Calculate the proportions of a worm elements if it is of 2 module, 60 mm pitch diameter, single start and the speed ratio in 40:1

Solution

Pitch diameter = d_1 = 60mm (given)

Outside diameter = $d_1 + 2ha_1$

= d_1 + 2 m

= 60 + 2 x 2 = 64 mm

Root diameter = $d_1 - 2hf_1$

= d_1 – 2 x 1.25 m

= 60 – 2 x 1.2 5 x 2

= 60 – 5 = 55 mm

Number of starts of the worm = 1 (given)

Number of teeth in worm wheel = Z_2

Z_2 = Speed ratio x Z_1

= 40 x 1 = 40

Length of worm = $b_1 = 2\ m(1+\sqrt{Z_2}\ )$

= $2 \times 2(1+\sqrt{40})$

= 4(1 + 6.32)

= 4 x 7.32 = 29.28 = 30.00 mm

Lead of worm = $\pi\ mZ_1 = \pi \times 2 \times 1$ = 6.3 mm

Lead angle of worm = $\tan r = \dfrac{\text{Leadof worm}}{\pi d_1} = \dfrac{6.3}{\pi \text{ x } 60}$

= 0.0334 = 1°55'

Total depth of thread = $ha_1 + hf_1$

= $ha_1 + hf_1$ = m + 1.25m

= 2 + (1.25 x 2)

+ 2 + 2.5 = 5 mm

Tooth thickness = $\dfrac{\pi m}{2}$ x $\text{Cos}\, r = \dfrac{6.3}{2} \text{x}\, \text{Cos}\, 1°55'$ (0.995)

= 3.14 mm.

Keys and their applications

Objectives : At the end of this lesson you shall be able to
- **state the purpose of key and key way**
- **mention the types of keys**
- **state the specifications of keys**
- **mention the applications of different keys.**

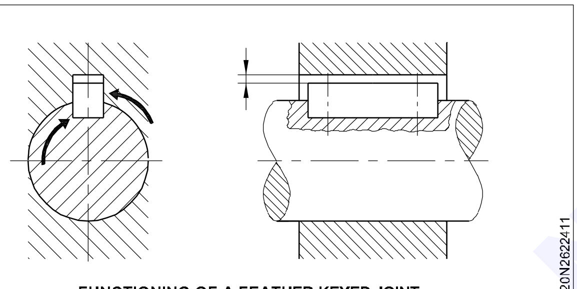
Purpose of Key

A key is an insert which is housed in the keyway to fit together a hub or a pulley to transmit torque. A keyway is provided on the shaft and also on the hub or on a pulley to connect together the conjugate parts by inserting the key in between. The key can be withdrawn at will to disengage the mating components.

Common types

Parallel key or feather key (Fig 1)

Fig 1



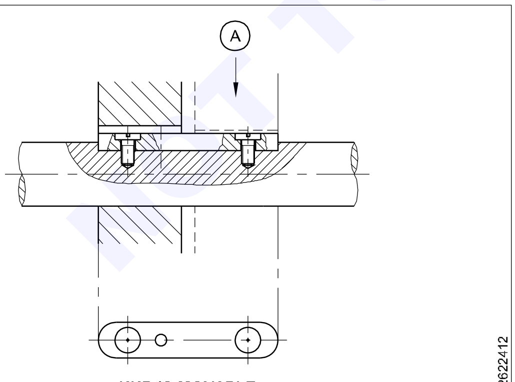
FUNCTIONING OF A FEATHER-KEYED JOINT

Parallel or feather key

This is the most commonly used key, used for transmitting unidirectional torque. A hub or a pulley is engaged to the shaft by a key which prevents relative motion. The functioning of the feather key assembly is shown in Fig 1.

In many cases the key is screwed to the shaft keyway. (Fig 2)

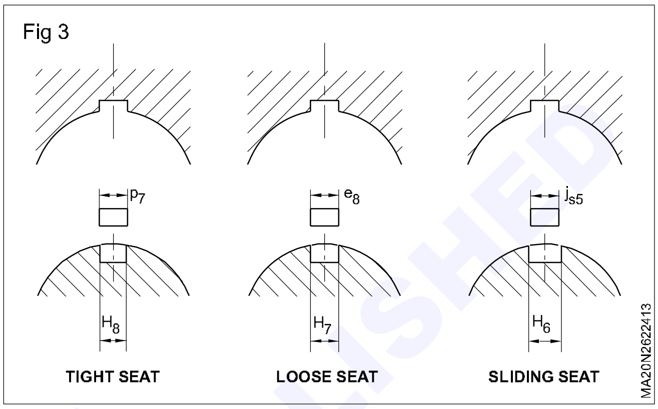
Fig 2



HUB IS MOVABLE
VIEW 'A'

Where axial movement of the hub is required, a clearance fit is provided between the hub and the shaft and the hub and the key. Three types of fits are shown for feather key in Fig 3.

Fig 3



Approximate proportion of parallel or taper keys

If D is the dia. of the shaft, width of the key W = 1/4D+2 mm.

Nominal thickness T = 2/3 w.

Example

Diameter of shaft = 40 mm

$$\text{Width} = \frac{1}{4} \times 40+2 = 12 \text{ mm}$$

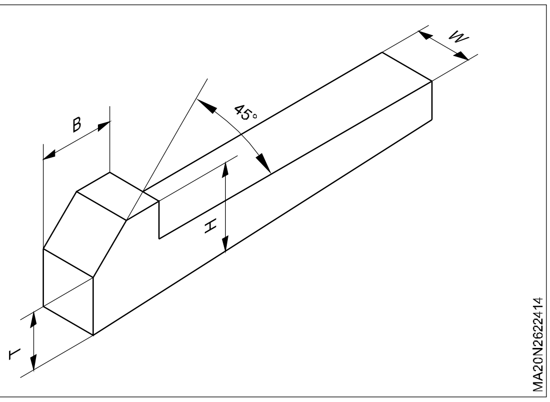
$$\text{Thickness} = \frac{2}{3} \times 12 = 8 \text{ mm}$$

Thickness at the large end is the nominal thickness of the taper key.

Taper is 1 in 100 on the top face only.

Taper and jib-headed key (Fig 4)

Fig 4



The key is having a jib-head with a taper (1 in 100) on the top face. It is driven on to the keyway by hammering on the jib to have a tight fit. The taper rectangular key without a jib-head is also in use. A jib-headed key can be widthdrawn easily and used for transmitting more torque. It is not good for high speed applications.

Approximate proportion of gib-headed key (Fig 4)

$$H = 1.75T$$

$$B = 1.5\ T$$

$$W = \frac{1}{4}\ D+2$$

$$\text{Nominal thickness } T = \frac{2}{3} W$$

$$\text{Angle of chamfer} = 45^{o}$$

Example

Diameter shaft = 46 mm

$$\text{Width(w)} = \frac{1}{4} \times 46+2 = 11.5+2$$

$$= 13.5 \text{ rounded off to 14 mm.}$$

$$\text{Thickness(T)} = \frac{2}{3} \times 13.5 = 9 \text{ mm}$$

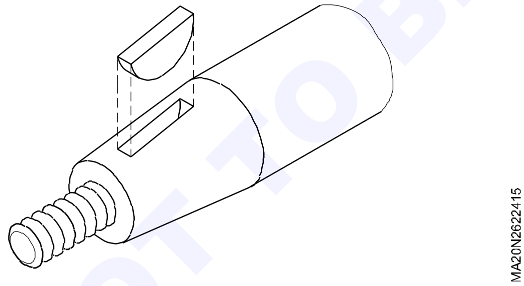
$$H = 1.75 \times 9 = 15.75$$

say 16 mm

$$B = 1.5 \times 9 = 13.5 \text{ mm.}$$

Woodruff key (Fig 5)

Fig 5



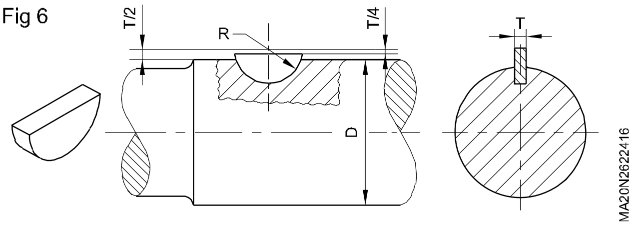
It is a semicircular key used for transmitting light torque. Its key way is milled to the profile of the key on the shaft which tends to weaken the shaft. This type of key positions itself in the keyway to accommodate the hub to have an easy assembly.

Approximate proportion of woodruff key (Fig 6)

$$\text{Radius of the key (R)} = \frac{D}{3}$$

$$\text{Thickness(T)} = \frac{D}{6}$$

Fig 6



Example

For shaft ø 30.

R = 30/3 = 10 mm

T = 30/6 = 5 mm

Saddle key

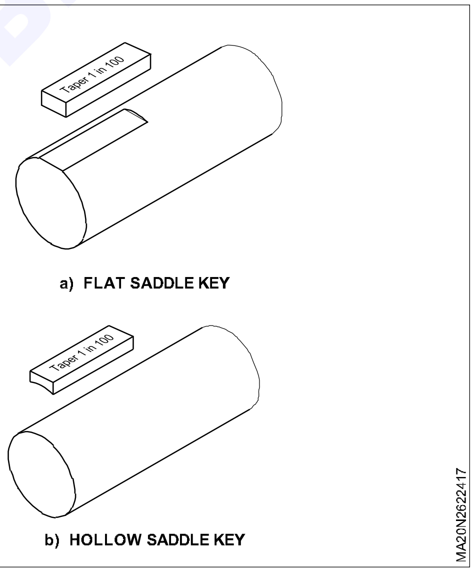
Saddle keys are of two types.

Flat saddle

Hollow saddle

These keys are used for transmitting very low torque. No keyway is made on the shaft. In the case of a flat saddle the shaft is flattened a bit while the hollow saddle's curved portion of the key matches the profile of the shaft as shown in Figs 7 & 8.

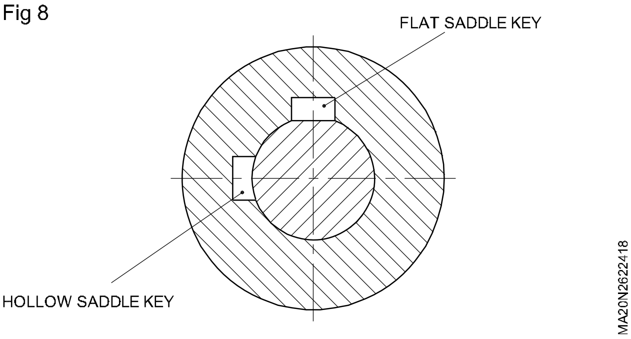
Fig 7



a) **FLAT SADDLE KEY**

b) **HOLLOW SADDLE KEY**

Fig 8



Approximate proportion

If D is the diameter of the shaft,

width of the key (W) = $\frac{1}{4}$ D+2 mm

nominal thickness (T) = $\frac{1}{3}$ W.

Example

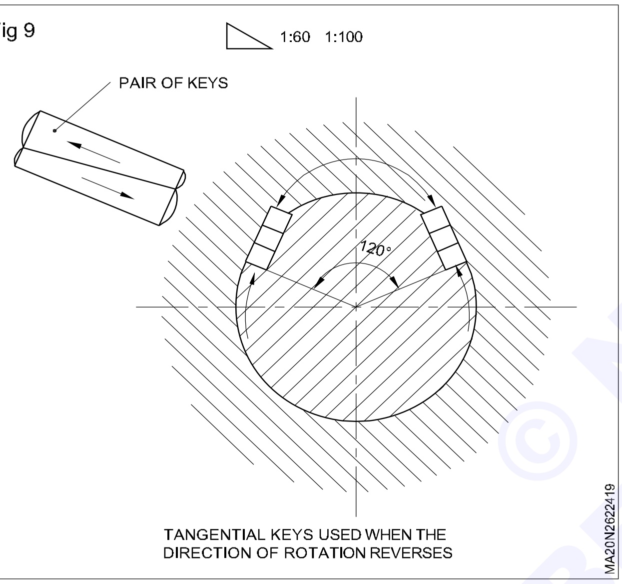
diameter shaft = 24 mm

$$W = \frac{1}{4} \times 24+2 = 8 \text{ mm}$$

$$T = \frac{1}{3} \times 8 = 2.7 \text{ or } 3 \text{ mm.}$$

Tangential key (Fig 9)

Fig 9

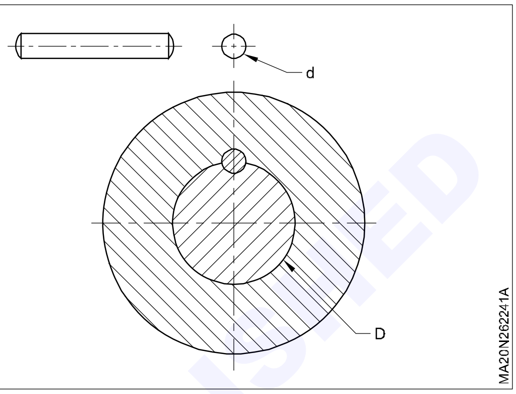


TANGENTIAL KEYS USED WHEN THE DIRECTION OF ROTATION REVERSES

These keys are used when very high torque of impact type is to be transmitted in both directions of rotation. Common applications are found in flywheels, rolling mills etc. A tangential key consists of two taper rectangular wedges, positioned one over the other in opposite directions. Two sets of keys are fixed at 120° angle as shown in Fig 9 and should be such that the broad side is directed along a tangent to the shaft circle while the narrow side sits along the radius of the shaft.

Round key (Fig 10)

Fig 10



It is of cylindrical cross-section and is used in assemblies to secure the mating components where the torque is light. The key is fitted parallel to the shaft into the drilled hole made partly on to the shaft and partly on to the mating part.

Approximate proportion of round key

If dia. of the shaft = D

Dia. of the key (d) = $\frac{1}{6}$ D

Example

Dia. of shaft = 30 mm

Dia of key = $\frac{1}{6} \times 30 = 5$ mm

Gear inspection/testing

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

- **list the gear elements to be measured**
- **brief the methods of checking each elements of the gear.**

Gear inspection begins with the everyday tasks on the shop floor. Following gear elements are checked

- Size of the gear blank
- Pitch circle diameter of the gear
- Gear tooth thickness of the gear
- Run out of the gear
- Profile testing
- Helix inspection
- Pitch inspection
- Single flank inspection

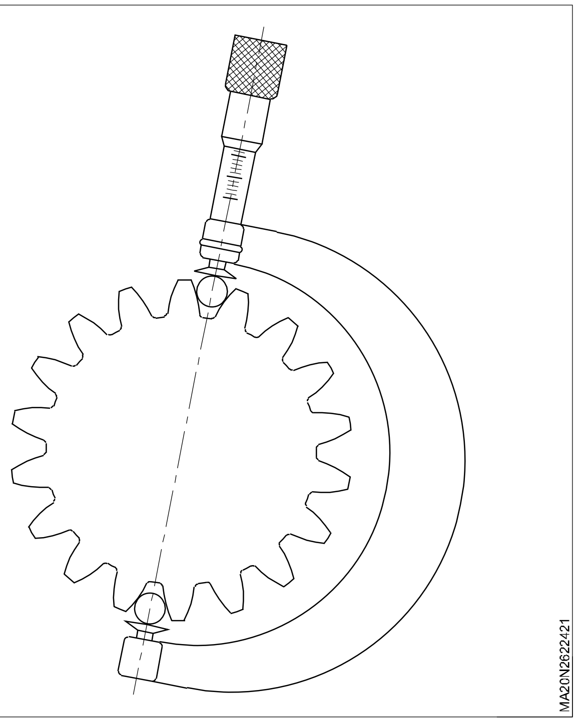
Condition of the production equipment

The following information summarizes the basic elements of gear inspection beginning with the simplest and leading to the more complex.

Inspection: The traditional method of inspecting a gear for correct size is the measurement over pins or balls with a micrometer. Pin measurement provides an accurate and convenient method of determining tooth thickness of a gear of any diameter within the capacity of the available micrometers. For larger diameter gears a span measurement or gear tooth calipers can be used. Composite testing can also provide a measurement of gear tooth size. Size measurement is used to provide the correct backlash when the gear is mounted with its mating gear at operating centre distance.

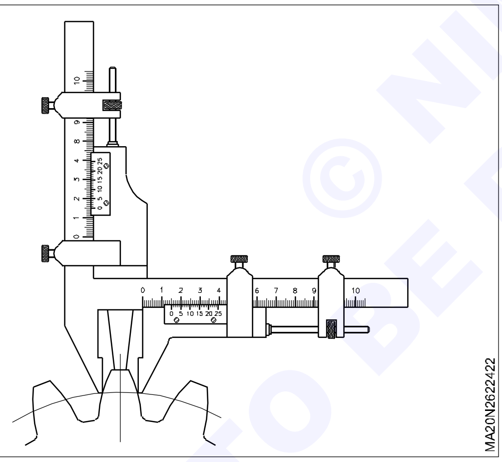
Measurement of pitch circle diameter with micrometer and pins or balls (Fig 1)

Fig 1



Measurement of tooth thickness with gear tooth calliper (Fig 2)

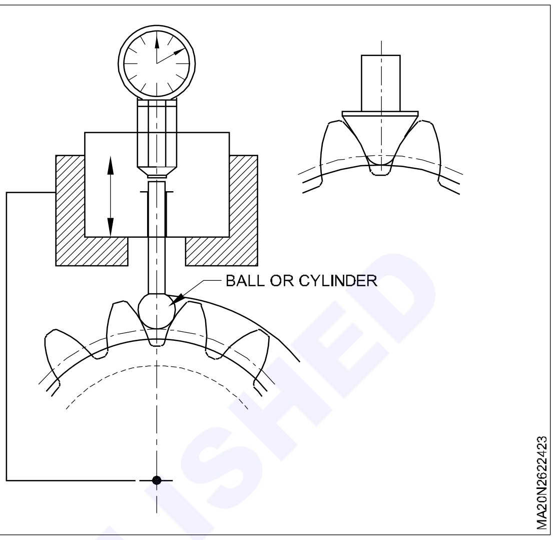
Fig 2



Runout Inspection

Runout is the maximum variation of the distance between a surface of revolution and a datum surface, measured perpendicular to that datum surface. Runout of a gear can be measured with a dial indicator over a pin or ball placed in successive tooth spaces. On modern CNC gear measuring machines this inspection can be provided in a fully automatic cycle. Runout measurement is used to assure correct backlash and minimum variation of rotary motion. (Fig 3)

Fig 3



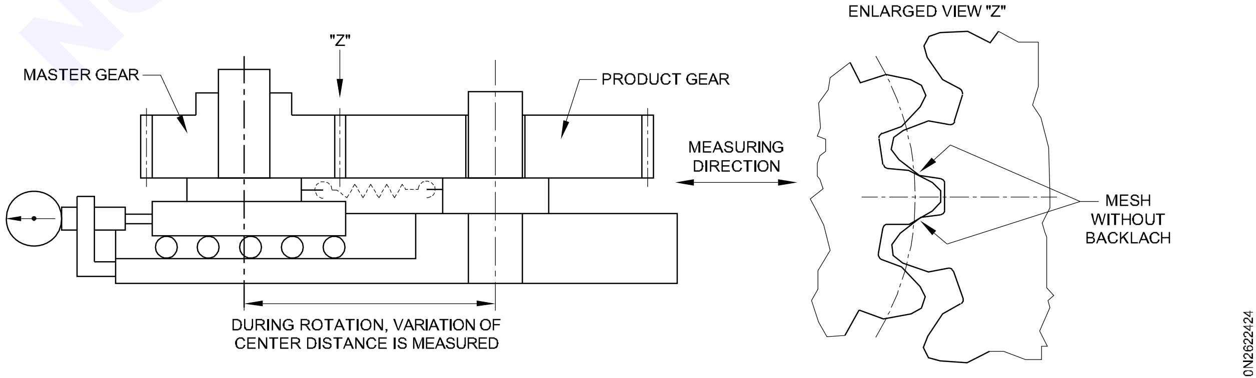
Composite Inspection

The composite test of a gear is a method of inspection in which the work gear is rolled in tight double flank contact with a master gear. AGMA defines this type of inspection as "radial composite deviation." No backlash is provided, as the work gear is spring-loaded against the reference gear on the inspection machine. The composite action test is made on an inspection instrument that will allow variation in the centre distance during rolling. This variation in centre distance will yield a "tooth- to-tooth" and a "total composite" indication that can be read on a simple dial indicator or recorded graphically.

Composite inspection is a useful shop-friendly tool to determine the general quality of a gear including size, runout, tooth-to-tooth rolling action, and to detect nicks. It is not an appropriate method to determine individual tooth flank errors.

Schematic concept of gear rolling device (Fig 4)

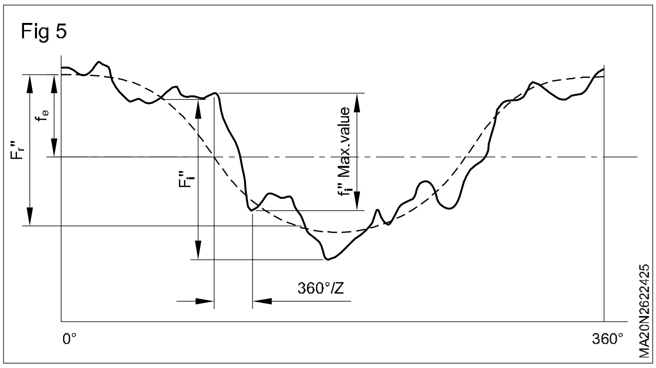
Fig 4



SCHEMATIC CONCEPT OF GEAR ROLLING DEVICE

Graphical record from composite inspection (Fig 5)

Fig 5



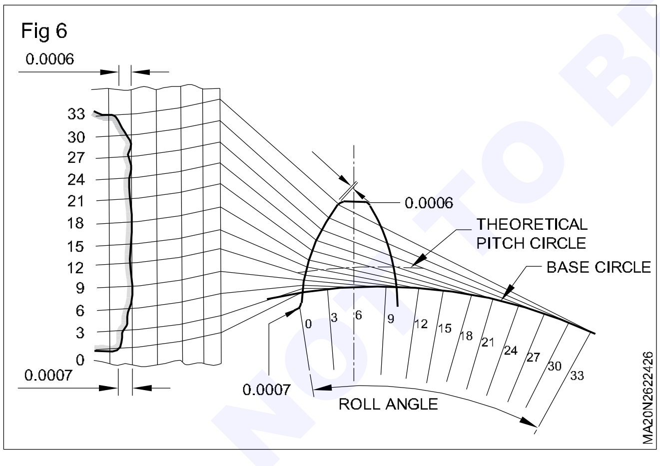
Profile Inspection

Profile is the shape of the gear tooth curve and is measured from the root to the tip of the gear tooth. The functional, or operating, portion of the profile is the area that is in actual contact during tooth mesh. Typically, this area is from just above the root fillet to the tip of the tooth. On most parallel axis gears, the shape of the profile curve is an involute. In practice, an appropriate measuring machine aligns the measuring probe on the test gear in the middle of the gear face. Most gear measuring machines use the generative principle to create a reference profile to compare to the gear's actual profile. The profile is traced and recorded graphically, with a correct "unmodified profile being represented as a straight line on the chart.

Profile inspection with degrees of roll, roll angle. (Fig 6)

Incorrect profile will cause a non-uniform rolling action of the gear, which may cause a large tooth-to-tooth error, uneven loading, and noise problems. In extreme cases, premature gear failure may occur.

Fig 6

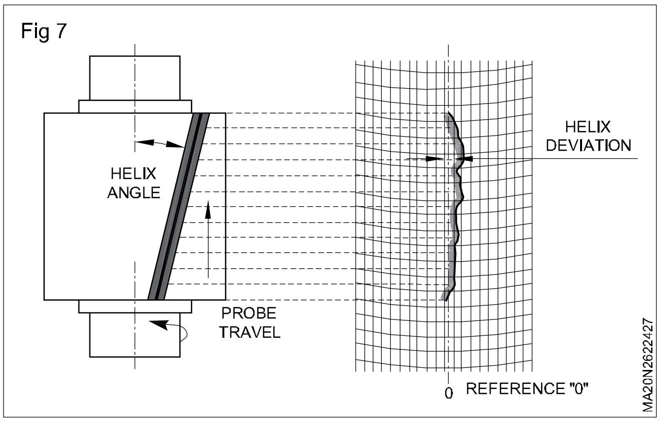


Helix Inspection

AGMA's current inspection handbook defines "helix deviation" (formerly tooth "alignment variation and lead variation) as the difference between the measured "helices to the design helices. In practice an appropriate measuring machine aligns the measuring probe on the test gear at the pitch circle diameter and the "lead" is traced and recorded graphically, with a correct unmodified helix being represented as a straight line on the chart. Helix measurement is used to determine correct face contact between mating gears. Incorrect helix will create uneven loading and noise.

Graphic charting of helix deviation. (Fig 7)

Fig 7



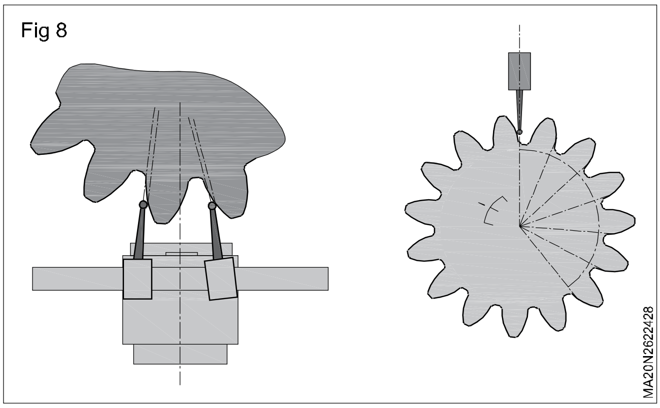
Pitch or Index Inspection "Spacing is the theoretical true position of each tooth around the circumference of the gear. Pitch deviation is the difference between the theoretical position and the actual position of each tooth. These values can be plus or minus. Index variation is the displacement of any tooth from its theoretical position relative to a datum tooth.

Total pitch variation and total index variation are identical values and are generally referred to as "accumulated spacing." Total index variation is the maximum algebraic difference between the extreme values of index variation.

Two distinct methods are available to arrive at tooth "spacing." One utilizes a single -probe measuring device with a precision indexing system. This indexing system can be electronic, as on a CNC measuring machine, with an encoder-controlled rotary axis. It can also use mechanical devices such as index plates, circular divider, or optical scales. The second system utilizes two probes to obtain successive data from adjacent tooth flanks as the gear is rotated. The data obtained from the two-probe system must be mathematically corrected to obtain spacing values. It is recognized today that the single probe system is the most accurate and the preferred system.

Pitch measurement using a pitch comparator and angular indexing. (Fig 8)

Fig 8



Index measurements are used to determine the correct spacing of gear teeth. Spacing error is the principle source of gear noise due to total pitch variation or accumulated spacing. Although the main component of total pitch variation is from part runout, it may not be possible in all cases to detect this from a simple runout or composite inspection check.

Single Flank Inspection

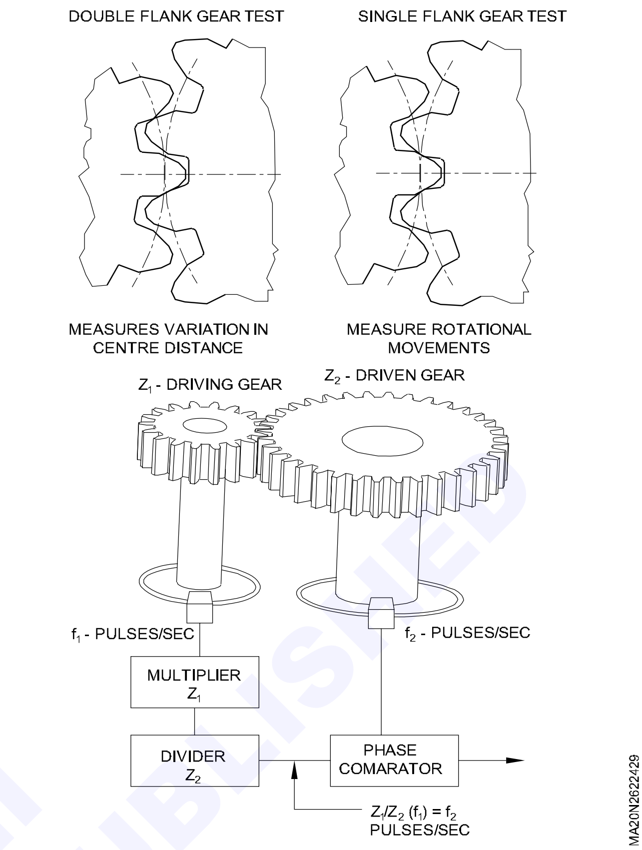
Single flank inspection appears to be identical with the composite, or double flank, inspection technique. In fact, it is quite different due to the fact that the test gear is rolled at its design centre distance and backlash with a master or reference gear. This closely simulates the operation of the actual gear.

Difference between double and single flank inspection (Fig 9)

A single flank inspection instrument utilizes encoders on the two axes of rotation either as a fixed or portable unit. The rotational data from each encoder is then processed electronically, and the resulting phases are compared with each other to yield a phase differential. This will indicate errors of rotational motion from the ideal constant angular velocity of perfectly conjugate gears. The results of this phase difference are graphically recorded as an analog waveform, similar to a composite inspection chart.

The most important aspect of single flank inspection is its ability to measure profile conjugacy. The data is also related to profile variation, pitch variation, runout, and accumulated pitch variation. Single flank testing does not eliminate the need for analytical inspection of helix deviation, and it is not as effectively applied to gear sets with increased contact ratios such as helical gears.

Fig 9



Gear defects

Objectives : At the end of this lesson you shall be able to
- **list the defects in gears**
- **brief the causes of each defects in gears.**

Bending failure

Gear tooth behaves like a cantilever beam subjected to repetitive bending stress. The tooth may crack due to repetitive bending stress

To avoid such failure, the module and face width of the gear is adjusted so that the beam strength is greater than the dynamic load.

Pitting

It is a surface fatigue failure due to repetitive contact stresses. Pitting starts when total load acting on the gear tooth exceeds the wear strength of the gear.

To avoid the pitting, the dynamic load between the gear tooth should be less than the wear strength of the gear tooth.

The initial or corrective pitting is a localized phenomenon, characterized by small pits at high spots. Such high spots are progressively worn out and the load is redistributed. Initial pitting is caused by the errors in tooth profile, surface irregularities, and misalignment.

The remedies against initial pitting are precise machining of gears, adjusting the correct alignment of gears so that the load is uniformly distributed across the full face width, and reducing the dynamic loads.

This is a major cause of gear failure accounting for nearly 60% of the gear failures. Pitting is the formation of craters on the gear tooth surface. These craters are formed due to the high amount of compressive contact stresses in the gear surface occurring during transmission of the torque or in simple terms due to compressive fatigue on the gear tooth surface.

Spalling

Although it is similar to severe pitting, the pits tend to be shallow and larger in diameter. Additionally, the area that is showing spalling does not tend to be uniform. It is a common problem when high contact stress exists.

Scoring

It is a lubrication failure. Inadequate lubrication along with high tooth load & poor surface finish results in the breakdown of the oil film and causes the metal to metal contact.

This type of failure can be avoided by properly designing the parameters such as speed, pressure, and proper flow of the lubricant so that the temperature at the rubbing faces is within the permissible limits.

Excessive surface pressure, high surface speed, and inadequate supply of lubricants result in the breakdown of the oil film. This results in excessive frictional heat and overheating of the meshing teeth.

Scoring is a stick-slip phenomenon, in which alternate welding and shearing takes place rapidly at the high spots. Here, the rate of wear is faster.

Scoring can be avoided by selecting the parameters, such as surface speed, surface pressure, and the flow of lubricant in such a way that the resulting temperature at the contacting surfaces is within permissible limits.

The bulk temperature of the lubricant can be reduced by providing fins on the outside surface of the gearbox and a fan for forced circulation of air over the fins.

Scoring is due to the combination of two distinct activities: First, lubrication failure in the contact region and second, the establishment of metal to metal contact. Later on, welding and tearing action resulting from metallic contact removes the metal rapidly, and continuously so far the load, speed, and oil temperature remain at the same level.

The scoring is classified into initial, moderate, and destructive.

Initial scoring

Initial scoring occurs at the high spots left by previous machining. Lubrication failure at these spots leads to initial scoring or scuffing. Once these high spots are removed, the stress comes down as the load is distributed over a larger area. The scoring will then stop if the load, speed, and temperature of oil remain unchanged or reduced. Initial scoring is non-progressive and has corrective action associated with it.

Moderate scoring

After initial scoring if the load, speed, or oil temperature increases, the scoring will Spread over to a larger area. The Scoring progresses at a tolerable rate. This is called moderate scoring.

Destructive scoring

After the initial scoring, if the load, speed, or oil temperature increases appreciably, then severe scoring sets in with heavy metal torn regions spreading quickly throughout. Scoring is normally predominant over the pitch line region since elastohydrodynamic lubrication is the least in that region. In dry running, surfaces may seize.

Corrosive wear.

It is due to chemical action by the improper lubricant or sometimes it may be due to the surrounding atmosphere which may be corrosive nature. To avoid this type of “wear, proper anti-corrosive additives should be used.

Abrasive wear

It is a surface damage caused by particles trapped in between the matting teeth “surfaces.

This type of failure can be avoided by providing filters for the lubricating oil or by “using high viscosity lubricant oil which enables the formation of thicker oil film and hence permits easy passage of such particles without damaging the gear surface.

Foreign particles in the lubricant, such as dirt, rust, weld spatter, or metallic debris can scratch or Brinell the tooth surface. Remedies against this type of wear are the provision of oil filters, increasing surface hardness, and the use of high viscosity oils. A thick lubricating film developed by these oils allows fine particles to pass without scratching.

Frosting

This issue usually shows up in the reddendum area of the driving gear. The wear pattern gives a frosted appearance, which are many micro pits on the surface. Frosting is a “common issue when the heat breaks down the lubrication film.

Breakage

It is possible for the entire tooth or a piece of the tooth to break away. It often leaves “evidence of the focal point of the fatigue that led to the break, which results from any “number of issues, including high stress or excessive tooth loads.

When an issue with the gears takes place, it is necessary to determine if a rebuild, “upgrade or replacement is the best choice available. We can help you to make the best decision according to the problems that exist and care for the work for you to ensure that your equipment is up and operating again.