CNC MACHINING TECHNICIAN

NSQF LEVEL - 4

1st Year

TRADE THEORY

SECTOR: CAPITAL GOODS & MANUFACTURING

(As per revised syllabus 2023 - 1200 Hrs)

Directorate General of Training

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

NATIONAL INSTRUCTIONAL MEDIA INSTITUTE, CHENNAI

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

Sector : CG & M

Duration : 2 Years

Trade : CNC Machining Technician - 1st Year - Trade Theory - NSQF Level - 4 (Revised 2023)

Developed & Published 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 : February 2025 Copies:

Rs.300/-

FOREWORD

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

The National Instructional Media Institute (NIMI), Chennai, has now come up with instructional material to suit the revised curriculum for **CNC Machining Technician 1st Year Trade Theory** in **Capital Goods & Manufacturing Sector** under **Annual Pattern.** The NSQF Level - 4 (Revised 2023) 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 2023) trainees will also get the opportunities to promote life long learning and skill development. I have no doubt that with NSQF Level - 4 (Revised 2023) the trainers and trainees of ITIs, and all stakeholders will derive maximum benefits from these Instructional Media Packages IMPs and that NIMI's effort will go a long way in improving the quality of Vocational training in the country.

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

Jai Hind

Atul Kumar Tiwari, I.A.S.,
Secretary
Director General (Training)
Ministry of Skill Development & Entrepreneurship,
Government of India.

February 2025
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 (NSQF) under the Craftsman and Apprenticeship Training Schemes.

The instructional materials are created keeping in mind, the main objective of Vocational Training under NCVET/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 **CNC Machining Technician - NSQF Level - 4 (Revised 2023)** under **Capital Goods and Manufacturing** Sector for ITIs.

MEDIA DEVELOPMENT COMMITTEE MEMBERS

Shri. Manu Kumar H.D	–	TO, NSTI , Chennai.
Shri. Haribabu D	–	Trade Expert , Chennai.
Shri. Srinivasa Naika	–	JTO, Govt. ITI, Mysuru.
Shri. R.S. Mallanavar	–	JTO, Govt. ITI, Dhaawad.
Shri. K.O. Gurusamy	–	JTO, Govt. ITI, Davangere .
Shri. Manjunath	–	JTO, Govt. ITI , KGF.
Shri. Jaspreet Singh	–	SME, TATA Technologies.
Shri. Raman Kumar	–	SME, TATA Technologies.
Shri. Zaid Farooq	–	SME, TATA Technologies.
Shri. Khalidsaifulla Lad	–	SME, TATA Technologies.
Shri. Jagjit Singh	–	SME, TATA Technologies.
Shri. A. Vijayaraghavan	–	ADT, Rtd., MDC Member, NIMI.
Shri. N. Manohar	–	TO (Rtd), DET, Tamil Nadu. MDC Member, NIMI.
Shri. M. Sampath	–	TO (Rtd), MDC Member, NIMI.
Shri. G. Mani	–	DDP JWM, Rtd., MDC Member, NIMI.

NIMI CO-ORDINATORS

Shri. T. V. Rajasekar	-	Joint Director, NIMI, Chennai - 32
Shri. Nirmalya Nath	-	Deputy Director, NIMI, Chennai - 32.
Shri. V. Gopalakrishnan	-	Manager, NIMI, Chennai - 32.

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

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

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

INTRODUCTION

TRADE 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 1st Year Course of **CNC Machining Technician under Capital Goods and Manufacturing Sector.** Trade supplemented and supported by instructions/ informations to assist in performing the exercises. These exercises are designed to ensure that all the skills in compliance with NSQF LEVEL - 4 (Revised 2023) syllabus are covered. The manual is divided into Ten modules.

Module 1	-	**Safety and House Keeping**
Module 2	-	**Basic Turning and Milling**
Module 3	-	**Industrial Drawing and Specification**
Module 4	-	**Product Inspection**
Module 5	-	**CNC Tooling's and Work Holding Devices**
Module 6	-	**CNC Turning - Basic**
Module 7	-	**CAM - Basic**
Module 8	-	**CNC Turning - Advanced**
Module 9	-	**CAM - Advanced**
Module 10	-	**CNC Maintenance**

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 **CNC Machining Technician 1st Year NSQF Level - 4 (Revised 2023)** 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

Lesson No.	Title of the Lesson	Learning Outcome	Page No.
	Module 1: Safety and House Keeping		
1.1.01	Guidance to trainees to familiar on industrial training institute		1
1.1.02	Approach on soft skills		3
1.1.03	Safety and general precautions in industry/shop floor		4
1.1.04 - 07	First-aid	1	6
1.1.08	Response to emergencies		13
1.1.09	Importance of housekeeping		14
1.1.10	Occupational safety and health		16
1.1.11	Basic understanding on hot work, confined space work and material handling equipment		18
	Module 2: Basic Turning and Milling		
1.2.12	Fundamentals of work piece rotation		21
1.2.13-15	Three jaw / four jaw chucks, face plate - clamping of work piece		22
1.2.16-19	Tool posts		26
1.2.20	Center height adjustment of tool		29
1.2.21 & 22	Inspection quality of product by using measuring instruments like vernier, micrometer etc.	2 & 3	30
1.2.23	Outside micrometers, parts, principle		34
1.2.24	Types of milling machines		39
1.2.25-27	Different types of milling cutter and their uses - cutter nomenclature		41
1.2.28	Milling cutter holding devices		51
1.2.29	Cutting speed, feed and machining time calculation		54
1.2.30	Identification of defective milling cutters		57
	Module 3: Industrial Drawing and Specification		
1.3.31	Introduction to product design and development		58
1.3.32 & 33	Customer's requirements, specification and importance of customer relationships		60
1.3.34	Introduction to engineering drawing		63
1.3.35	Fundamentals of limits, fits & tolerances & importance of inter-changeability and ISO standards		67
1.3.36	Industrial engineering special characteristic symbol	4, 5 & 6	72
1.3.37	Geometrical dimensions & tolerances		73
1.3.38	Machining procedure from raw material to finished product		79
1.3.39	Concept of process flow of machining operation		83
1.3.40	Concept work-piece holding		86
1.3.41	Multi-stage engineering drawings		88
	Module 4: Product Inspection		
1.4.42 & 43	Surface finish and its importance		90

Lesson No.	Title of the Lesson	Learning Outcome	Page No.
1.4.44	Surface finish calculation		95
1.4.45	Post process manufacturing operation to improve surface finish quality	7 & 8	97
1.4.46 - 51	Introduction to quality of a product		100
	Module 5: CNC Toolings and Work Holding Devices		
1.5.52	CNC toolings, holders & its types		105
1.5.53	Nomenclature of cutting tools & selection of cutting tools, its machining process parameter	9	109
1.5.54	Cutting fluid & coolant used for machining		119
1.5.55	Workpiece holding devices		121
1.5.56	Jigs and Fixtures		125
	Module 6: CNC Turning - Basic		
1.6.57 - 59	Programming sequence, ISO 'G' codes and 'M' codes		127
1.6.60 & 61	CNC lathe types		135
1.6.62 - 64	Machine operation modes - Jog - MPG - edit memory - Fanuc system		137
1.6.65 - 67	Axes convention of CNC machines	10 - 19	141
1.6.68 & 69	Concepts of coordinate geometry		143
1.6.70 - 72	Stock removal in CNC turning external and internal		147
1.6.73 - 74	Canned cycles		150
1.6.75 - 80	Tool nose radius compensation (TNRS)		155
	Module 7: CAM - Basic		
1.7.81 - 85	Introduction to the computer aided manufacturing software concept of toolbar & ribbon setting attribute & user interface orientation	20 & 21	158
1.7.86 - 89	3D model - Import and simulation in CAM software		168
1.7.90 & 91	NC program for machining		179
	Module 8: CNC Turning - Advanced		
1.8.92	Manufacturing processes		185
1.8.93	Introduction to CNC Turning, VMC and NC		187
1.8.94	CNC machine controllers		189
1.8.95	Tooling systems for CNC turning centres	22 - 25	194
1.8.96	Workholding for CNC turning		198
1.8.97	Proving the CNC program		202
1.8.98 & 99	Machine control and its functions		204
1.8.100 - 102	Concept of co-ordinate geometry and polar co-ordinates		208
1.8.103 - 105	Tool wear and necessity of wear offset		211
1.8.106 & 107	Concept of Sub programming in CNC Machining		214

Lesson No.	Title of the Lesson	Learning Outcome	Page No.
	Module 9: CAM - Advanced		
1.9.108	Introduction to CNC turning centre program using CAM		216
1.9.109	Selection of tools depending on material to be cut		218
1.9.110	Program creation tool and technique		221
1.9.111	Generation of complex machining part program with the help of CAM software	26 & 27	230
1.9.112	Tool path optimization and cycle time calculation		236
1.9.113	Machine off set and cutter tool nose radius compensation		238
1.9.114	Concept of interpolation and canned cycles		240
1.9.115	Importance of program exchange between system and machine		242
1.9.116	Concept of importing & exporting of CNC program		244
	Module 10: CNC Maintenance		
1.10.117	Basic maintenance of turning machine	28	246

LEARNING / ASSESSABLE OUTCOME

On completion of this book you shall be able to

Sl. No.	Learning Outcome	Exercise No.
1	**Identify & comply with the safe working practices, environmental regulation and housekeeping. (NOS: CSC/N9511)**	**1.1.01 - 1.1.11**
2	**Perform Turning operations on simple parts. (NOS: CSC/N9513)**	**1.2.12 & 1.2.23**
3	**Perform milling operations on simple components. (NOS: CSC/N9407)**	**1.2.24 - 1.3.30**
4	**Identify customer needs & Product specification. (NOS: CSC/N9550)**	**1.3.31 & 1.3.33**
5	**Draw and Interpret industrial engineering drawing & its requirements. (NOS: CSC/N9551)**	**1.3.34 & 1.3.37**
6	**Construct the detail drawing of Machining stages. (NOS: CSC/N9556)**	**1.3.38 - 1.3.41**
7	**Check the quality of surface finish adhering to Surface roughness factor. (NOS: CSC/N9563)**	**1.4.42-1.4.45**
8	**Identify the measuring instruments and inspect the quality of final product. (NOS: CSC/N9563)**	**1.4.46 - 1.4.51**
9	**Identify the cutting tools & apply work-piece holding techniques. (NOS: CSC/N9563)**	**1.5.52 - 1.5.56**
10	**Apply 'M' code & 'G' Code used in CNC Lathe & VMC machines. (NOS: CSC/N9552)**	**1.6.57 - 1.6.59**

LEARNING / ASSESSABLE OUTCOME

On completion of this book you shall be able to

Sl.No.	Learning Outcome	Exercise No.
11	Identify CNC machines over travel limits & emergency stop, machine parts, various modes in CNC machines (Jog, MDI, Edit, Auto, Single Block, MPG). (NOS: CSC/N9564)	1.6.60 - 1.6.64
12	Create and edit the Linear interpolation, Rapid traverse program of CNC turning center. (NOS: CSC/N9553)	1.6.65 - 1.6.67
13	Create Absolute & Incremental program in CNC turning center. (NOS: CSC/N9553)	1.6.68 & 1.6.69
14	Create and edit the Circular interpolation CW &CCW programs in turning center. (NOS: CSC/N9553)	1.6.70 & 1.6.71
15	Create, simulate, execute an external profile turning operation using stock removal cycles. (NOS: CSC/N9561)	1.6.72
16	Create, simulate, execute an external Grooving, parting -off & threading operation using Canned cycles (NOS: CSC/N9561)	1.6.73 & 1.6.74
17	Demonstrate Tool nose radius compensation in CNC turning program. (NOS: CSC/N9557)	1.6.75 - 1.6.77
18	Perform Computer aided machining & Wire-frame Geometry Creation, Surface and Solid Modeling, Dimension, Importing and Exporting of files. (NOS: CSC/N9562)	1.6.78
19	Create, simulate, execute an internal profile turning operation using stock removal cycles. (NOS: CSC/N9561)	1.6.79 & 1.6.80
20	Create, simulate, execute an internal Grooving, parting -off & threading operation using Canned cycles (NOS: CSC/N9561)	1.7.81 - 1.7.85
21	Verify Toolpath Generation & Programming by using Computer Aided Manufacturing Software. (NOS: CSC/N9558)	1.7.86 - 1.7.91
22	Explain the need of CNC turning, VMC machines & the machining component. (NOS: CSC/N9557)	1.8.92 - 1.8.94
23	Explain the need of advanced CNC Turning Centre. (NOS: CSC/N9554)	1.8.95 &1.8.96
24	Perform operation on advanced CNC Turning Centre. (NOS: CSC/N9554)	1.8.97 & 1.8.105
25	Run the CNC program or subprogram. (NOS: CSC/N9555)	1.8.106 & 1.8.107
26	Perform Programming of CNC Turning Centre using CAM. (NOS: CSC/N9555)	1.9.108 - 1.9.113
27	Perform Importing & Exporting of CNC turning Program. (NOS: CSC/N9555)	1.9.114 - 1.9.116
28	Perform routine maintenance & basic troubleshooting of CNC turning center. (NOS: CSC/N9555)	1.10.117

Guidance to trainees to familiar on industrial training institute

Objectives: At the end of this lesson you shall be able to
- **state what is ITI and brief objective**
- **describe organisational chart**
- **list out infrastructure of ITI.**

The objective of industrial training institute (ITI) is to train people for suitable industrial employment and as well as for self-employment.

The institute imparts training in engineering and non-engineering courses approved by the Government of India in consultation with the national council for vocational, educational and training New Delhi.

The industrial training institute & centers (ITI & ITC) comes under National Skill Training scheme to provide vocational training in various trades functioning under Directorate General of Training (DGT), Ministry of Skill Development and Entrepreneurship, Government of India.

ITI and ITC are one and the same

ITI's are governed by the State/Union Government

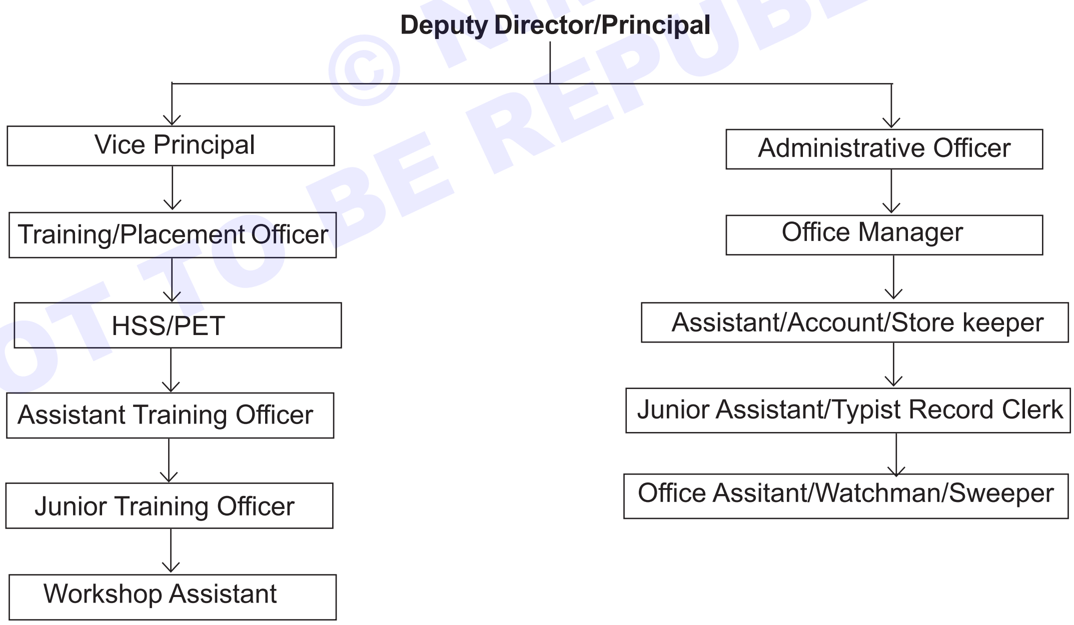
ITCs are self-financing institution.

Trade test are common for ITI and ITC. National Trade Certificate issued by National Council for Vocational, Educational Training. (NCVET)

The organisational structure of the training institute may vary state to state, but order of flow is from senior officials to ground level staff.

For trainees the overall incharge is Trade master.

Organisational chart of Industrial Training Institute



Stores

The complete inward and outward stores and controlled by store keeper. Store are house with all tools, equipment and consumable. The instructor will intend tools, equipments, consumable and raw materials as per syllabus of the trades trained in the institute.

Infrastructure available in ITI's

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

The following facilities are available in ITI's

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

Admission process

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

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

Craftsman training scheme exam system

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

Approach on soft skills

Objectives: At the end of this lesson you shall be to
- **state the concept of soft skill**
- **list the important common soft skills**
- **brief the employability aspect of training**
- **brief the further learning scope.**

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

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

Common soft skills

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

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

Various job available in different industries in India and Abroad.

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

Self-employment

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

Further learning scope

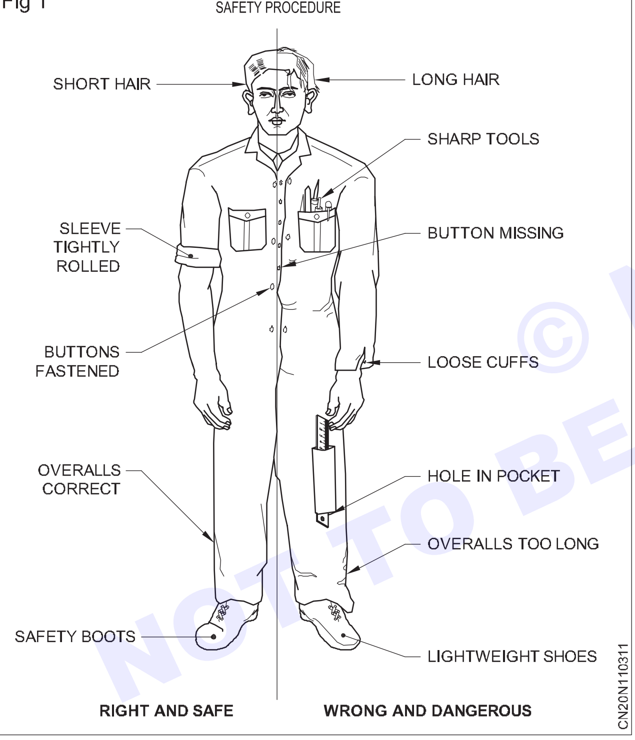
- Apprentice training in designated trade.
- Craft Instructor certificate course.
- Diploma in relevant Engineering

Safety and general precautions in industry/shop floor

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

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

Fig 1



Safety in a workshop can be broadly classified into 3 categories.

- General safety
- Personal safety
- Machine safety

General safety

Keep the floor and gangways clean and clear.

Move with care in the workshop, do not run.

Don't leave the machine which is in motion.

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

Don't walk under suspended loads.

Don't crack practical jokes while on work.

Use the appropriate tools for the job.

Keep the tools at their proper place.

Wipe out split oil immediately.

Replace worn out or damaged tools immediately.

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

Ensure adequate light in the workshop.

Clean the machine only when it is not in motion.

Sweep away the metal cuttings.

Know everything about the machine before you start it.

Personal safety

Wear a one piece overall or boiler suit.

Keep the overall buttons fastened.

Don't use ties and scarves.

Roll up the sleeves tightly above the elbow.

Wear safety shoes or boots

Cut the hair short.

Don't wear a ring, watch or chain.

Never lean on the machine.

Don't clean hands in the coolant fluid.

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

Don't use cracked or chipped tools.

Don't start the machine until

– the workpiece is securely mounted
– the feed of machinery is in the neutral
– the work area is clear & neat.

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

Never touch the electrical equipment with wet hands.

Don't use any faulty electrical equipment.

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

Concentrate on your work. Have a calm attitude.

Do things in a methodological way.

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

Don't distract the attention of others.

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

Machine safety

Switch off the machine immediately, if something goes wrong.

Keep the machine clean.

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

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

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

Stop the machine before changing the speed.

Disengage the automatic feeds before switching off.

Check the oil level before starting the machine.

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

Take measurements only after stopping the machine.

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

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

First-aid

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

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

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

Important guideline for first aiders

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

Remember A-B-Cs

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

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

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

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

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

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

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

Fig 1

Look, listen and feel for signs of breathing

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

If the victim is not breathing, see the section below

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

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

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

- **Stop bleeding:** Control of bleeding is one of the most important things to save a trauma victim. Use direct pressure on a wound before trying any other method of managing bleeding.

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

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

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

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

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

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

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

First aid

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

Do not

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

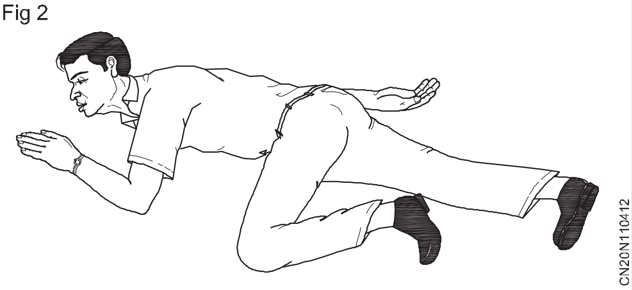
Loss of consciousness may threaten life if the person is on his back and the tongue has dropped to the back of the throat, blocking the airway. Make certain that the person is breathing before looking for the cause of unconsciousness. If the injuries permit, place the casualty in the recovery position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

How to diagnose an unconscious injured person

- **Consider alcohol** look for signs of drinking, like empty bottles or the smell of alcohol.

- **Consider epilepsy** are there signs of a violent seizure, such as saliva around the mouth or a generally disheveled scene?
- **Think insulin** might the person be suffering from insulin shock (see 'How to diagnose and treat insulin shock")?

Fig 2



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

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

Operation of electrical mains

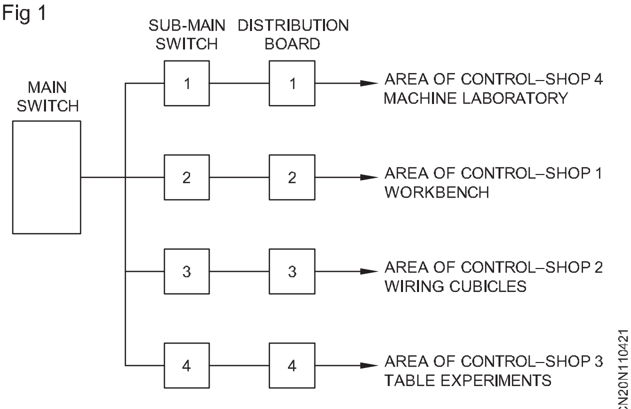
Objectives: At the end of this lesson you shall be able to
- **explain the term 'emergency'**
- **explain the need to switch off the circuit during emergency**
- **explain the method of locating the area sub-main and switches in the shop floor**
- **explain the position of handle with respect to ON &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

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

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

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

Fig 1



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

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

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

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

Fig 2

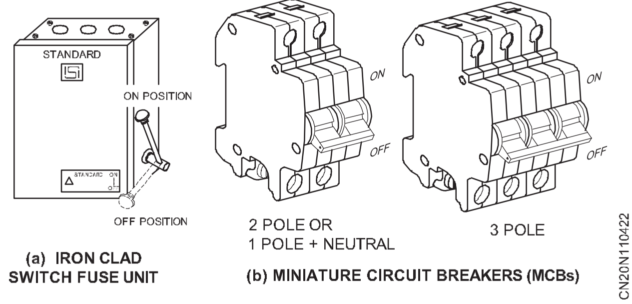
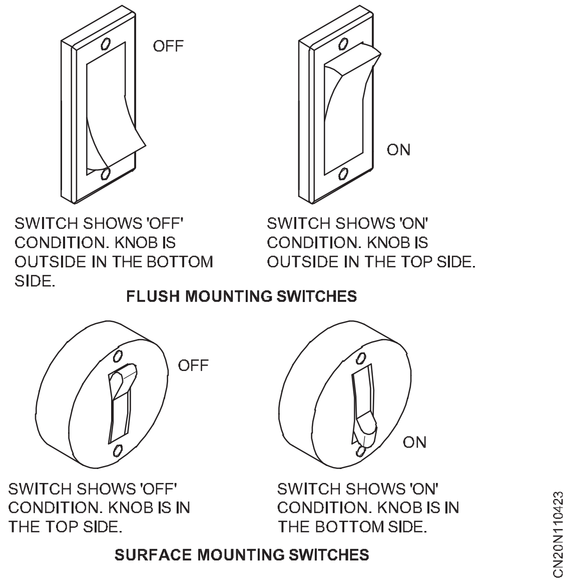


Fig 3



Electrical safety

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

Safety rules

Necessity of safety rules

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

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

Safety rules

- Only qualified persons should do electrical work
- Keep the workshop floor clean, and tools in good condition.
- Do not work on live circuits, if unavoidable, use rubber gloves rubber mats, etc.
- Use wooden or PVC insulated handle screwdrivers when working on electrical circuits.
- Do not touch bare conductors.
- When soldering, place the hot soldering irons in their stand. Never lay switched 'ON' or heated soldering iron on a bench or table as it may cause a fire to break out.
- Use only correct capacity fuses in the circuit. If the capacity is less it will blow out when the load is connected. If the capacity is large, it gives no protection and allows excess current to flow and endangers men and machines, resulting in loss of money.
- Replace or remove fuses only after switching off the circuit switches.
- Use extension cords with lamp guards to protect lamps against breakage and to avoid combustible material coming in contact with hot bulbs.
- Use accessories like sockets, plugs and switches and appliances only when they are in good condition and be sure they have the mark of BIS (ISI). (Necessity using BIS (ISI) marked accessories is explained under standardisation.
- Never extend electrical circuits by using temporary wiring.
- Stand on a wooden stool, or an insulated ladder while repairing live electrical circuits/appliances or replacing fused bulbs. In all the cases, it is always good to open the main switch and make the circuit dead.
- Stand on rubber mats while working/ operating switch panels, control gears etc.
- Position the ladder, on firm ground.
- While using a ladder, ask the helper to hold the ladder against any possible slipping.
- Always use safety belts while working on poles or high rise points.

- Never place your hands on any moving part of rotating machine and never work around moving shafts or pulleys of motor or generator with loose shirt sleeves or dangling neck ties.
- Only after identifying the procedure of operation, operate any machine or apparatus.
- Run cables or cords through wooden partitions or floor after inserting insulating porcelain tubes.
- Connections in the electrical apparatus should be tight. Loosely connected cables will heat up and end in fire hazards.
- Use always earth connection for all electrical appliances along with 3-pin sockets and plugs.
- While working on dead circuits remove the fuse grips; keep them under safe custody and also display 'Men on line' board on the switchboard.
- Do not meddle with inter locks of machines/switch gears
- Do not connect earthing to the water pipe lines.
- Do not use water on electrical equipment.
- Discharge static voltage in HV lines/equipment and capacitors before working on them.

Personal Protective Equipment (PPE)

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

Personal protective equipment

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

Categories of PPE-Small's'

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

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

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

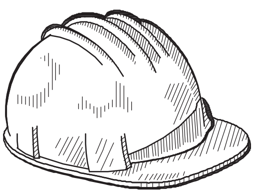
They are to meet the applicable BIS (Bureau of Indian Standards) standards for different types of PPE.

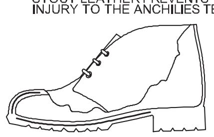
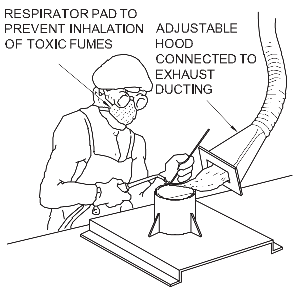
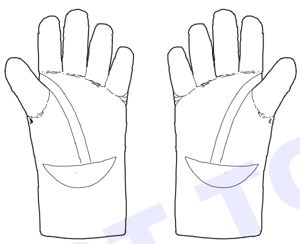
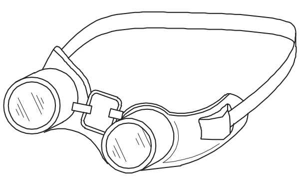
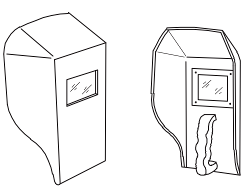
The guidelines on 'Personal Protective Equipment' is issued to facilitate the plant management in maintaining an effective program with respect to protection of persons against hazards, which cannot be eliminated or controlled by engineering methods listed in table 1.

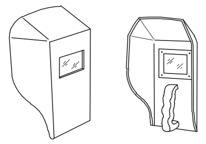
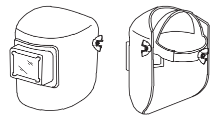
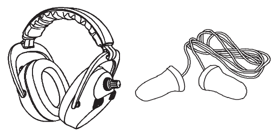
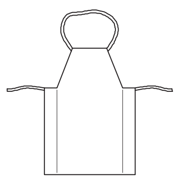
Table 1

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

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

Types of protection	Hazards	PPE to be used
Head protection (Fig 1) HELMET 	1 Falling objects 2 Striking against objects 3 Spatter	Helmets

Types of protection	Hazards	PPE to be used
Foot protection (Fig 2)   	1 Hot spatter 2 Falling objects 3 Working wet area	Leather leg guards Safety shoes Gum boots
Nose (Fig 3)  	1 Dust particles 2 Fumes/Gases/Vapours	Nose mask
Hand Protection (Fig 4)  GLOVES	1 Heat burn due to direct contact 2 Blows spark moderate heat 3 Electric shock	Hand gloves
Eye protection (Figs 5 & 6)   HAND SCREEN	1 Flying dust particles 2 UV rays, IR rays heat and 3 High amount of visible	Goggles Face shield radiation Hand shield Head shield

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

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

Selection of PPE's requires certain conditions

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

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

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

Response to emergencies

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

Power failure, System failure & Fire

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

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

2 **System failure**

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

1 Assassin bug

2 Lightening bug

3 Brain bug

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

3 **Fire**

When fire alarm sounds in your buildings

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

Report an emergency

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

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

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

Call emergency service

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

Report your location

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

Importance of housekeeping

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

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

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

Good shop floor practices followed in industry: Good Shop floor practices are motivating action plans for improvement of the manufacturing process.

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

Introduction to 5S concept and its application

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

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

1 Sort

2 Set in order

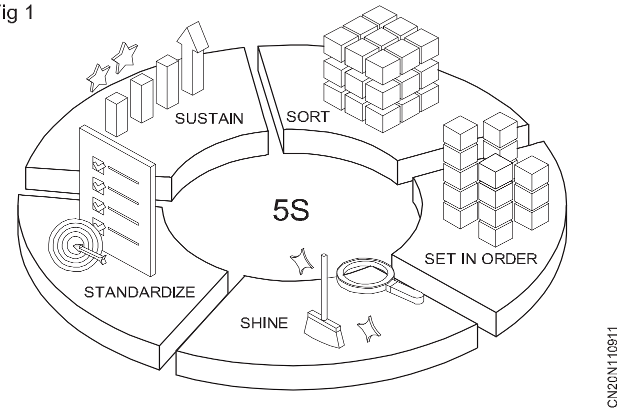
3 Shine

4 Standardize

5 Sustain

The steps of 5S (Fig 1): 5S was created in Japan, and the original "S" terms were in Japanese, so English translations for each of the five steps may vary. The basic ideas and the connections between them are easy to understand, though.

Fig 1

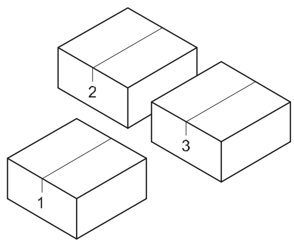


Step Name	Japanese term	Explanation
1	Sort Seri (tidiness)	Remove unnecessary items from each area
2	Set In Order	Seiton (orderliness) Organize and identify storage for efficient use
3	Shine Seiko (clean lines)	Clean and inspect each area regularly
4	Standardize	Seiketsu (standardization) Incorporate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

Step 1 sort

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

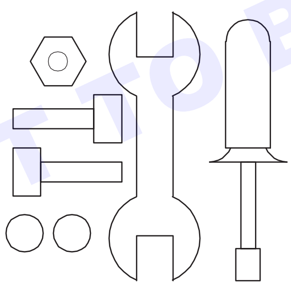
Fig 2



Step 2: Set in order

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

Fig 3



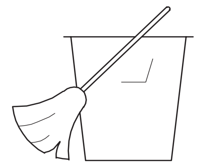
Implementation steps of set in order

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

Step 3: Shine

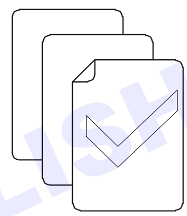
The third step of 5S is Shine, or "seiso," which means "cleanliness." While the first and second steps cleared up space and arranged the area for efficiency, this step attacks the dirt and grime that inevitably builds up underneath the clutter, and works to keep it from coming back. (Fig 4)

Fig 4



Step 4: Standardize: The fourth step is Standardize, or "seiketsu," which simply means standardization. By writing down what is being done, where, and by whom, you can incorporate the new practices into normal work procedure. This paves the way for long-term change. (Fig 5)

Fig 5

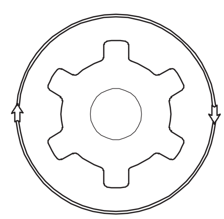


Tools for standardizing

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

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

Fig 6



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

- Management support
- Department tours
- Updated training
- Progress audits
- Perfo

Occupational safety and health

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

Occupational safety, and health

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

The goal of Occupational safety and health

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

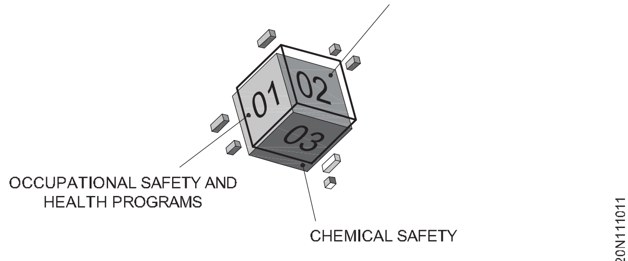
Environmental safety

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

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

Environmental safety is usually divided into three subcategories: (Fig1) Occupational safety and Health Programs, Environmental Control, and Chemical Safety. (Fig 1)

Fig 1



ENVIRONMENTAL SAFETY SEGMENTS

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

Similarly, government of India is enacted the following acts

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

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

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

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

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

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

The four important aims of health and safety legislation is to

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

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

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

iv eliminate workplace risks at the source.

Occupational safety and health tips

- Be aware of your surroundings
- Maintain a correct posture
- Take break regularly
- Use equipment properly
- Locate emergency exits
- Report unsafe conditions
- Practice effective housekeeping
- Make use of mechanical aids
- Wear the correct safety equipments
- Reduce workplace stress

Basic understanding on hot work, confined space work and material handling equipment

Objectives: At the end of this lesson you shall be able to
- **state what is hot working**
- **brief confined space work**
- **use of material handling equipment.**

Hot work

Hot work is defined as forging, gas cutting, welding, soldering and brazing operations for construction, maintenance/repair activities.

Hot work fire and explosive hazards. Workers performing hot work such as welding, gas cutting, brazing, soldering are exposed to the risk of fires from ignition or flammable or combustible materials in the space, and from leaks of flammable gas into the space, from hot work equipment.

A confined space also has limited or restricted means for entry or exist and is not designed for continuous occupancy. It includes but are not limited to tanks, vessels, silos, storage bins, hoppers, vaults, pits, manholes, tunnels, equipment housings, duct work, pipelines, etc.

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

Lifting and handling loads

Objectives: At the end of this lesson you shall be able to
- **state the types of injury caused by the improper method of lifting and carrying loads and how to prevent them**
- **state the 6 points in the process of manual lifting methods.**

Many of the accidents reported involve injuries caused by lifting and carrying loads. Wrong lifting techniques can result in injury.

A load need not necessarily be very heavy to cause injury The wrong way of lifting may cause injury to the muscles and joints even though the load is not heavy.

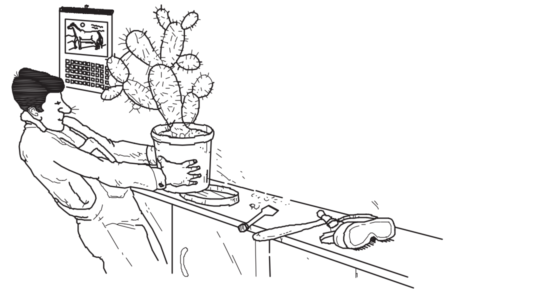
Further injuries during lifting and carrying may be caused by tripping over and object and falling or striking an object with a load.

Type of injury and how to prevent them?

Cuts and abrasions: Cuts and abrasions are caused by rough surfaces and jagged edges:

By splinters and sharp or pointed projections. (Fig 1)

Fig 1



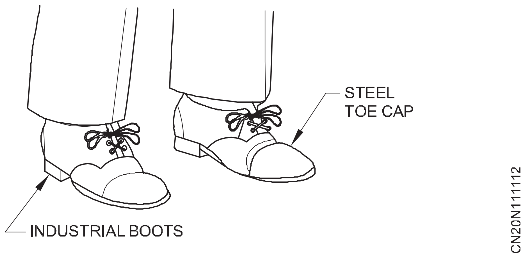
Leather hand gloves will usually be sufficient for protection, but the load should be checked to make sure of this, since large or heavy loads may involve body contact as well.

Crushing of feet or hands

Feet or hands should be so positioned that they will not be trapped by the load. Timber wedges can used when raising and lowering heavy loads to ensure fingers and hands are not caught and crushed.

Safety shoes with steel toe caps will protect feet (Fig 2)

Fig 2



Strain to muscles and joints

Strain to muscles and joints may be result of:

- Lifting a load which is too heavy, or of lifting incorrectly.

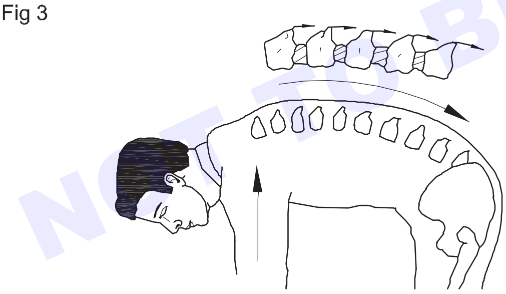
Sudden and awkward movements such as twisting or jerking during a lift can put severe strain on muscles.

Stop lifting'-lifting from a standing position with the back rounded increases the chance of back injury.

The human spine is not an efficient weight lifting machine and can be easily damaged if incorrect techniques are used.

> **The stress on a rounded back can be about six times greater than if the spine is kept straight. Fig 3 shows and example of stoop lifting.**

Fig 3



Preparing to lift

Before lifting or handling any load ask yourself the following questions.

What has to be moved?

Where from and where to?

Will assistance be required?

Is the route through which the load has to be moved is clear of obstacles?

Is the place where the load has to be kept after moving is clear of obstacles?

Load which seems light enough to carry at first will become progressively heavier, the farther you have to carry it.

The person who carries the load should always be able to see over or around it.

The weight that a person can lift will vary according to:

- Age
- Physique, and
- Condition

It will also depend on whether one is used to lifting and handling heavy loads.

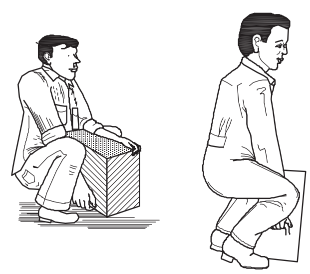
What makes an object difficult to lift and carry?

- Weight is not the only factor which makes it difficult to lift and carry.
- The size and shape can make an object awkward to handle.
- Loads high require the arms to be extended in front of the body, place more strain on the back and stomach.
- The absence of hand holds or natural handling points can make it difficult to raise and carry the object.

Correct manual lifting techniques

- Approach the load squarely, facing the direction of travel
- The lift should start with the lifter in a balanced squatting position, with the legs slightly apart and the load to be lifted held close to the body.
- Ensure that a safe firm hand grip is obtained. Before the weight is taken, the back should be straightened and held as near the vertical position as possible. (Fig 4)

Fig 4

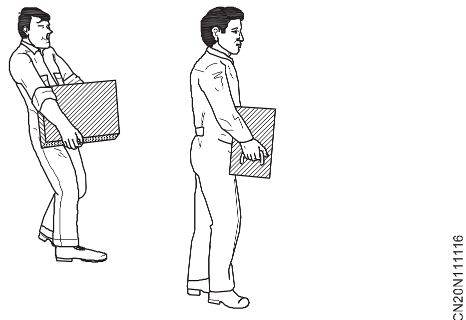


- To raise the load, first straighten the legs. This ensures that the lifting strain is being correctly transmitted and is being taken by the powerful thigh muscles and bones.
- Look directly ahead, not down at the load while straightening up, and keep the back straight, this will ensure a smooth, natural movement without jerking or straining. (Fig 5)

To complete the lift, raise the upper part of the body to the vertical position. When a load is near to an individual's maximum lifting capacity it will be necessary to lean back on the hips slightly (to counter balance the load) before straightening up. (Fig 6)

Fig 5

Fig 6

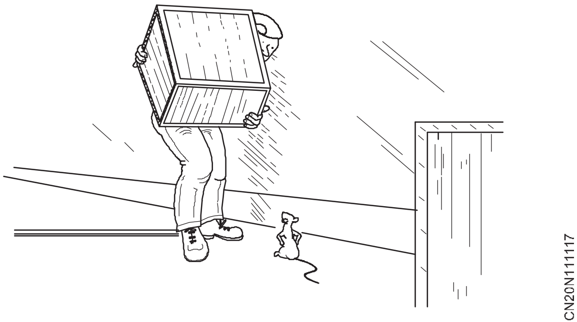


Keeping the load well near to the body, carry it to the place where it is to be set down. When turning, avoid twisting from the waist- turn the whole body in one movement.

Lowering the load

Make sure the area is clear of any obstructions. (Fig 7)

Fig 7



Bend the knees to a semi- squatting position, keep the back and head erect by looking straight ahead, not down at the load. It may be helpful to rest the elbows on the thighs during the final stage of lowering.

Fundamentals of work piece rotation

Objectives: At the end of this lesson you shall be able to
- **state the factors of workpiece rotation**
- **explain various features of workpiece rotation.**

Work piece rotation

In machining operations, work piece rotation is crucial for achieving accurate and precise machining operations, ensuring proper work piece rotation and minimizing errors and defects.

Various features of work piece rotation

1 Speed

The speed at which work piece rotates. Measured as revolution per minute (RPM)

2 Direction

This is orientation with cutting tool action. By which the work piece rotates, either clockwise (CW) or counter clockwise (CCW).

3 Axis of rotation

The axis around which the work piece rotates, like z-axis in turning and x-axis milling machine.

4 Accuracy

The factors such as runout, vibration and wobbling of work piece to be eliminated.

5 Balance

Balancing factors such as weight distribution centre of gravity and moment of inetia to be considered. Including factors such as angular velocity/acceleration and torque.

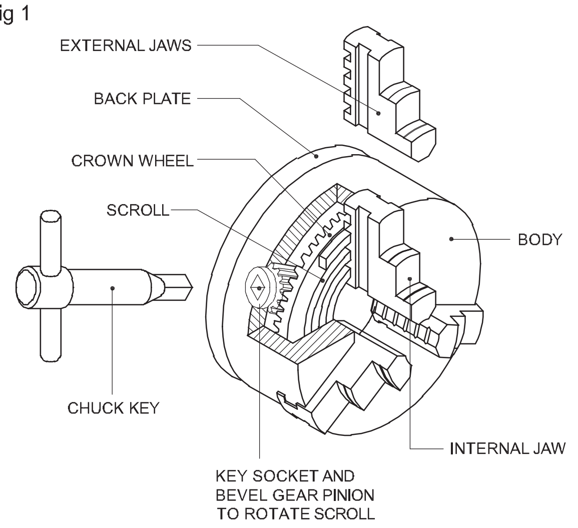
By controlling the work piece rotation, machinists can achieve precise control over the machining process.

Three jaw / four jaw chucks, face plate - clamping of work piece

Objectives: At the end of this lesson you shall be able to
- **identify the parts of three jaw/four jaw and face plate**
- **state the uses of three jaw/four jaw and face plate**
- **list the comparison between three jaw and four jaw chucks.**

The three jaw chuck (Fig 1)

Fig 1



The three jaw chuck is also known as self-centering chuck. Only perfectly round work or work with equally spaced flats, divisible by three should be held in a three jaw chuck.

The construction of a three jaw chuck shows that the scroll not only clamps a component in place but also locates the component.

The jaws of this type of chuck are not reversible and separate internal and external jaws have to be used.

Parts of a three jaw chuck (Fig 1)

- Back plate
- Body
- Jaws
- Crown wheel
- Pinion

Back plate

The back plate is fastened at the back of the body by means of alien screws. It is made out of cast iron. Its bore is tapered to suit the taper of the spindle nose.

Body

The body is made out of cast steel and the face is hardened. The body has three openings - 120° apart to assemble the jaws and operate them. Three pinions are fixed on the periphery of the body to operate the jaws by means of a chuck key. The body is hollow in cross section. The crown wheel is housed inside the body.

Jaws

The jaws are made out of high carbon steel, hardened and tempered, which slide on the openings of the body. Generally, there are two sets of jaws, viz. external jaws and internal jaws.

Crown wheel

The crown wheel is made out of alloy steel, hardened and tempered. On one side of the crown wheel a scroll thread is cut to operate the jaws and the other side is tapered on which bevel gear teeth are cut to mesh the pinion. When the pinion is rotated by means of the chuck key, the crown wheel rotates, thus causing the jaws to move inward or outward depending upon the rotation.

Pinion

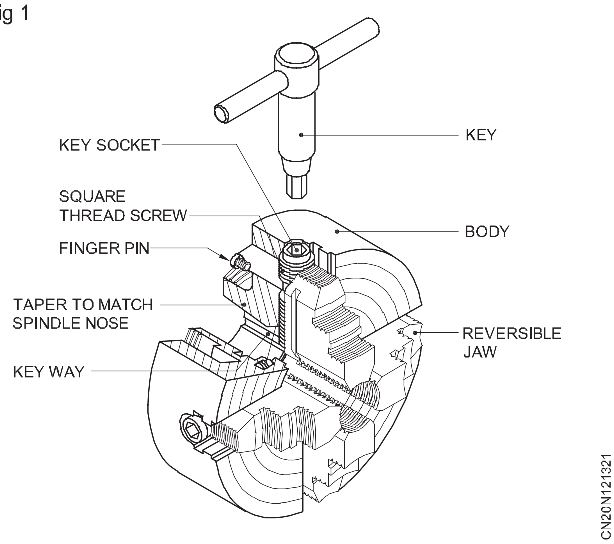
The pinion is made out of high carbon steel, hardened and tempered. It is fitted on the periphery of the body. On the top of the pinion, a square slot is provided to accommodate the chuck key.

Four jaw chuck

Objectives: At the end of this lesson you shall be able to
- **state comparison between three jaw and four jaw chucks.**

Four jaw chuck (Fig 1)

Fig 1



The four-jaw chuck is also known as an independent chuck, since each jaw can be adjusted independently; a work can be trued to within 0.001" or 0.02 mm accuracy using this chuck.

This type of chuck is much more heavily constructed than the self-centering chuck and has much greater holding power. Each jaw is moved independently by a square thread screw. The jaws are reversible for holding large diameter jobs. The independent four-jaw chuck has four jaws each working independently of the others in its own slot in the chuck body and actuated by its own separate square threaded screw. By suitable adjustment of the jaws, a workpiece can be set to run either true or eccentric with the machine centre.

Finished jobs when held in a four-jaw chuck can be trued with the help of a dial test indicator. (Fig 2)

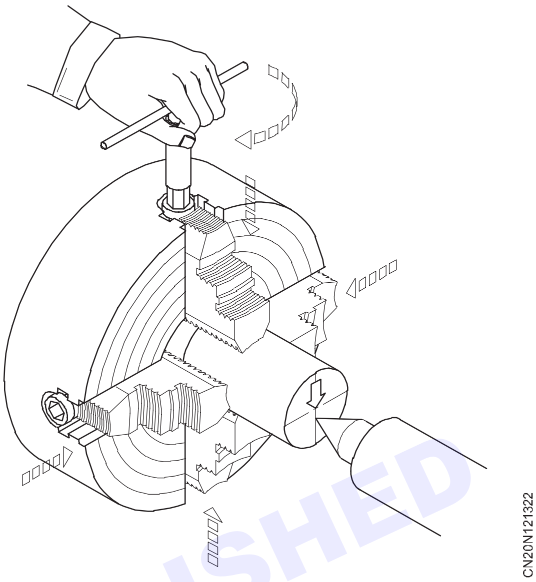
The parts of a four-jaw chuck are the

- back plate
- body
- jaws
- square threaded screw shaft

Back plate

The back plate is fastened to the back of the body by means of allen screws. It is made out of cast/iron steel. Its bore is tapered to suit the taper of the spindle nose. It has a keyway which fits into the key provided on the spindle nose.

Fig 2



Body

The body is made out of cast iron / cast steel and the face is flame-hardened. It has four openings 90° aprt to assemble the jaws and operate them. Four screws shafts are fixed on the periphery of the body by means of finger pins. The screw is rotated by means of a chuck key.

Jaws

The jaws are made out of high carbon steel, hardened and tempered which slide on the openings of the body. These jaws are reversible for holding hollow work.

The back side of the jaw is square-threaded which helps in fixing the jaws with the operating screws.

Screws shaft

The screw shaft is made out of high carbon steel, hardened, tempered and ground. The top portion of the screws shaft is provided with a square slot to accommodate the chuck key.

Comparison between a three jaw chuck and four jaw chuck

Three jaw chuck	Four jaw chuck
Only cylindrical or hexagonal work can be held	A wide range of regular and irregular shaped jobs can be held
Internal and external jaws are available	Jaws are reversible for external and internal holding
Setting up of work is easy	Setting up of work is difficult
Less gripping power	More gripping power.
Depth of cut is comparatively less	More depth of cut can be given
Heavier jobs cannot be turned	Heavier jobs can be turned
Workpieces cannot be set for eccentric turning	Workpieces can be set for eccentric turning
Concentric circles are not provided on the face	Concentric circles are provided
Accuracy decreases as chuck gets worn out	There is no loss of accuracy as the chuck gets worn out.

Specification of a chuck

To specify a chuck, it is essential to provide details of the

- type of chuck
- capacity of the chuck
- diameter of the body
- width of the body
- the method of mounting to the spindle nose

Examples

Three jaws self-centering chuck

Gripping capacity 450mm

Diameter of the body 500mm

Width of the body 125 mm

Tapered or threaded method of mounting

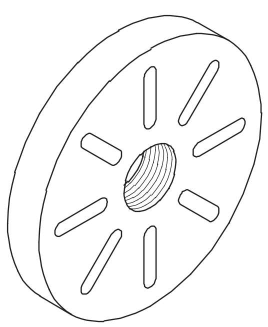
Face-plates

They are similar in construction to that of the lathe catch plates but are larger in diameter.

The different types of face-plates are:

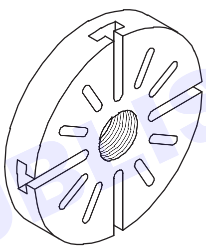
- Face-plates with only elongated radial slots (Fig 3)

Fig 3



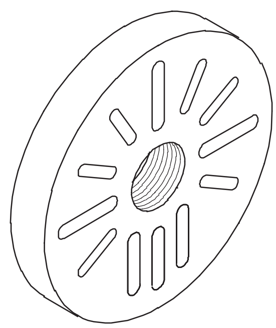
- Face-plates with only elongated slots and 'T' slots (Fig 4)

Fig 4



- Face-plates with elongated radial slots and additional parallel slots. (Fig 5)

Fig 5



Face-plates are used along with the following accessories.

Clamps, 'T' bolts, angle plates, parallels, counterweights, stepped blocks, 'V' blocks etc.

Uses

Large, flat, irregular shaped work pieces, castings, jigs and fixtures may be firmly clamped to a face-plate for various turning operations.

A work can be mounted on a face-plate while the face-plate is on the lathe spindle or on the workbench. If the workpiece is heavy or awkward to hold, the workpiece is mounted while the face-plate is on the workbench.

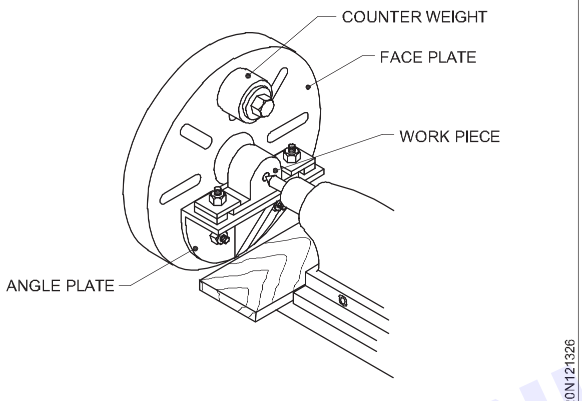
Before mounting the face-plate and it is set up to the spindle, it is advantageous to locate the workpiece on the face-plate and centre the workpiece. Centre a punch mark or hole approximately on the face of the workpiece. This makes it easier to true the work after the face-plate is mounted on to the spindle.

The position of the bolts and clamps is very important, if a workpiece is to be clamped effectively.

If a number of duplicate pieces are to be machined, the face-plate itself can be set up as a fixture, using parallel strips and stop blocks.

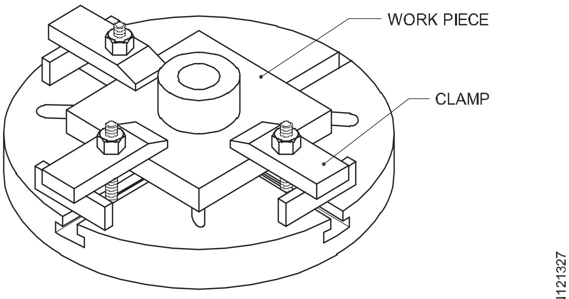
The application of the face-plate with the accessories in different set ups is shown in the sketches below. (Figs 6 to 8).

Fig 6



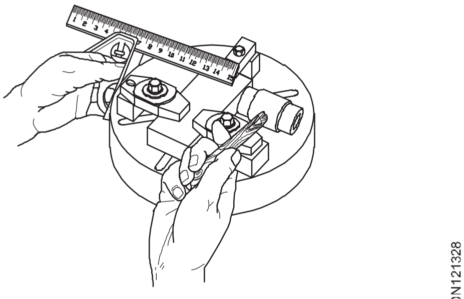
WORK HELD BY AN ANGLE PLATE

Fig 7



WORK CLAMPED DIRECTLY TO FACE PLATE

Fig 8



LOCATING WORK ON A FACE PLATE

Tool posts

Objectives: At the end of this lesson you shall be able to
- **identify and name the different types of tool posts**
- **state the constructional features of each types of tool post**
- **indicate the application of each type of tool post.**

Tool post

The tool post holds the tool or tools meant for the operation to be performed on the work.

The tool post is assembled to the top slide.

The three types of tool posts most commonly found lathes are listed here.

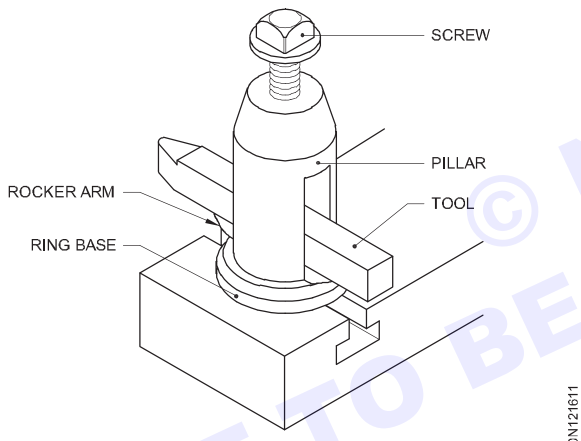
1 American type tool post or single way tool post

2 Indexing type of tool post or square tool post.

3 Quick change tool post.

Single way tool post (Fig 1)

Fig 1



AMERICAN PILLAR TYPE TOOL POST

It consists of a circular tool post body with a slot, for accommodating the tool or tool-holder. A ring base, a rocker arm, and a tool clamping screw complete the assembly of this type of tool post. The tool is positioned on the rocker arm and clamped. The centre height of the tool tip can be adjusted with the help of the rocker arm and the ring base. Only one tool can be fixed in this type of tool post. The rigidity of the tool is less as it is clamped with only one bolt.

Indexing type tool post (Fig 2)

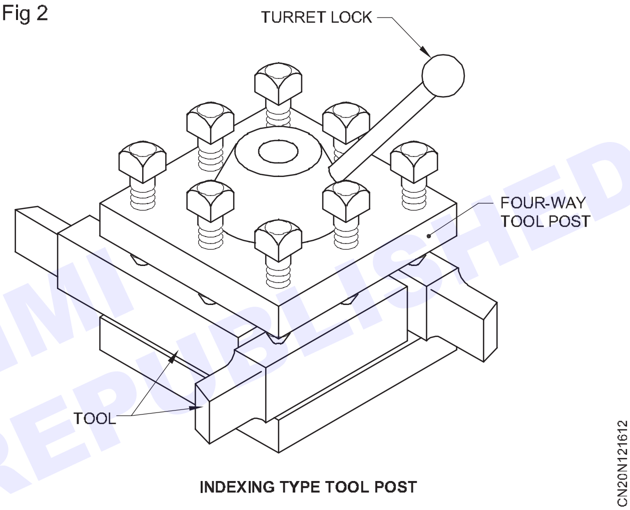
It is also called a square tool post or a four-way tool post. Four tools can be fixed in this type of tool posts and any one can be brought to the operating position and the square head is clamped with the help of the locking lever. By loosening the locking lever, the next tool can be indexed and brought to the operating position. The indexing may be manual or automatic.

The advantages are each tool is secured in the tool post by more than one bolt and so rigidity is more.

frequent changing of the tool for different operations need not be done as four tools can be clamped simultaneously.

The disadvantage is that skill is required to set the tools and it takes more time to set to the centre height.

Fig 2

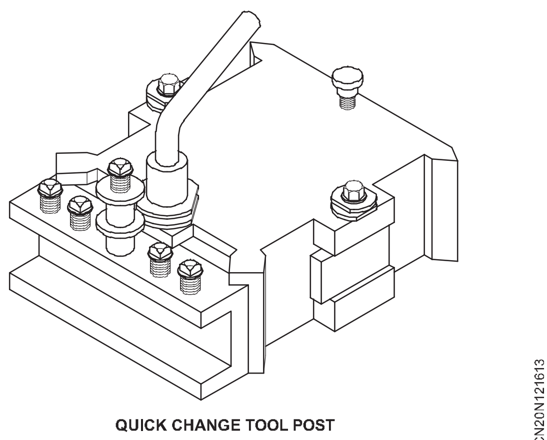


INDEXING TYPE TOOL POST

Quick change tool post/universal tool post (Fig 3)

Modern lathes are provided with this type of tool posts. Instead of changing the tools, the tool holder is changed in which the tool is fixed. This is expensive and requires a number of tool holders. But it has the advantage of ease with which it can be set to the centre height and has the best rigidity for the tool.

Fig 3



QUICK CHANGE TOOL POST

Tailstock and its setting to adjust taper

Objectives: At the end of this lesson you shall be able to
- **explain taper turning by setting over the tailstock**
- **name the parts of a tailstock**
- **state the purposes of a tailstock**
- **state the functioning of a tailstock.**

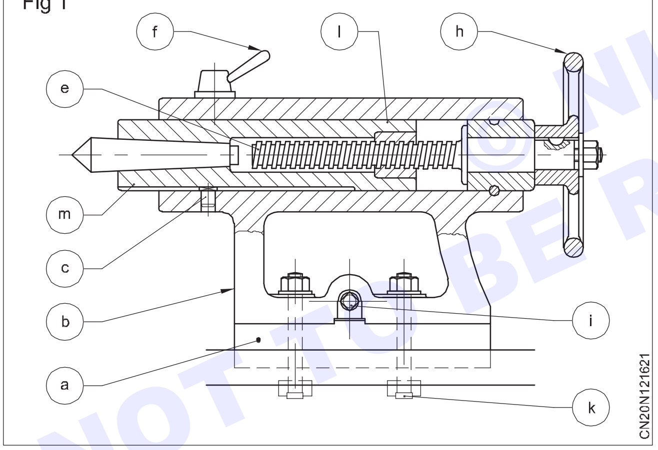
Tailstock

It is a sliding unit on the bed-ways of the lathe bed. It is situated on the right hand side of the lathe. It is made in two parts, namely the 'base' and the 'body'. The base bottom is machined accurately and has 'V' grooves corresponding to the bed-ways. It can be sided over the bed and clamped in any position on the bed by means of the clamping unit. The body of the tailstock is assembled to the base and has a corresponding longitudinal movement as to that of the base, along the bed. It has a limited transverse movement as well, with respect to the base. Graduations are marked on the rear end of the base and a zero line is marked on the body.

When both zero lines coincide the axis of the tailstock is in line with the axis of the headstock.

The body and base are made of cast iron. The parts of a tailstock are: (Fig 1)

Fig 1



- base (a)
- body (b)
- spindle (barrel) (c)
- spindle-locking lever (f)
- operating screw rod (e)
- operating nut (l)
- tailstock hand wheel (h)
- key(m)
- clamping unit (k)
- set over screw (i).

Functioning of tailstock

By rotating the hand wheel, the screw rod is operated. This causes the barrel, which carries the nut, to move forward and backward according to the direction of rotation. The key, which fits in the keyway milled at the bottom of the barrel, prevents the barrel from rotation. The thread in the screw rod is mostly of left hand square thread to have forward movement for anticlockwise rotation of the hand wheel. The barrel may be locked in any required position. The hollow end of the barrel at the front is provided with a Morse taper to accommodate the cutting tools with the taper shank. Graduations may be marked on the barrel to indicate the movement of the barrel. The screw rod is made of alloy steel and the operating nut is made of bronze. With the help of the adjusting screws, the body can be moved over the base laterally and the amount of movement may be read approximately referring to the graduations marked.

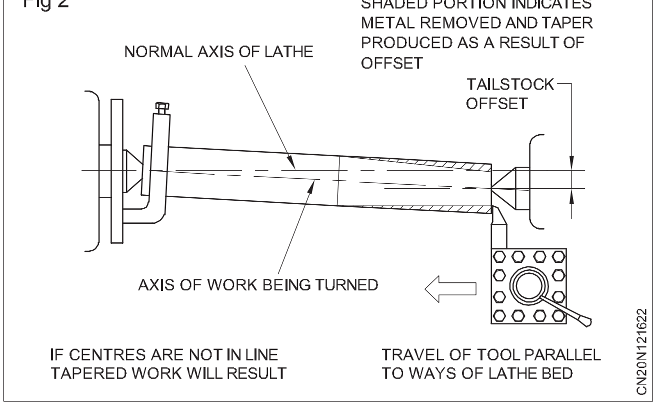
Purpose of the tailstock

To accommodate the dead centre to support a lengthy work for carrying out lathe operations.

To hold cutting tools like drills, reamers, drill chucks which are provided with taper shank.

To turn the external taper by offsetting the body of the tailstock with respect to the base. (Fig 2)

Fig 2



To perform external operations on the shaft held between centres.

Taper turning by the tailstock offset method

Objectives: At the end of this lesson you shall be able to
- **state the principle of taper turning by the tailstock offset method**
- **identify the parts involved while taper turning by the tailstock offset method**
- **calculate the amount of offset according to the expression of taper.**

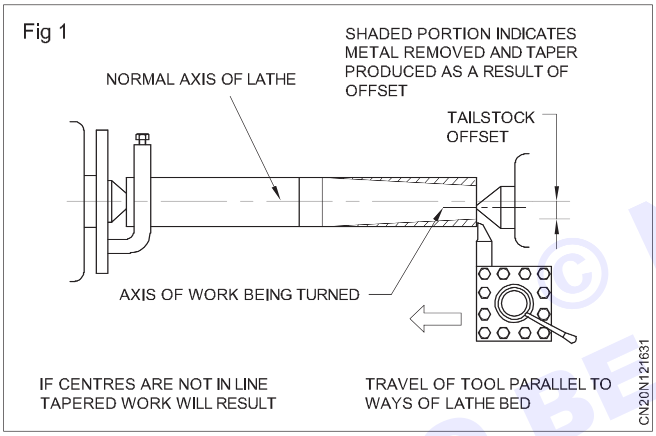
Principle of taper turning by the tailstock offset method

The job is held at an angle to the lathe axis, equal to half the included angle of the taper, and the tool is fed parallel to the axis.

As the job is held at an angle, it is possible to hold the work in between centres only as shown in the Fig 1. The parts involved during turning, the taper by the offsetting tailstock are:

- Live centre and dead centre.
- Tailstock assembly of body and base.
- Driving plate/catch plate.
- Lathe carrier.

Fig 1



The centres used should preferably be ball centres to avoid distortion or damage to the centre - drilled holes of the job. To avoid more load and wear and tear on centres, the tailstock will not be usually offset more than 1/50th of the length of the workpiece.

Calculation of the amount of offset

If the taper is expressed by giving the total length of taper (L) big dia. (D) the small dia. (d) the length of taper (l), then

$$\text{offset} = \frac{(D - d) \times L}{2l}$$

where L = total length of job, l length of taper

Example

The big diameter of a tapered job (D) = 30 mm.

The small diameter of the tapered job (d) = 26 mm.

The length of taper portion (l) = 100 mm.

Total length of job (L) = 200 mm.

$$\text{offset} = \frac{(D - d) \times L}{2l}$$

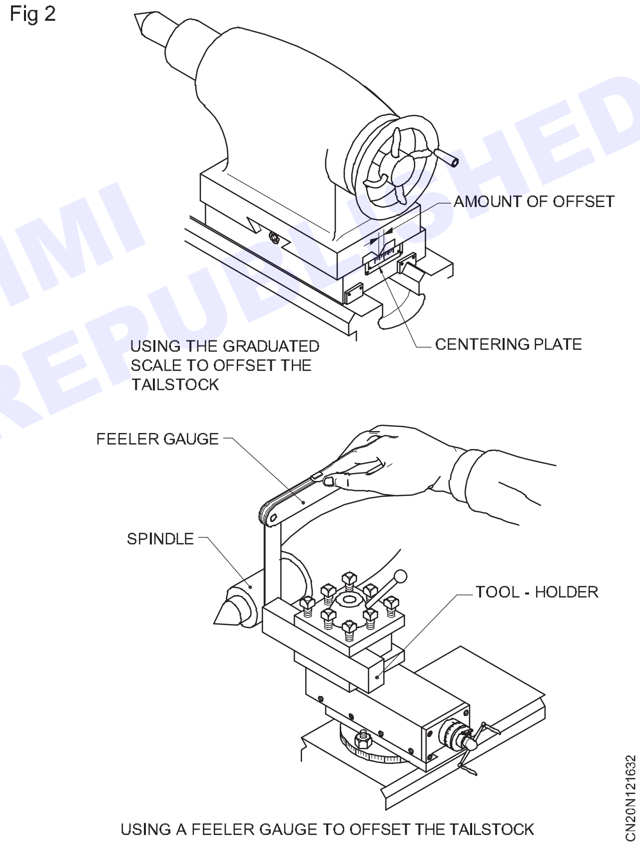
$$= \frac{(30 - 26) \times 200}{2 \times 100}$$

$$= \frac{4 \times 200}{2 \times 100}$$

= 4 mm

Different methods of offsetting the tailstock (Fig 2)

Fig 2



Setting offset with the help of the inside measuring jaws of a vernier caliper to the required mm, if directed graduation is not provided on the base of the tailstock.

Using a dial test indicator.

Using a cross - slide graduated collar and feeler gauge

Center height adjustment of tool

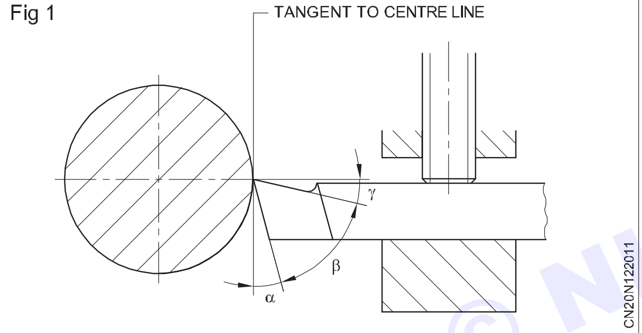
Objectives: At the end of this lesson you shall be able to
- **set the tool in the tool post for performing the operation.**

For optimum cutting, the effective rake angle and clearance angle of the clamped tool must be equal to the ground angles of the tool. This requires clamping of the tool to have its axis perpendicular to the lathe axis, with the tool tip at the workpiece centre. (Fig 1)

It is difficult to determine the effective angles of the tool when it is not set to the centre height.

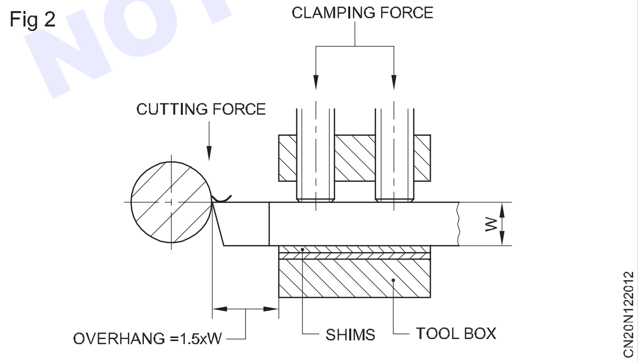
The tool nose can be set to the work centre by means of a tool-holder with adjustable height. (Fig 1)

Fig 1



The tool nose can be set to the exact centre height by placing the tool in the tool post on the shims or packing strips. These packing strips should be preferably a little less in width than the width of the tool but should never be more. The length of these strips should be according to the shank length and the tool seating face of the tool post. (Fig 2)

Fig 2

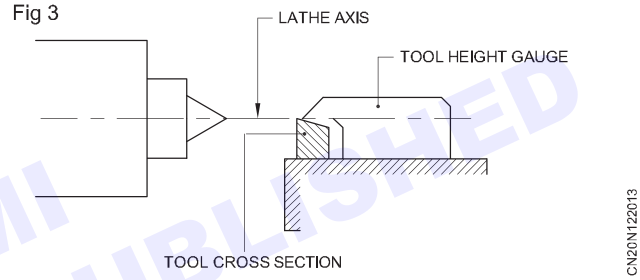


Use a minimum number of shims for height adjustment

The unsupported length of the overhanging end of the turning tool should be kept to a minimum. As a rule, the overhanging length of tool is equal to the tool shank width x 1.5.

Check the centre height with a height setting gauge. (Fig 3)

Fig 3



Tighten the other two tool-holding screws alternately applying the same amount of pressure.

When both the screws have a full gripping pressure, tighten the centre screw fully.

Check once again with a tool height setting gauge.

The gauge should be made according to the size of the machine. If a gauge is not available, use a surface gauge and set the pointer tip to the dead centre height fixed in the tailstock. Use this as the height to which the tool is to be set.

Inspection quality of product by using measuring instruments like vernier, micrometer etc.

Objectives: At the end of this lesson you shall be able to
- **explain the inspection parameters for quality product**
- **state the inspection characteristics of product.**

Inspection quality of a product using measuring instruments involves verifying its conformity to specified requirements and tolerances.

The specified parameters are as listed below:

1 **Accuracy:** Products dimensions, shape and features

2 **Precision:** Ensured by precision measuring instruments.

3 **Repeatability:** Verifying the measurements results are consistent.

4 **Tolerance:** Products dimensions and features are within specified tolerance.

5 **Surface finish:** Evaluating the products surface finish.

6 **Geometrical tolerance:** Verifying products features, roundness and cylindricity etc.

For inspecting above parameters the following instruments like vernier caliper, micrometers, dial indicators, co-ordinate measuring machines, surface roughness testers etc.

The above instruments are periodically calibrated, inspectors are to be trained and certified and to maintain inspection record.

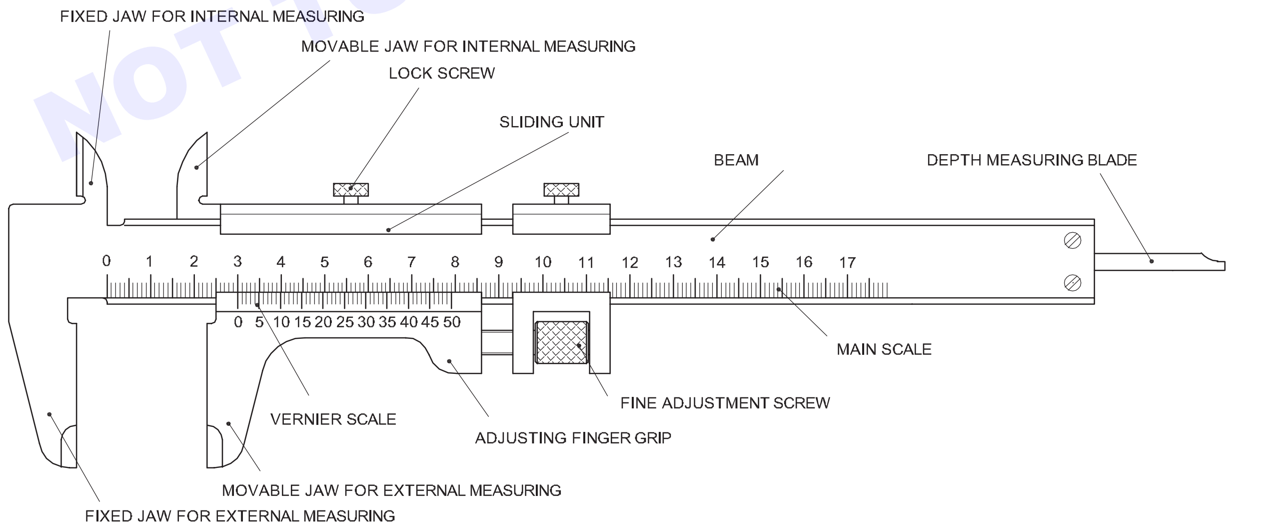
Vernier caliper

Objectives: At the end of this lesson you shall be able to
- **list out the parts of a vernier caliper**
- **state the constructional features of the vernier caliper**
- **state its functional features**
- **read a measurement.**

One of the precision instruments having the principle of vernier applied to it is the vernier caliper. It is known as a vernier caliper because of its application to take outside, inside and depth measurements. Its accuracy is 0.02 mm.

Vernier principle: The vernier principle state that two different scales are constructed on a single known length of line and the difference between then is taken for the measurement. (Fig 1)

Fig 1



Parts of a vernier caliper

A universal vernier caliper consists of a:

- Beam
- Fixed jaw for external measurements
- Movable jaw for external measurements
- Movable jaw for internal measurements
- Blade for depth measurement
- Main scale

- Vernier scale
- Fine adjustment screw
- Set of locking screws.

All parts are made out nickel-chromium steel or invar steel heat-treated and ground. They are machined to a high accuracy. They are stabilized to avoid distortion due to temperature variations.

Constructional features

The beam is the main part and the main scale graduations are marked on it. The markings are in millimeters and every tenth line is drawn a little longer and brighter than the other graduations and numbered as 1,2,3 ….

To the left of the beam the fixed jaws for external and internal measurements are fixed as integral parts., The vernier unit slides over the beam.

At the bottom face of the beam a keyway-like groove is machined for its full length, permitting the blade to slide in the groove.

At the bottom right hand end, a unit is fixed serving as a support for the blade when it slides in the groove.

The vernier unit has got the vernier graduations marked on it. The movable jaws for both external and internal measurements are integral with this.

The fixed and movable jaws are knife-edged to have better accuracy during measurement. When the fixed and movable jaws are made to contact each other, the zero of the vernier scale coincides with the zero of the main scale.

At this position in the blade will be in line with the right hand edge of the beam.

When the vernier scale unit slides over the beam, the movable jaws of both the measurements as well as the blade advance to make the reading.

To slide the vernier unit, the thumb lever is pressed and pulled or pushed according to the direction of movement of the vernier unit.

Sizes

Vernier calipers are available in sizes of 150 mm, 200 mm, 900 mm and 1200 mm. The selection of the size depends on the measurements to be taken. Vernier calipers are precision instruments, and extreme care should be taken while handling them.

> **Never use a vernier caliper for any purpose other than measuring.**
>
> **Vernier calipers should be used only to measure machined or filed surfaces.**
>
> **They should never be mixed with any other tools.**
>
> **Clean the instrument after use, and store it in a box.**

Graduations and reading of vernier calipers

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

Vernier calipers

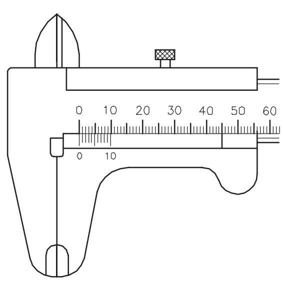
Vernier calipers are available with different accuracies. The selection of the vernier caliper depends on the accuracy needed and the size of the job to be measured.

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

Determining least count of vernier calipers

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

Fig 1



i.e. One main scale division (MSD)	=	1 mm
One vernier scale division? (VSD)	=	9/10 mm
Least count is 1 mm - 9/10 mm	=	1/10 mm
The difference between one MSD and one VSD	=	0.1 mm

Example

Calculate the least count of the vernier given in Fig 2.

Reading vernier measurements

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

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

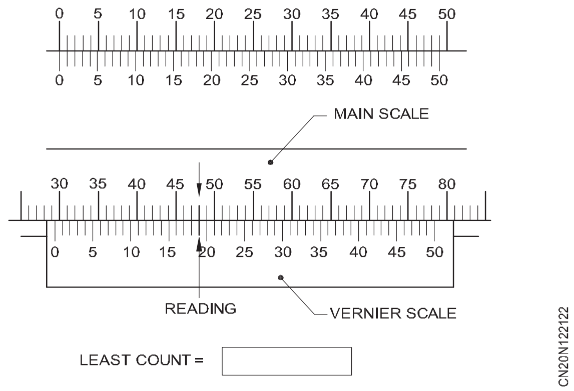


Fig 2

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

One Vernier scale division? (VSD) = 49/50 mm

Least count = 1 MSD - 1 VSD

= 1mm - 49/50

= 50 - 49/50 = 1/50 = 0.02 mm

Example for vernier caliper (Fig 3)

Main scale reading 60 mm.

The vernier division coinciding with the main scale is the 28th division. Value = 28 x 0.02

= 0.56 mm

Reading = 60+0.56 = 60.56mm

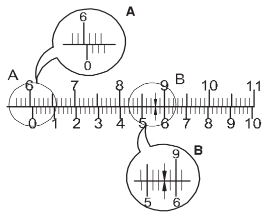


Fig 3

Classroom Exercise

In Figs 3, 4, 5, 6 and 7, 49 main scale divisions are divided into 50 equal parts on the vernier scale. Value of-one M.S.D. is 1 mm.

1 Calculate the least count.

2 Record the reading of each, figure in the space provided.

Fig 4

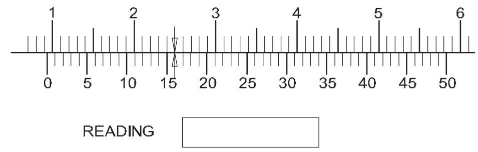


Fig 5

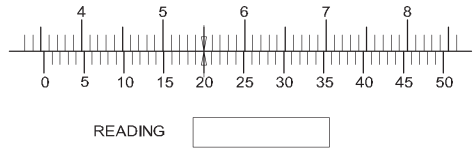


Fig 6

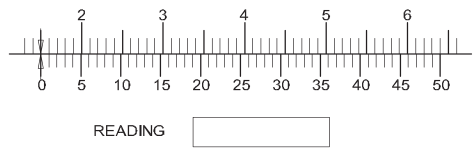
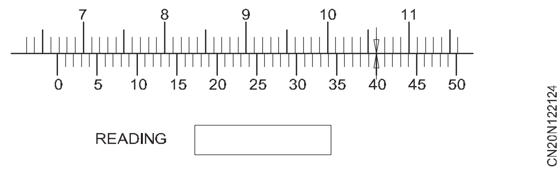


Fig 7



Disadvantages

Accuracy of reading depends on the skill of the operator.

Loses its accuracy by constant usage as slackness in the sliding unit develops.

Cannot be used to measure components having deviations less than ± 0.02 mm.

Possibility of parallax error during noting down the coinciding line may cause the reading of the measurement to be wrong.

To read a measurement

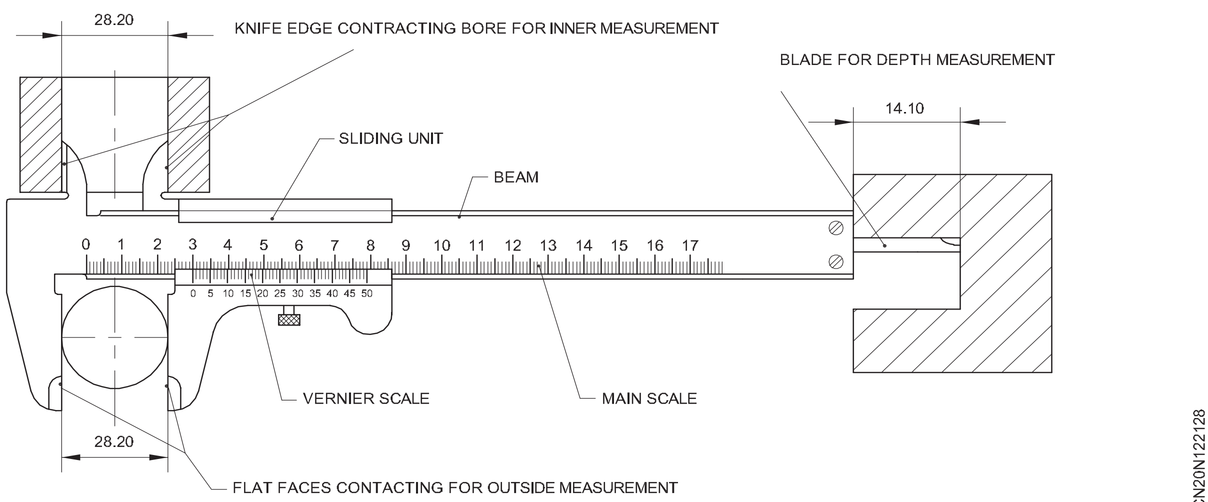
Note the number of graduations on the main scale passed by the zero of the vernier. This gives the full mm.

Note which of the vernier scale division coincides with any one line on the main scale.

Multiply this number with the least count.

Add the multiplied value to the main scale reading.

Fig 8



Digital vernier caliper

Objectives: At the end of this lesson you shall be able to
- **state the uses of digital caliper**
- **name the parts of a digital caliper**
- **brief the zero setting of a digital caliper.**

The digital caliper (sometime incorrectly called the digital vernier caliper) is a precision instrument than can be used to measure internal and external distance accurately to 0.01mm, the digital vernier caliper is shown in Fig 1. The distance or the measurements are read from LED drawing. The parts of digital calipers are similar to the ordinary vernier caliper except the digital display and few other parts. The parts are indicated in Fig 1.

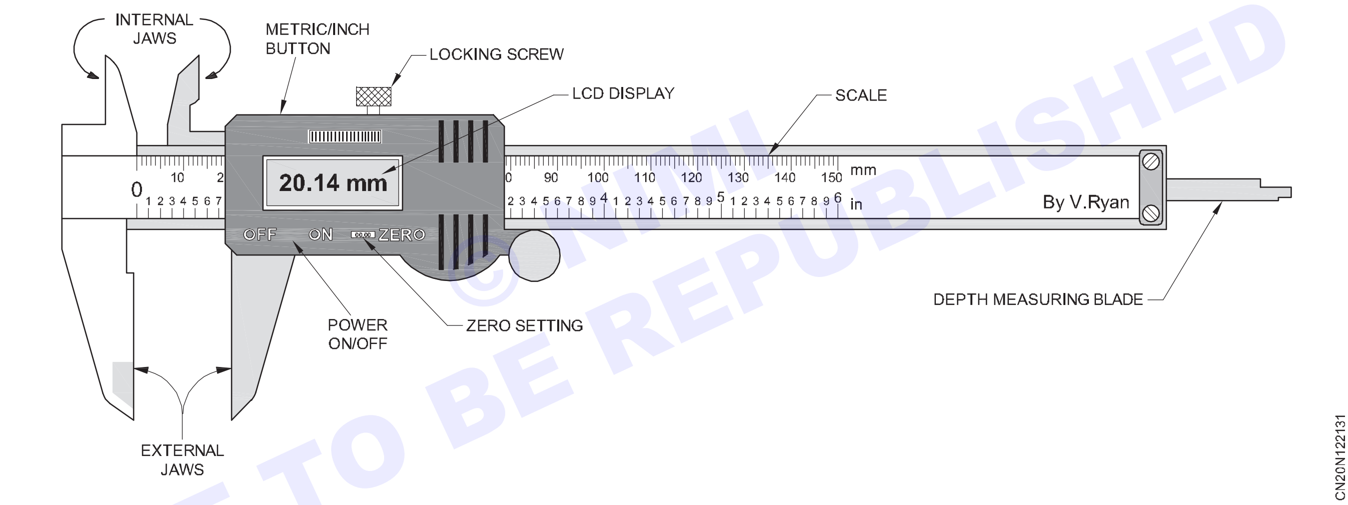
Earlier versions of the type of measuring instrument had to read by looking carefully at the inch or metric scale and there was a need for very good eye sight in order to read the small sliding scale. Manually operated vernier caliper are remain popular because they are much cheaper than the digital version.

The digital caliper requires a small battery whereas the manual version does not need any power source.The digital calipers are easier to use as the measurement is clearly display and also, by pressing inch/mm button the distance can be read as metric or inch.

The display is turned on with the ON/OFF button. before measuring, the zero setting to be done, by bringing the external jaws together until they touch each other and then press the zero button. Now the digital caliper is ready to use.

Always set zero position when turning on the display for the first time.

Fig 1



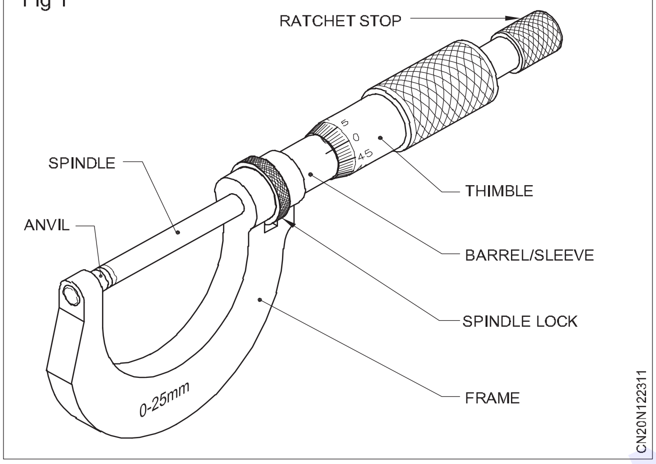
Outside micrometers, parts, principle

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

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

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

Fig 1



The parts of a micrometer are listed here.

Frame

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

Barrel/Sleeve

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

Thimble

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

Spindle

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

Anvil

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

Spindle lock nut

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

Ratchet stop

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

Graduations of metric outside micrometer

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

Working principle

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

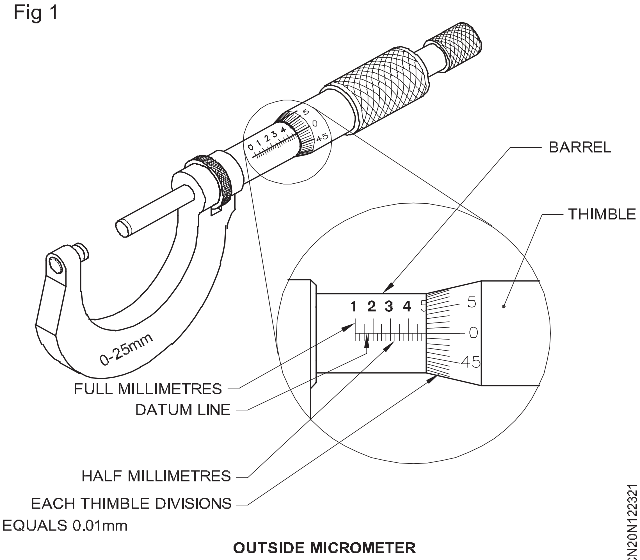
Graduations (Fig 1)

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

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

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

Fig 1



OUTSIDE MICROMETER

The circumference of the bevel edge of the thimble is graduated into 50 divisions and marked 0-5-10-15 .... 45-50 in a clockwise direction.

The distance moved by the spindle during one rotation of the thimble is 0.5 mm.

Movement of one division of the thimble = 0.5 x 1/50

= 0.01 mm

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

Reading dimensions with an outside micrometer

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

Ranges of outside micrometer

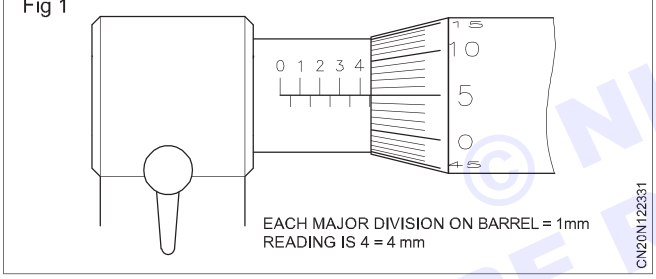
Outside micrometers are available in ranges of 0 to 25 mm, 25 to 50 mm, 50 to 75 mm, 75 to 100 mm, 100 to 125 mm and 125 to 150 mm.

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

Method of reading

Read on the barrel scale the number of whole millimeters that are completely visible from the bevel edge of the thimble. It reads 4 mm. (Fig 1)

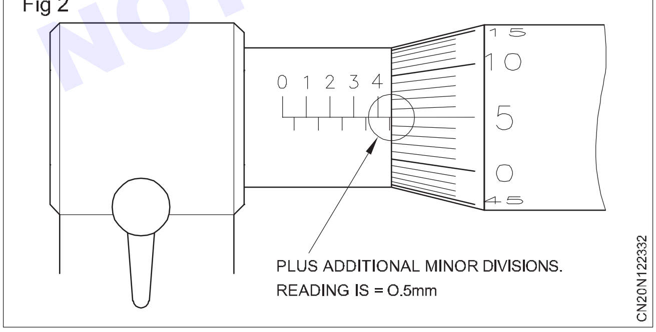
Fig 1



Add to this any half millimeters that are completely visible from the bevel edge of the thimble.

The figure reads 1/2 = 0.5 mm.(Fig 2)

Fig 2



Add the thimble reading to the two earlier readings.

The figure shows the 5th division of the thimble is coinciding with the index line of the sleeve. Therefore, the reading of the thimble is 5x0.01 mm = 0.05 mm. (Fig 3)

The total reading of the micrometer.

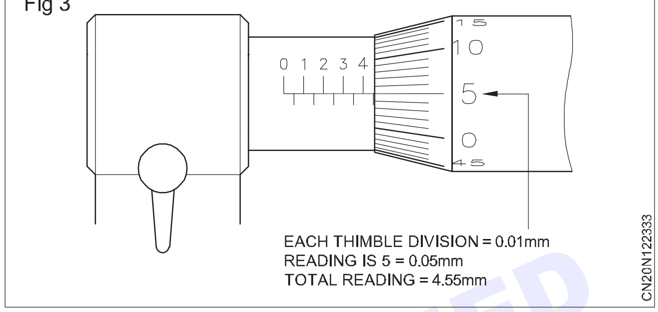
a 4.00 mm

b 0.50 mm

c 0.05 mm

Total reading 4.55 mm (Fig 3)

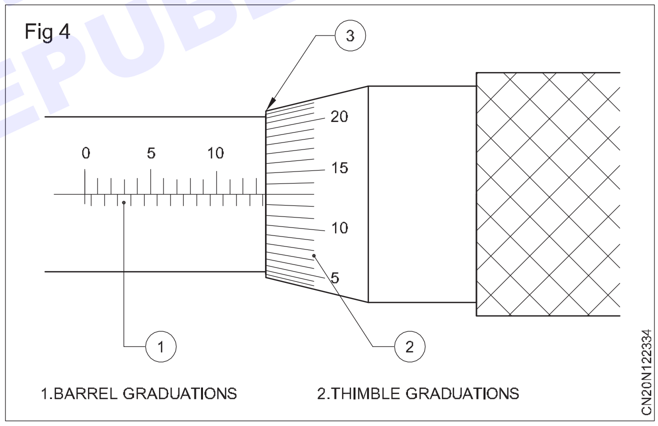
Fig 3



Reading micrometer measurements

How to read a measurement with an outside micrometer? (Fig 4)

Fig 4



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

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

13.00 mm

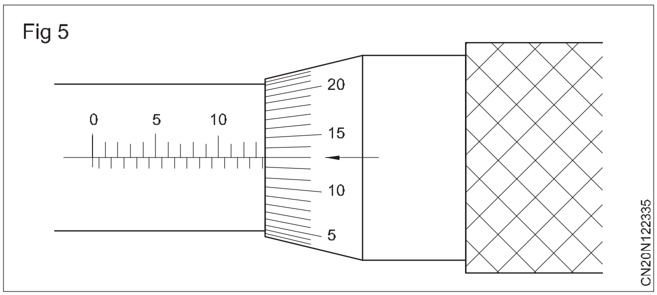
+ 00.50 mm

13.50 mm

Next read the thimble graduations.

Read the thimble graduations in line with the barrel datum line, 13th div. (Fig 5)

Fig 5



Multiply this value with 0.01 mm (least count).

13 x 0.01 mm = 0.13 mm.

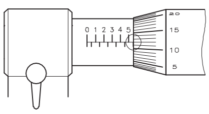
Add

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

Total	63.63 mm

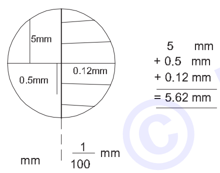
The micrometer reading is 63.63 mm.

Some examples of metric micrometer readings and their solution



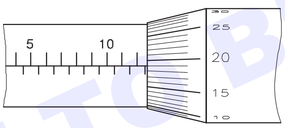
i 5.00 mm
0.50 mm
0.12 mm

Total 5.62 mm



iii 12.00 mm
0.50 mm
0.19 mm

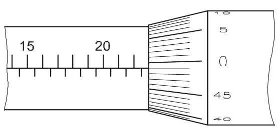
Total 12.69 mm



12.69mm

iiii 22.00 mm
0.50 mm
0.49 mm

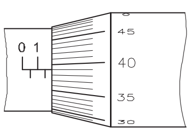
Total 22.99 mm



23.99mm

iv 1.00 mm
0.50 mm
0.39 mm

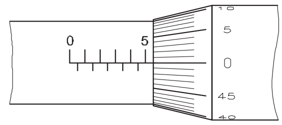
Total 1.89 mm



1.89mm

v 5.00
0.50 mm
0.00 mm

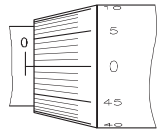
Total 5.50 mm



5.50 mm

vi 0.00 mm
0.50 mm
0.00 mm

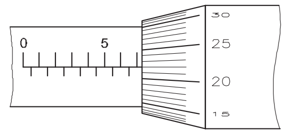
Total 0.50 mm



0.50 mm

vii 7.00 mm
0.00 mm
0.22 mm

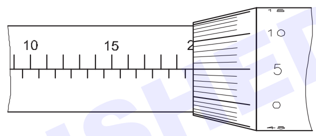
Total 7.22 mm



7.22 mm

viii 19.00 mm
0.50 mm
0.05 mm

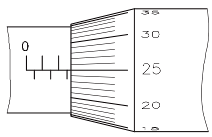
Total 19.55 mm



19.55 mm

ix 2.00 mm
0.50 mm
0.25 mm

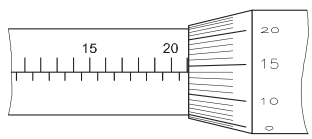
Total 2.75 mm



2.75 mm

x 21.00 mm
0.00 mm
0.14 mm

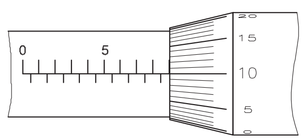
Total 21.14 mm



21.14 mm

xi 9.00 mm
0.00 mm
0.10 mm

Total 9.10 mm



9.10 mm

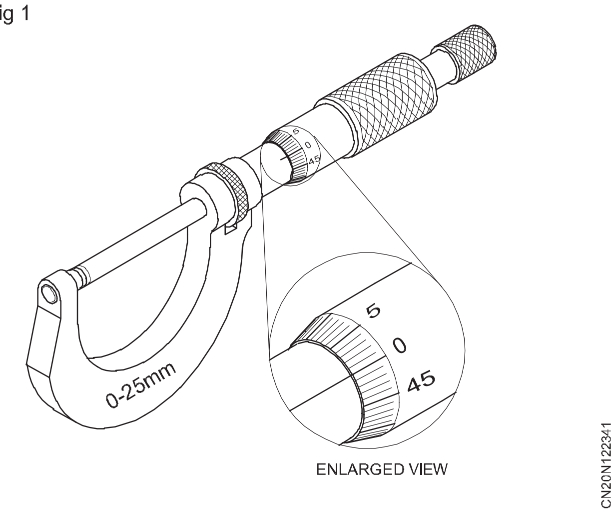
Error in micrometer

Objective: At the end of this lesson you shall be able to
- **check outside micrometer for '0' error.**

No zero error

When the measuring faces are in contact if the zero of the thimble should be coincide with the datum line No zero errors (Fig 1)

Fig 1



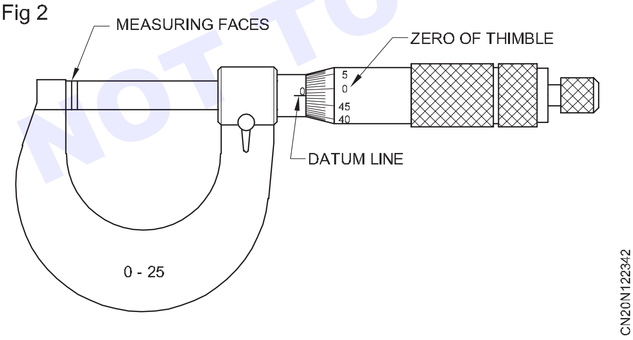
Zero error

When the measuring faces are in contact, (Fig 2) if the zero of the thimble do not coincide with the datum line (the zero of the thimble will be above or below the datum line) the micrometer is said to be with zero error. There are two types of zero error.

a Positive error

b Negative error

Fig 2

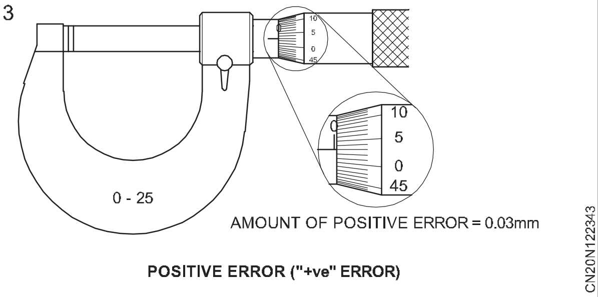


All micrometers should be checked for its zero error and the error should be noted if any before using it on checking dimensions.

Clean measuring faces with clean cloth before checking for zero error.

Positive error: When the anvil and spindle faces are in contact in case of a 0-25 mm micrometer or 0-1 micrometer and with a test piece in between the measuring faces in case of a higher range micrometer. If the zero of the thimble rest below the datum line of the sleeve the error is called as "Positive". (Fig 3)

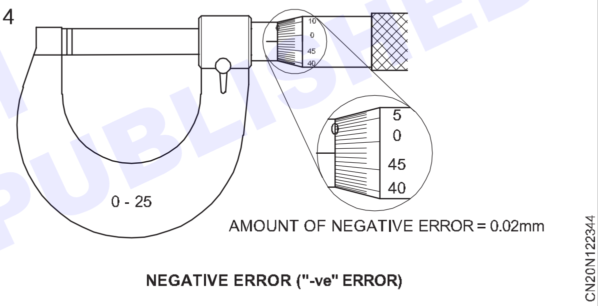
Fig 3



To get the correct reading the amount of error should be subtracted from the reading dimension.

Negative error: When the anvil and spindle faces are in contact, if the zero of the thimble passes above the datum line of the sleeve, the error is called as "Negative". (Fig 4)

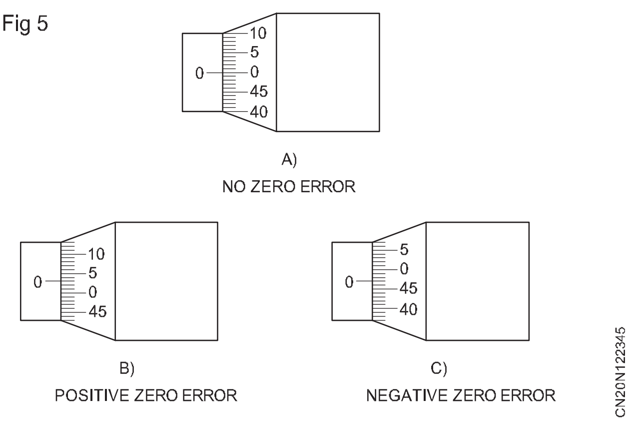
Fig 4



To get the correct reading the amount of negative error should be added to the reading dimension.

Caution: When you come across with micrometer having "zero error", inform your instructor and get it corrected by him. Do not try yourself to correct at this stage.

Fig 5



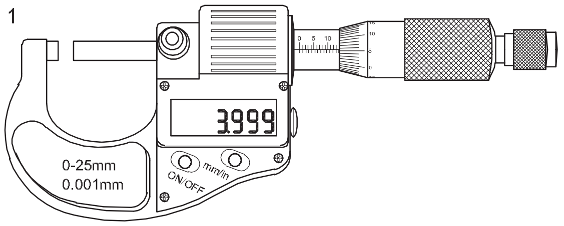
Digital micrometers

Objectives: At the end of this lesson you shall be able to
- **state the uses of digital micrometer**
- **list the parts of digital micrometer**
- **read the reading from LED display and thimble and barrel**
- **brief the maintenance, maintenance of digital micrometers.**

Digital micrometers is one of the simplest and most widely used measuring equipment in any manufacturing industry. Its simplicity and the versatile nature make Digital micrometers so popular. Different kinds of Digital micrometers available in the market.

Feature of digital micrometers (Fig 1)

Fig 1



FEATURE OF DIGITAL MICROMETERS

CN20N122351

- LCD displays measuring data and makes direct read out with resolution of 0.001mm.
- Origin setting mm/inch conversion, switch for absolute and incremental measurement.
- Carbide tipped measuring faces.
- Ratchet ensures invariable measurement and accurate repeatable reading

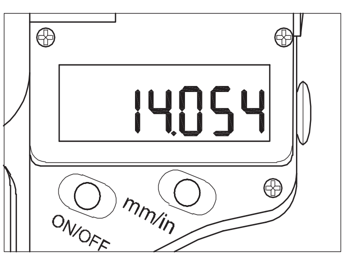
Accuracy of digital micrometers

Digital micrometers provide 10 times more precision and accuracy: 0.00005 inches or 0.001mm resolution, with 0.0001 inches or 0.001mm accuracy.

Reading of the digital micrometer

The digital micrometers are provided with high precision reading with LCD display. The reading is 14.054 mm as shown in Fig 2.

Fig 2

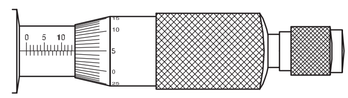


READING OF THE DIGITAL MICROMETERS

CN20N122352

Reading also by reading the marks on the sleeve and the thimble. Usually, the reading from the large LCD display for the digital micrometer because the digital reading is more accurate. The reading on the sleeve and the thimble is just for reference. Read the markings on the sleeve and the thimble, firstly, read the point which the thimble stops at it on the right of the sleeve (It is 14mm here, because each line above the centre long line represents 1mm while each line below the centre long line represent 0.5mm) (Fig 3)

Fig 3



CN20N122353

Secondly, read the markings on the thimble, It is between 5 and 6, So you need to estimate the reading. (It is 0.055mm for each line here represents 0.001mm). At last, add all the reading up: 14mm + 0.055 mm = 14.055mm. So the total reading is 14.055mm.

Maintenance of a digital micrometers

Never apply voltage (e.g. engraving with an electric pen) on any part of the Digital Micrometers for fear of damaging the circuit.

Press the ON/OFF button to shut the power when the Digital micrometers stands idle; take out the battery if it stands idle for a long time.

As for the battery, abnormal display (digit flashing or even no display) shows a flat battery. Thus you should push the battery cover as the arrow directing and then replace with a new one. Please note that the positive side must face out If the battery bought from market doesn't work well (the power may wear down because of the long-term storage or the battery's automatic discharge and etc.) Please do not hesitate to contact the supplier.

Flashing display shows dead battery. If this is the case, please replace the battery at once. No displace shows poor contact of a battery or short circuit of both poles of the battery. Please check and adjust pole flakes and battery insulator cover. In case water enters the battery cover, open the cover immediately and blow the inside of the battery cover at a temperature of no more than 40˚C till it gets dry.

Types of milling machines

Objectives: At the end of this lesson you shall be able to
- **state the introduction of milling machines**
- **classify the types of milling machines**
- **state the specification of milling machine.**

Introduction

A milling machine is a machine tool that removes metal as the work is fed against a rotating multipoint cutter. The cutter rotates at a high speed and because of the multiple cutting edges it removes metal at a very fast. The machine can also hold one or more number of cutters at a time. This is why a milling machine finds wide application in production work. This is superior to other machines as regards accuracy and better surface finish, and is designed for machining a variety of tool room work.

Principle of milling (cutting)

In milling, the cutter has a rotary movement, the speed of which depends upon the cutting speed required. Driving the milling arbor at various rotational speeds makes it possible to achieve approximately the same cutting speeds [peripheral speed] with cutters of different diameters.

While the milling cutter (a) rotates at a high speed, and because of the multiple points, it removes metal at a very fast rate, in comparison with other machine tools. (Fig 1)

Job (b) can be machined manually or automatically.

By milling we can produce flat (horizontal, vertical, angular) and formed surfaces. (Fig 2)

Fig 1

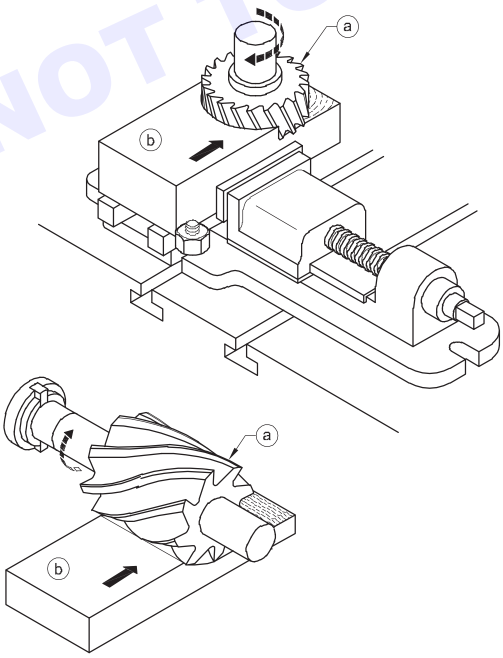
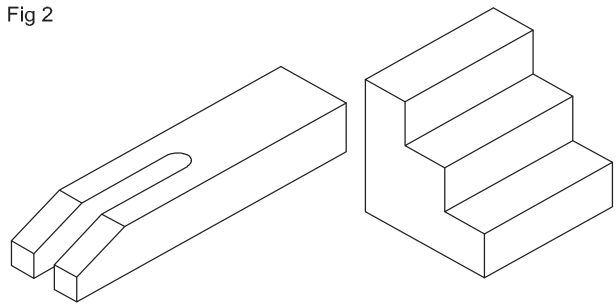


Fig 2



A milling machine finds wide application in production work as the machine can hold one or more number of cutters at a time, and is good in accuracy, surface finish etc.

Classification

The classification according to the general design of the milling machine is:

- column and knee type
- fixed bed type
- planer type
- special type

But out of these types the one that is used most in general workshop is the column and knee type machine.

In the column and knee type category the following machines are covered.

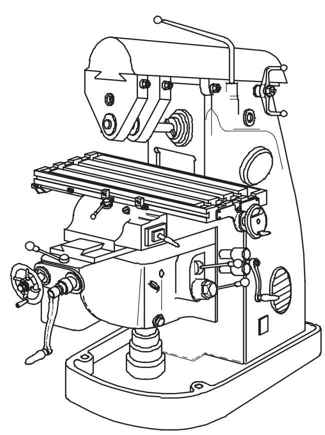
¨ Plain/horizontal milling machine (Fig 3)

¨ Vertical milling machine (Fig 4)

¨ Universal milling machine (Fig 5)

1 Plain milling machine (Fig 3)

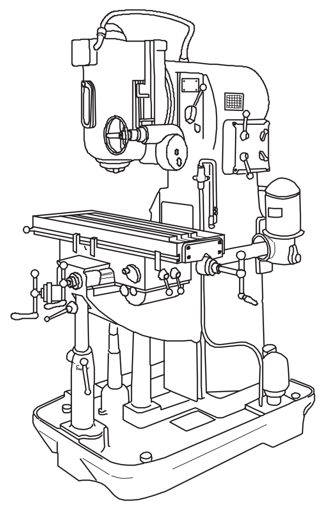
Fig 3



- More rigid and sturdy than other machines and accommodate heavy work.
- Table may be fed either by hand or by power against a rotating cutter and in the three directions namely longitudinately.
- Milling cutters mounted on the horizontal arbor.
- Spindle rotates horizontally and parallel to the machine table.

2 Vertical milling machine (Fig 4)

Fig 4

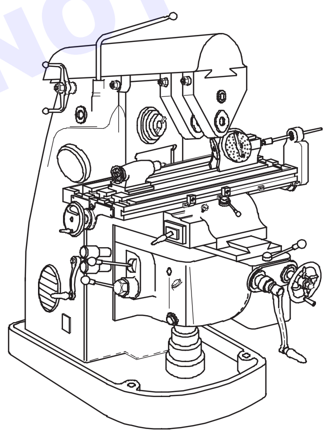


CN20N122414

- Distinguished from the horizontal milling machine by position of the spindle vertical or perpendicular to the work table.
- The spindle rotates about the vertical axis.
- Spindle moved up and down by spindle feed and also may be felted or swiveled.
- Most suitable for boring, pocket milling, profile milling and for making keyways.

3 Universal milling machine (Fig 5)

Fig 5

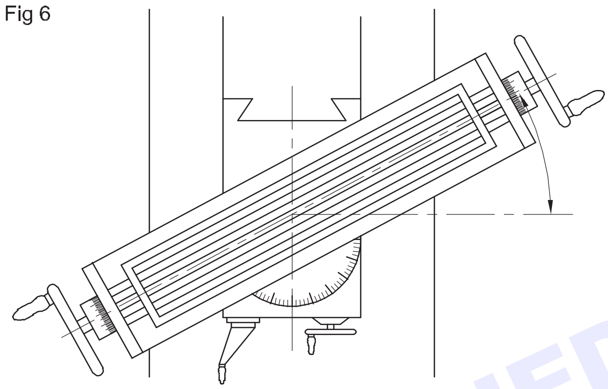


CN20N122415

- Table apart from having movements in 3 directions and also can be swivelled about its horizontal axis.
- Maximum swivelling can be made is 45° both in clockwise and anticlockwise direction.
- As angular feeding is possible suitable for milling helical gears, helical grooves etc,.

The universal milling machine is similar in construction to the plain machine. But its table, apart from having movements in 3 directions, can also be swivelled about the horizontal axis. The maximum swivelling is 45° both the clockwise and anticlockwise directions. (Fig 6)

Fig 6



CN20N122416

The swivelling of the table permits angular feeding. Because of this, the universal milling machine is suitable for milling helical gears, helical grooves, etc,.

This machine is supported with different attachments like vertical head, slotting head, rack-milling attachment.

Specification of a milling machine

The milling machine is normally specified by the

- dimension of the working surface of the table
- longitudinal travel of the table
- cross travel of the table
- vertical travel of the table
- number of spindle speeds
- spindle nose taper
- number of feeds
- floor space area, etc.

Different types of milling cutter and their uses - cutter nomenclature

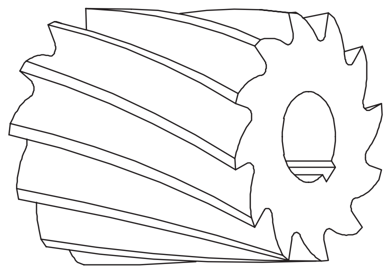
Objectives: At the end of this lesson you shall be able to
- **state the two main categories of milling cutters**
- **state the different types of plain milling cutters**
- **state the uses of plain milling cutters.**

Milling cutters

Milling cutters generally fall into two categories, solid cutters and inserted tool cutters.

Solid cutters (Fig 1)

Fig 1

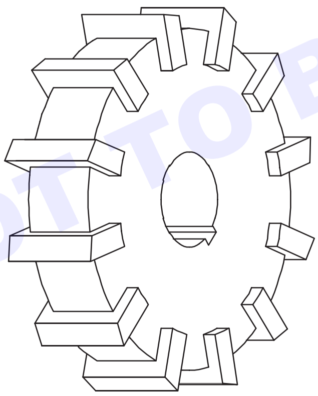


CN20N122511

These cutters are those in which the teeth have been cut into the body of the cutter. The teeth may be straight (parallel) or helical (at an angle) to the axis of the cutter. Solid type cutters are generally made of high speed steel.

Inserted tool cutter (Fig 2)

Fig 2



CN20N122512

These cutters have removable and replaceable teeth which are fastened or locked into the body of the cutter. The inserted tool construction is generally used on large cutters as the blades can be quickly replaced when they become dull.

Plain milling cutters

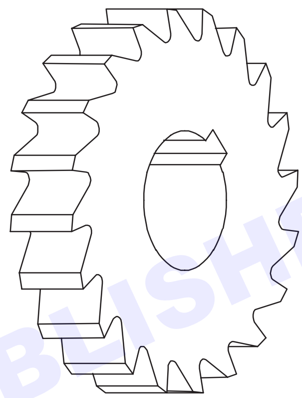
Plain milling cutters are cylindrical, having teeth on the periphery only. They are used to produce flat surfaces, by feeding the table longitudinally. The cutter teeth may be straight or helical according to the size of the cutter. Wider plain cutters are used for slab milling which are known as slab milling cutters.

Types of plain milling cutters

Light duty plain milling cutters

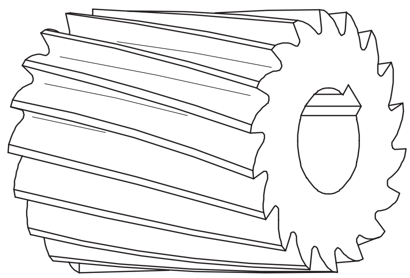
These are less than 19mm wide usually have straight teeth. (Fig 3) Those over 19mm wide have a helix angle of about 25°. (Fig 4)

Fig 3



CN20N122513

Fig 4



CN20N122514

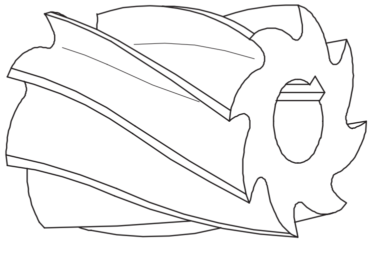
This type of cutter is used only for light milling operations since it has too many teeth to permit the chip clearance required for heavier cuts.

Heavy duty plain milling cutters (Fig 5)

These cutters have fewer teeth than the light duty ones, which provides for better chip clearance. The helix angle upto 45°.

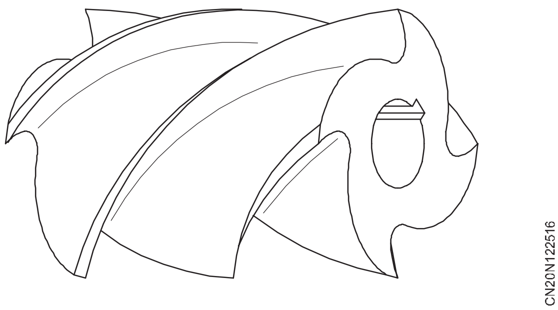
The greater helix angle on the teeth produces a smoother surface due to shearing action and reduces chatter. Less power is required for the cutter than what the straight tooth and small helix angle cutters require.

Fig 5



Helical plain milling cutters (Fig 6)

Fig 6



These cutters are high helix cutters with the helix angles from 45° to over 60°. They are particularly suited to this milling of wide and intermittent surfaces in contour and profile milling. These cutters are used for milling soft steels, brass, etc.

Plain milling cutters are also made in shank type. These are sometimes nicked on their periphery on a helical pattern for chip breaking and smooth operation.

Specification

The size of the plain milling cutter is specified by the outside diameter, length and the bore size.

Example

q 50 x 100 x 27 bore, 45°

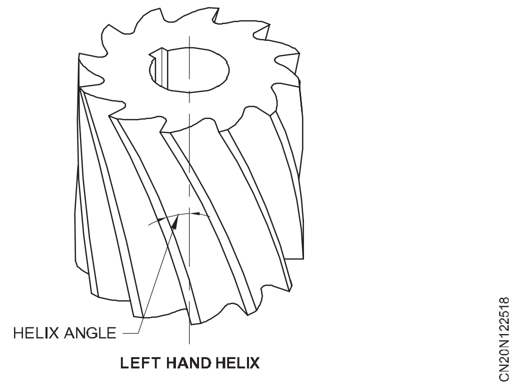
Direction of helix of the cutter

The teeth (cutting edge) of a cutter may be either straight or follow a helix.

If the cutter axis is hold vertically and the helix is towards the right side it is called a right hand helix cutter. (Fig 7) and if the helix is towards the left side, it is called a left hand helix cutter (Fig 8)

Fig 7

Fig 8



The helix angle generates a force directed along the cutter axis during cutting and a reaction to this force in the workpiece.

When a cutter has a helix and a cut of the same hand, this force will pull the cutter away from the spindle. (Fig 9) when the helix and cut are of opposite hands, the force will press the cutter into the spindle. (Fig 10) As a consequence, cutters having a helix and a cut of the same hand can only be safely used when they are positively attached to the spindle. The frictional hold of a taper is inadequate in this situation.

Fig 9

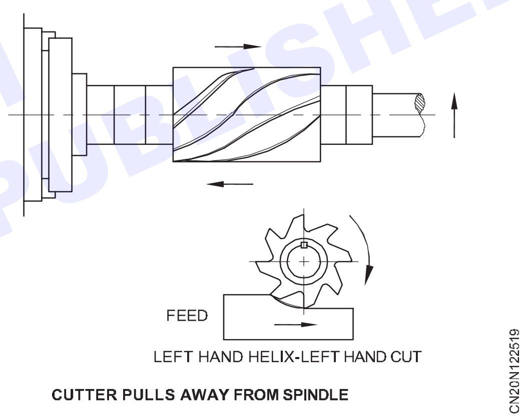
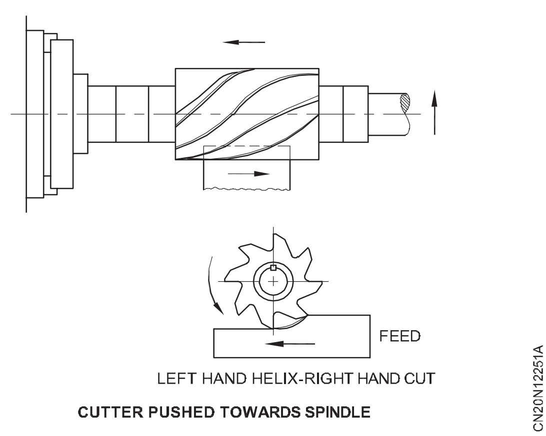


Fig 10



When mounting a cutter on the arbor of a milling machine, it is particularly important that the hands of the cut and the helix are checked.

Side and face cutters

Objectives: At the end of this lesson you shall be able to
- **state the different types of side and face milling cutters and their uses.**

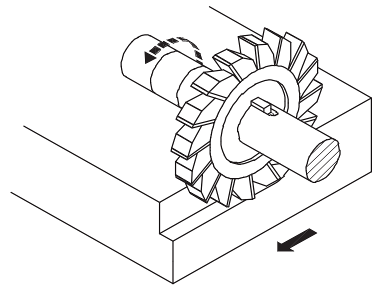
These cutters differ from plain milling cutters due to the fact that they have teeth on the periphery and face.

These cutters are mainly used for step milling, slot milling and straddle milling. These cutters are available from 50 to 100 mm in diameter and the width of the cutters ranges from 5 to 32 mm.

Types of side and face milling cutters

Half side milling cutter (Fig 1)

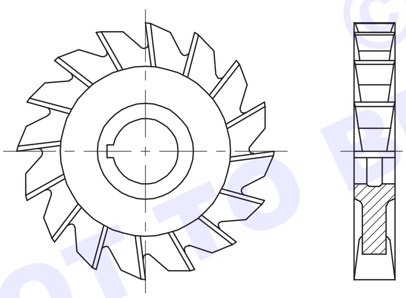
Fig 1



Cutters with teeth on one side only are called half side milling cutters and are used for heavy straddle milling, and for machining one side only.

Plain side and face milling cutter (Fig 2)

Fig 2



Inside and face milling cutters with teeth on both the sides are known as plain side and face milling cutters and are used for slot cutting (Fig 3) and face milling. These cutters are also used for straddle milling. (Fig 4).

Fig 3

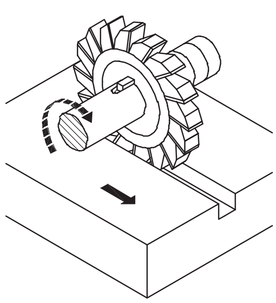
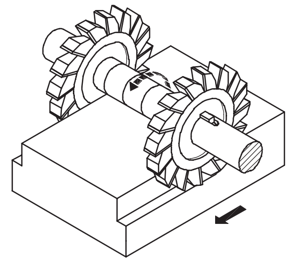


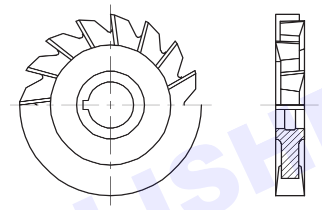
Fig 4



Staggered teeth side milling cutter (Fig 5)

These cutters have alternate teeth with opposite helix angles. Due to this design, the chip space increases to a great extent. These cutters are used for milling deep and narrow slots or keyways.

Fig 5

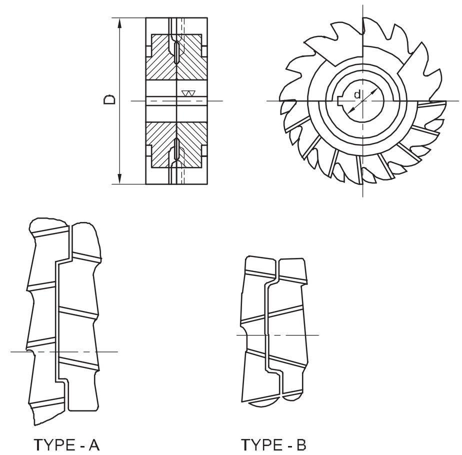


Interlocking side milling cutter (Fig 6)

This cutter is formed out of two half side milling cutters or two staggered teeth side milling cutters. They are made to interlock to form one unit. The teeth of the two cutters may be plain or of alternate helix. The cutters are used for milling wider slots of accurate width. The width of the cutter can be varied by inserting spacers between the two halves of the cutter.

The width of the cutter ranges from 10 to 32 mm with the diameters ranging from 50 to 200 mm. The width of the cutter may be adjusted to the max/min of 4 mm. The interlocking cutters can be adjusted to compensate for the wear and get sharpened as well.

Fig 6



End mill cutters

Objectives: At the end of this lesson you shall be able to
- **list the different types of end mill cutters and their uses**
- **state the application of slot mill cutters**
- **state the different types of methods of holding end mill cutters**
- **explain the influence of down-milling and up-milling in end mill cutters.**

End mill cutters

End mill cutters have the cutting teeth on the end as well as on the periphery, and are fitted to the spindle by a suitable adapter.

The end mill is used for milling small faces, slots, (Fig 1) for milling profiles (Fig 2) and milling recesses. (Fig 3) Some end mills have indexable inserts which can be replaced when worn out. (Fig 4)

Fig 1

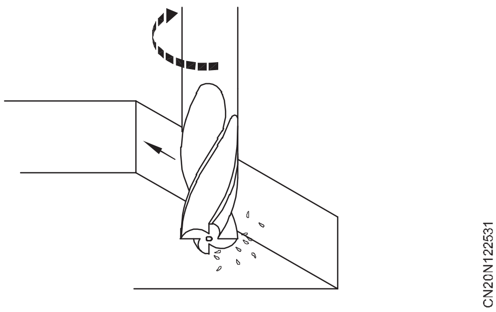


Fig 2

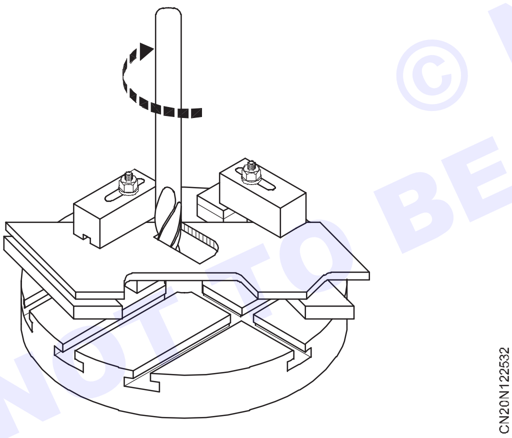


Fig 3

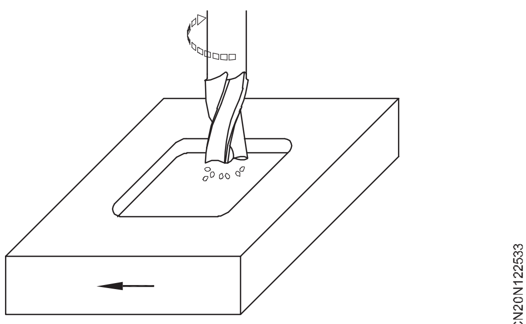
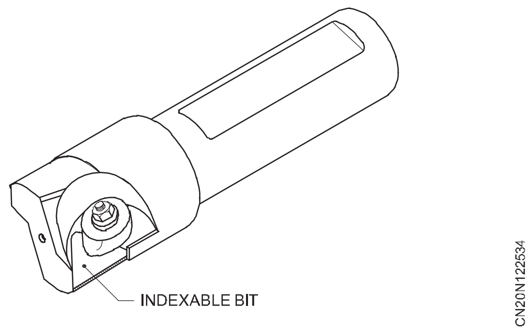
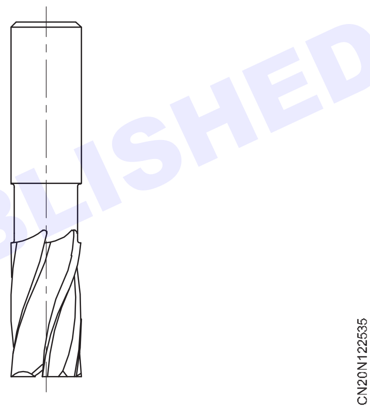


Fig 4



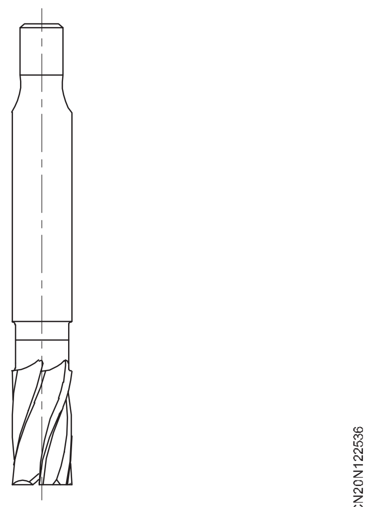
End mill cutters are solid type of cutters in which the shank and the cutters are integral. (Fig 5)

Fig 5



End mill cutters have straight shank (Fig 5) or taper shank. (Fig 6)

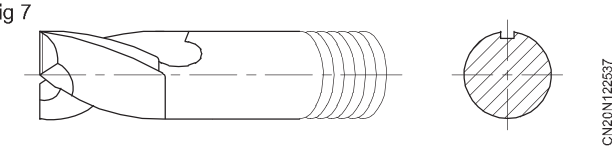
Fig 6



Slot drills

The two-flute type (Fig 7) is called a slot drill. The slot drills have flutes which meet at the cutting end, forming two cutting tips across the bottom. These tips are of different lengths, one extending beyond the central axis of the cutter. This permits the slot drill to be used in a milling machine for drilling a hole to start a slot that does not extend to the edge of the metal. It is used for plunge milling like keyways etc.

Fig 7



Angular and slitting saw milling cutter

Objectives: At the end of this lesson you shall be able to
- **state the different types of angular milling cutters and their uses**
- **state the specification of angular milling cutters**
- **explain slitting saw cutters and their uses**
- **state the specification of a slitting saw cutter.**

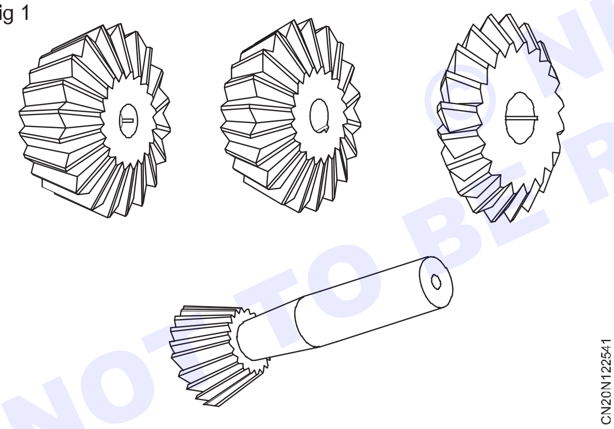
Angular milling cutters

These cutters have teeth on the periphery, and the cutting edges are placed on a conical surface.

Angular cutters have teeth that are neither parallel nor perpendicular to the cutting axis.

Angular milling cutters are made with a hole for use in horizontal milling, or with a shank for use in both horizontal and vertical milling. (Fig 1)

Fig 1



They may be divided into two groups.

Single angle milling cutters

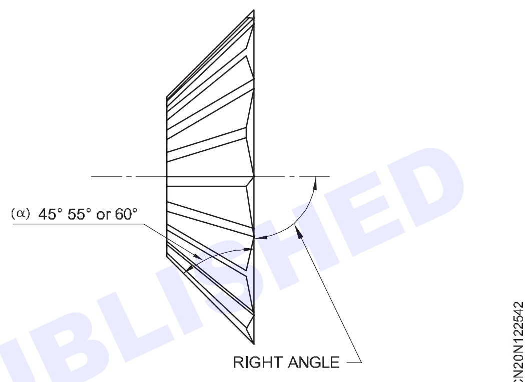
These cutters have teeth on the angular surface, and may or may not have teeth on the flat side. The included angle between the flat face and the angular face designates the cutters, such as 45° or 60° angular cutter. (Fig 2) They may be of the shell or shank type.

Specification

A shell end single angle cutter of diameter D = 80, angle μ= 50° of 'tool type' H and for right hand cutting shall be specified and designated as

Shell end single angle milling cutter 80 x 50°H IS:6256.

Fig 2



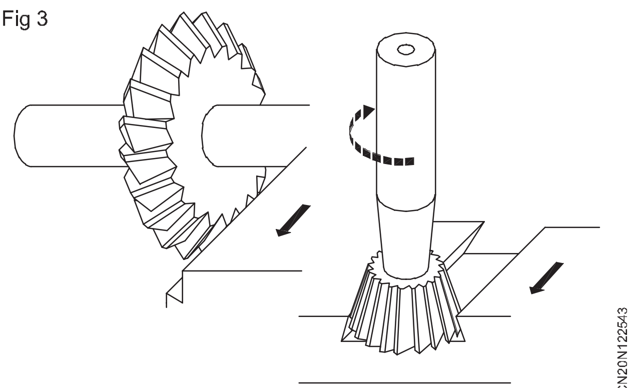
A dovetail milling cutter type A having diameter D = 20 mm, angle μ = 60° of tool type 'N' for right hand cutting shall be specified as

Dovetail milling cutter A20 x 60°N BIS 6255.

In type 'A', the small end is having less diameter and in type 'B' the small end is having more diameter than in type A.

Single angle cutters are used to dovetail guideways etc. (Fig 3)

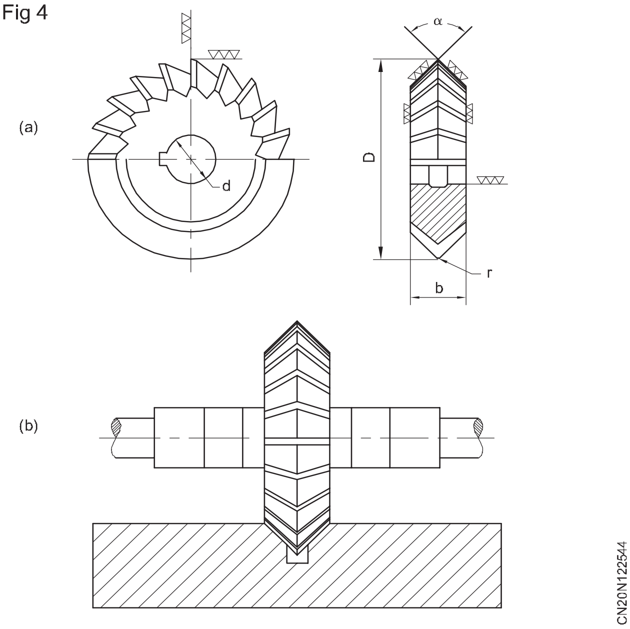
Fig 3



Equal angle milling cutters

These cutters have two intersecting angular surfaces with cutting teeth on both sides. When these cutters have equal angles on both sides of the line at right angle to the axis (symmetrical), they are designated as per the size of the included angle such as 45°, 60° or 90°. Double angle cutters have two cutting edges. (Fig 4a)

Fig 4



Specification

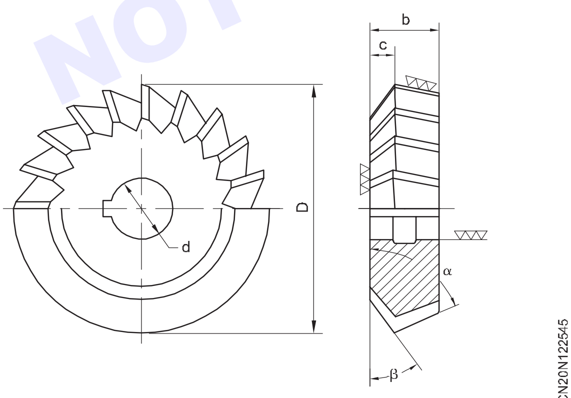
An equal angle milling cutter of diameter D = 56 mm for angle 60° of 'tool type' N shall be specified as Equal angle milling cutter 56 x 60° N IS 6326.

It is used to machine Vee slots. (Fig 4b)

Double unequal angle cutter

When the angles formed are not the same (unsymmetrical), the cutters are designated by specifying the angle on both sides of the plane or line. (Fig 5)

Fig 5



These cutters are generally used for milling the flutes on taps or reamers. The cutters are marked with the type of taps or reamers for which they should be used.

Specification

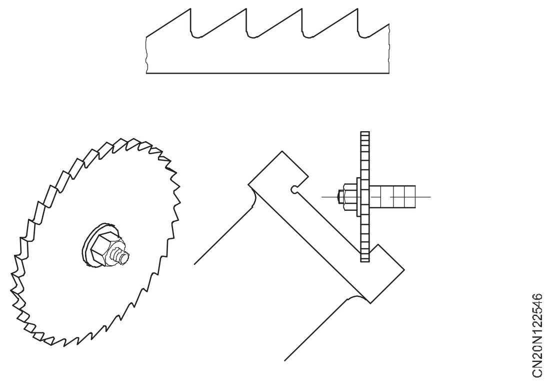
A double angle milling cutter of diameter D = 50 mm, b = 12 mm and angle 75° of 'tool type H and for right hand cutting shall be specification as

Double angle milling cutter 50 x 12 x 75°H IS 6325.

Slitting saw

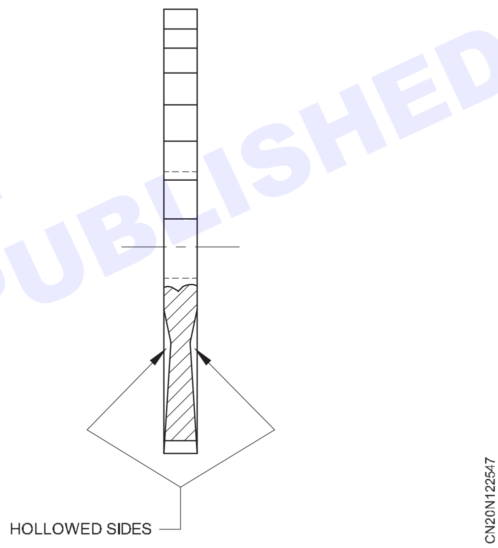
It is basically a thin plain milling cutter. It has a large number of teeth. (Fig 6)

Fig 6



In order to prevent the sides of the saw from rubbing or binding when in use, the sides are relieved or dished. (Fig 7)

Fig 7



Slitting saws are made in widths of 3 to 6 mm. Because of the thin cross-section, they should be operated at approximately one quarter to one eighth of the feed per tooth used for the other cutters. For non-ferrous metals, these speeds can be increased. Unless a special driving flange is used for slitting saws, it is not advisable to key the saw to the milling arbor.

The arbor nut should be pulled up as tightly as possible by hand only. Since slitting saws are so easily broken, some operators find it desirable to adopt climb or down-mill method when sawing. However, to overcome the play between the lead screw and nut, the backlash eliminator should be engaged.

A slitting saw is specified by its outside diameter, bore diameter and thickness.

Example: 150 x 6 x 27 mm bore

Form milling cutters

Objectives: At the end of this lesson you shall be able to
- **state the name of different types form milling cutters and their uses**
- **state the specification of form milling cutters.**

Form milling cutters have irregular profile on the cutting edges in order to generate an irregular outline of the work. They are normally solid but, sometimes, may have inserted teeth. Different types of standard form cutters are described below.

Convex milling cutter

These cutters have their teeth curved outwards on the circumferential surface to form the contour of a semi- circle. Concave semi-circular surfaces are produced with this cutter. (Fig 1) The diameter of the cutter ranges from 50 to 125 mm and the radius of the semi-circle ranges from 1.6 to 20.0 mm. (Fig 2)

Fig 1

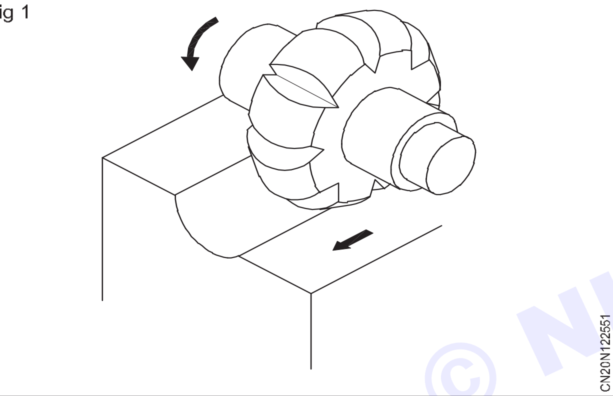
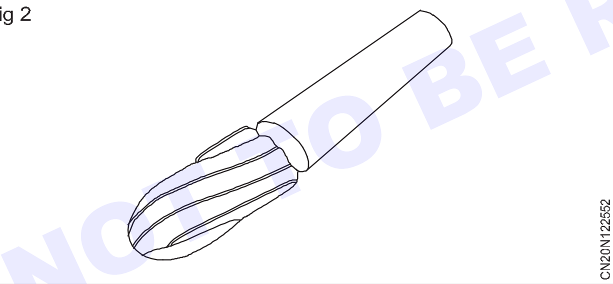


Fig 2

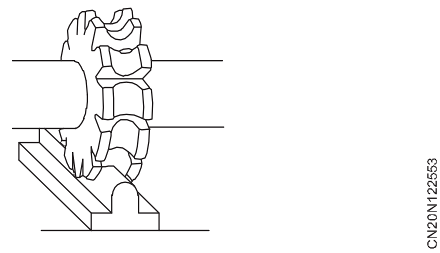


Concave milling cutter

These cutters have their teeth curved inwards on the circumferential surface to form the contour of a semi- circle. Convex semicircular surfaces are produced with this cutter. (Fig 3).

The diameter of the cutter ranges from 56 to 110 mm and the radius of the semi-circle ranges from 1.5 to 20.0 mm.

Fig 3



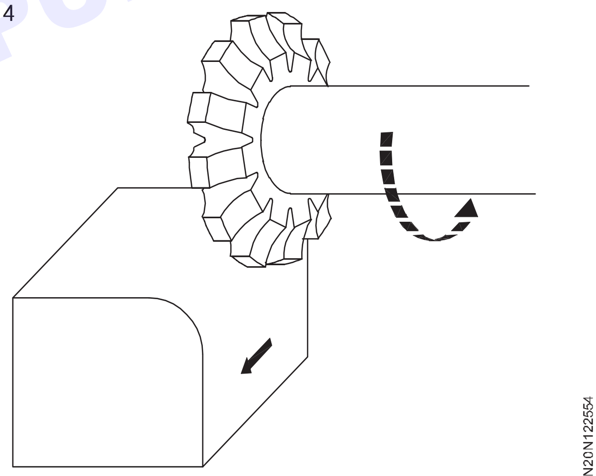
Corner rounding milling cutter

The corner rounding milling cutters have their teeth curved inwards on the circumferential surface to form the contour of a quarter circle. A convex surface is produced with this cutter. This cutter is used for cutting a radius on the corner or edges. It may be of either the shank or arbor type.

Corner rounding cutters are available with their teeth having placed on one side or both the sides. (Fig 4)

The cutters are specified by the type, diameter width, radius of the form and bore size.

Fig 4



'T' slots cutters

Objectives: At the end of this lesson you shall be able to
- **state the purpose of 'T' slot milling cutters**
- **state the specification of the 'T' slot milling cutters.**

'T' slot cutters: These cutters are profile-sharpened side milling cutters with a straight or taper shank. (Fig 1) They have staggered teeth and are either solid or tipped in construction. Due to the staggered teeth the chips are cleared without clogging.

The 'T' slot cutter is used to cut 'T' shaped slots in machine tool work tables. (Fig 2) Before cutting the 'T' slot, a narrow vertical groove is machined with an end mill or a slot milling cutter.

Fig 1

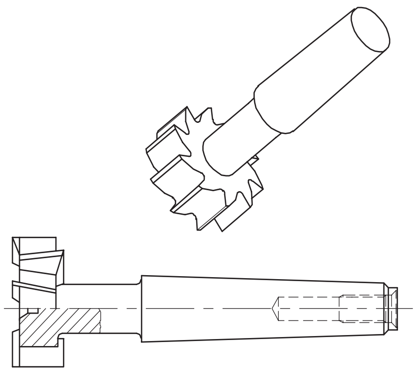
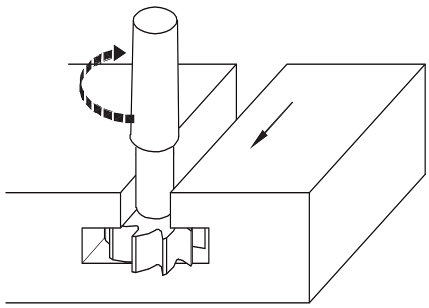
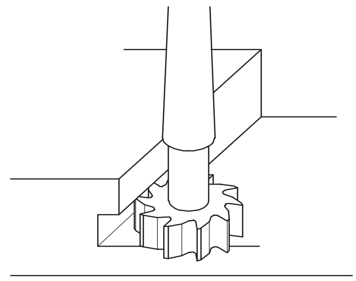


Fig 2



This cutter can also be used to mill undercuts in wider milled channels. (Fig 3)

Fig 3



Specification: A 'T' slot milling cutter with plain parallel shank for milling a 'T' slot of nominal size 12, tool type N, for right hand cutting is designated as plain parallel shank 'T' slot cutter 12 BIS:2668. When the cutter is required with a tool type other than N, an appropriate tool type H or S shall be added to the designation after the size.

Tool type

N - for mild steel, soft cast iron and medium hard non-ferrous metals.

H - for especially hard and tough metals.

S - for soft and ductile material.

A 'T' slot milling cutter with Morse taper shank with tapped hole for milling a 'T' slot of nominal size 18, tool type N, for right hand cutting is designated as taper shank 'T' slot cutter 18 BIS:2668. When the cutter is required with a tool type other than N, an appropriate tool type H or S shall be added in the designation immediately after the size.

Example: 16 N BIS 2668

Special milling cutters

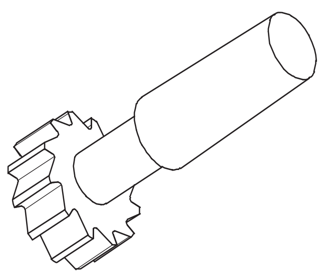
Objectives: At the end of this lesson you shall be able to
- **state the woodruff keyway cutter, thread cutter, gear cutter, tap and reamer cutter and state their uses**
- **state the sprocket cutter, spline cutter, fly cutter and bolted cutter and state their uses.**

Many types and sizes of cutters are available. The selection of an appropriate cutter for a particular type of operation is very important.

Woodruff key cutters

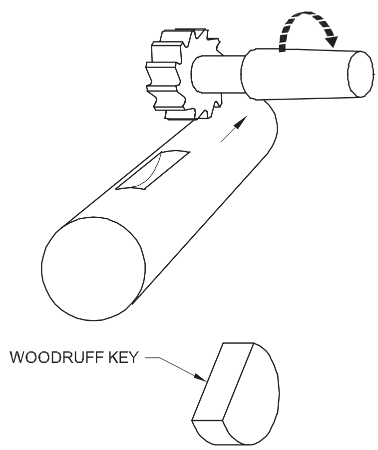
These are similar in appearance to 'T' slot cutters. These cutters have cutting edges on the periphery only. There are no side teeth. (Fig 1)

Fig 1



They are used for milling semi-cylindrical keyways in shafts for woodruff key seats. (Fig 2)

Fig 2



This cutter is provided with a shank which may be parallel or with Morse taper. These cutters are either solid or tapered in construction. They can either be of the arbor or shank type.

This cutter may have straight or staggered teeth.

Specification

A woodruff key slot milling cutter, type A of diameter d = 16.5 mm and width b = 5 mm and tool type 'N' for right hand cutting shall be specified as woodruff slot milling cutter A 16.5 x 5 N BIS2669.

When the cutter is required for left hand cutting, the letter 'L' shall be added before the size in the designation.

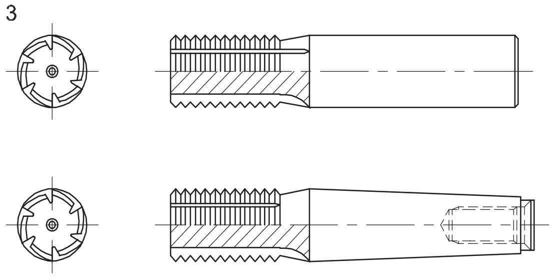
Type 'A' is for straight teeth and type B for staggered teeth.

Thread milling cutter (Fig 3)

For milling the threads of specific form and size, thread milling cutters are used. Generally, acme and worm-threads are produced on the workpiece by thread milling cutters.

Both parallel shank and taper shank thread milling cutters are available.

Fig 3



Gear cutter (Fig 4)

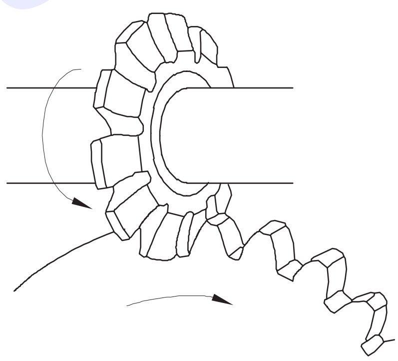
These cutters are used to machine gear teeth by milling.

These cutters having formed cutting edges reproduce the shape of the cutter teeth on the gear blank. According to the gear teeth profile the shape of the cutter teeth may be involute or cycloid. These cutters are available in a very wide range of sizes covering the various sizes of gear tooth.

Example:

Involute cutter, 3mm module, 27 mm bore, cutter No.5

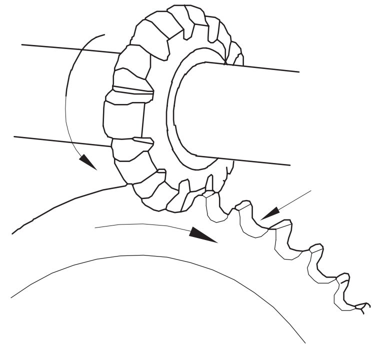
Fig 4



Sprocket cutters (Fig 5)

These cutters are designed to cut the teeth of sprocket wheels which are used in chain drives, such as those found on bicycles and on machinery in general.

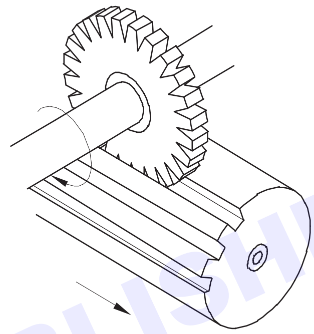
Fig 5



Spline cutters (Fig 6)

These cutters are used to cut splines. They are marked with the type and size of the spline that they should be used for.

Fig 6

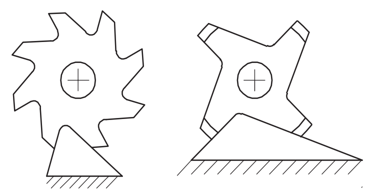


Tap and reamer cutter (Fig 7)

These cutters are used for producing grooves or flutes in taps and reamers. These are the special type of double angle cutters.

The point end of the tool is rounded and the tooth profile corresponds to the type of groove that it is to produce.

Fig 7



Fly cutters (Fig 8)

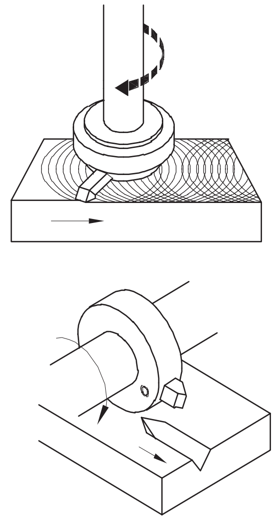
Fly cutters are single point tools having only one cutting edge. These tools are held in various types of holders.

These cutters are used to machine shapes which cannot be produced using standard milling cutters.

These are also used to mill flat surfaces which are truly flat to a very high standard of accuracy.

This cutter is used in tool room and in emergency when standard cutters are not available.

Fig 8



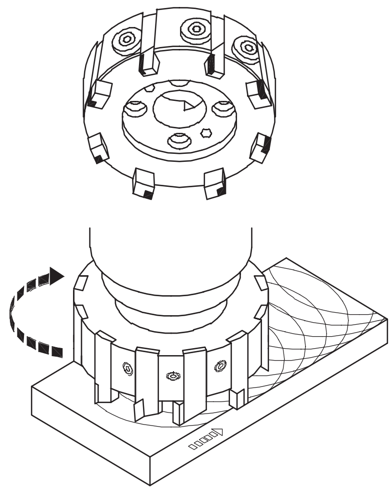
Bolted cutters (Fig 9)

The face milling cutters having no shank but one of a larger diameter and they are bolted directly on to the nose of the spindle.

This cutter is used for face milling on large areas of the workpiece. For utmost rigidity, this system is used.

The cutter has a body with slots and fixing devices for the cutting teeth which are fixed into the body. The cutting teeth, which may be made of high speed steel or carbide, can easily be replaced when worn out.

Fig 9



Cutter nomenclature

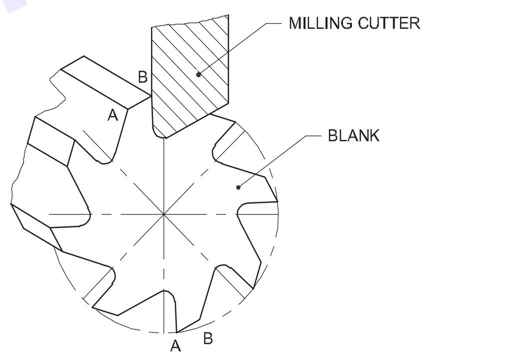
Objectives: At the end of this lesson you shall be able to
- **state the elements of a milling cutter**
- **state the influence of rake angles in machining.**

Milling cutter are multiple point cutting tools. They are made from cylindrical blanks with the teeth formed by milling the chip space. (Fig 1) The number of teeth so milled depends on the diameter of the cutter as well as on the type of operation, namely roughing and finishing. A roughing cutter will have less number of teeth as compared to a finishing cutter of the same diameter.

Angle of a milling cutter

A milling cutter tooth is more or less identical to that of a single point tool.

Fig 1



Milling cutter holding devices

Objectives: At the end of this lesson you shall be able to
- **state the different types of arbors**
- **state the uses of arbors**
- **name the parts of an arbor**
- **specify the arbor**
- **brief the different methods of holding end mills.**

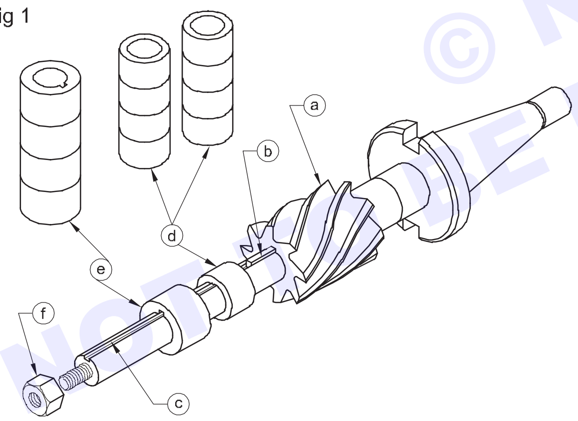
Types of arbors and their uses

An arbor is considered as an extension of the machine spindle on which milling cutters are mounted. Arbors are provided with quick-release taper shanks for proper alignment with the spindle. There are two types of arbors, normally used for holding the cutters. They are (1) long arbor and (2) short or stub arbor.

Long arbor (Fig 1)

Long arbors are used for holding cutters in both horizontal and universal milling machines. The milling cutter (a) is driven by a key (b) which fits into the keyway (c) on the arbor and cutter. This prevents the cutter from turning on the arbor. The spacer (d) and bearing bushings (e) hold the cutter in position on the arbor after the nut (f) has been tightened.

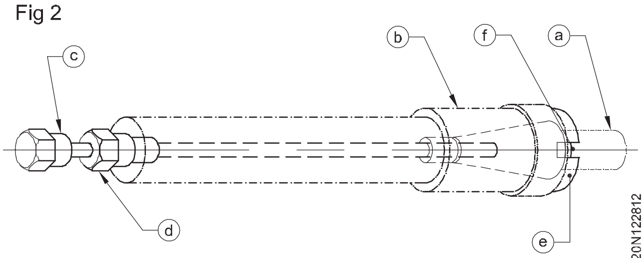
Fig 1



The tapered end of the arbor (a) is held securely with the machine spindle (b) by a draw-in bar (c) and lock-nut (d). (Fig 2) The flange (e) has two notches (f) to engage with the spindle tenon for transmitting the power.

The outer end of the arbor assembly is supported by the bushing and the arbor support.

Fig 2



Long arbors with I.S.O. taper shanks are available in different diameters. The normal diameters used commonly are Ø16, Ø22, Ø27, Ø32, Ø40 and the taper is ISO40/50.

The arbor is designated by the taper number, diameter and length.

Example ISO40 x Ø 22 x 500 mm.

ISO40 = Type of taper

Ø 22 = diameter of the arbor in 'mm'

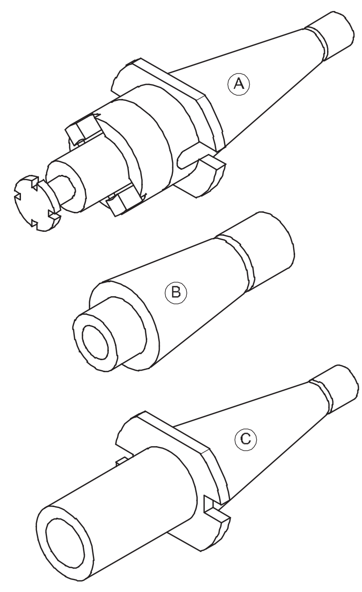
500mm = length of the arbor

Stub arbor

Stub arbors are used to mount various tapers of cutters in the spindle of horizontal and vertical milling machines.

The arbors are held with the machine spindle by a taper and a draw-in bar. The arbors are of three types (A), (B) & (C) as shown in Fig 3.

Fig 3

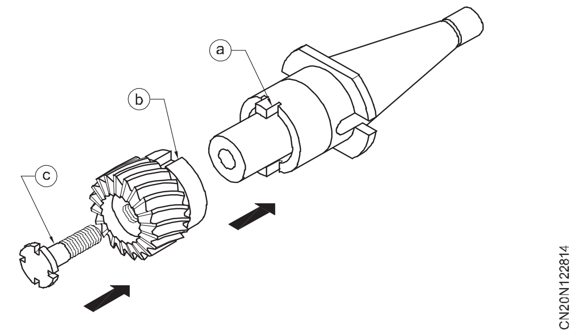


Types

Type A (Fig 4) is used to mount the shell end mills and similar cutters.

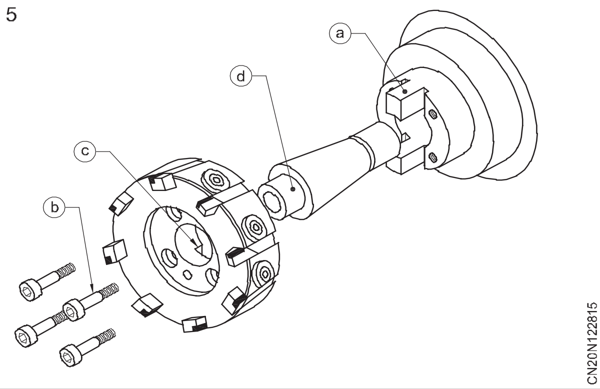
The cutter is pushed on the arbor so that the arbor key (a) fits with the slot (b) on the cutter. The cutter is tightened on the arbor using the screw (c).

Fig 4



Type B (Fig 5) is used to mount large face milling cutters. It is made with a centralizing spigot (a) to ensure that the cutter is centralized with the cutter spindle.

Fig 5



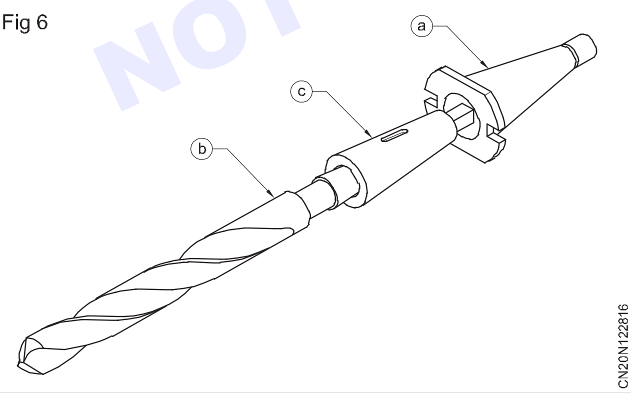
The cutter is held on the arbor by four screws (b). It has a slot (c) which fits over the spindle (d) to provide the drive.

Type C (Fig 6) is a Morse taper adapter arbor (a). It is used to hold drills, reamers, chucks (b), etc. which have taper shanks and also Morse taper sleeves (c) which are used to adapt a Morse taper to a larger taper.

According to B.I.S. specifications stub arbors with Morse taper shanks are available from 13 to 27 mm in diameters.

The stub arbor is designated by the taper number, diameter and length.

Fig 6



Collet chuck (Fig 7)

The chuck is supplied with a set of spring collets (Fig 8) in various sizes to suit the shanks of standard parallel shank cutters.

The chuck body Fig 9(a) is mounted in the machine spindle nose in the same way as the horizontal milling arbor.

Fig 7

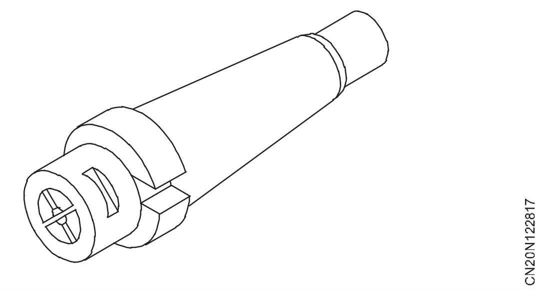


Fig 8

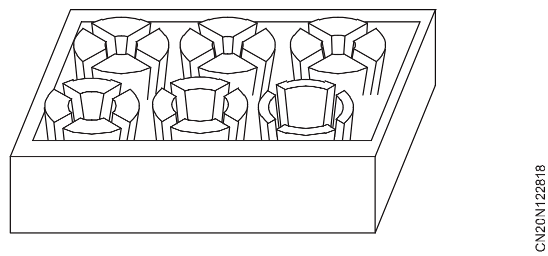
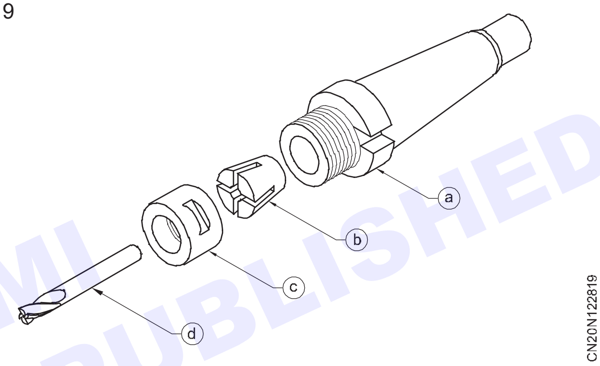


Fig 9



The collet (b) (of the same size as that of the cutter shank) is pushed in to the chuck body. The nut (c) is screwed on until it just grips the collet.

The cutter (d) is inserted into the collet and then the nut is tightened using the special spanner supplied with the chuck.

Adapters

These are used to reduce the internal taper in the work spindle, so that it fits on required arbor or cutter. (Fig 10) This type is used for holding the cutters with internal thread. Another type of adapters (shown in Fig 11) with Morse taper and flat tongs is used for holding taper shank end mills with tongs.

Fig 10

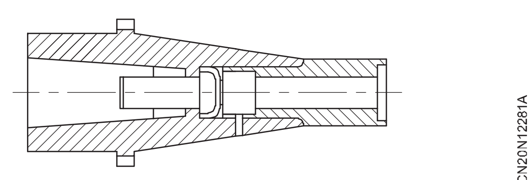
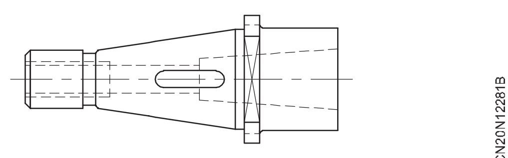


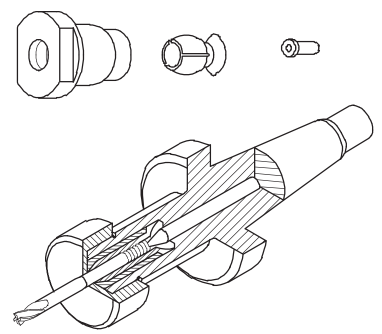
Fig 11



Self-locking chucks (Heavy duty chucks)

The cutter is provided with fine pitch thread at the end of the cutter shank. The cutter is mounted by turning with the thread provided into the chuck body and clamping. (Fig 12)

Fig 12



The common methods used to mount end mills in vertical milling machine are shown in Figs 13 to 16.

Fig 13

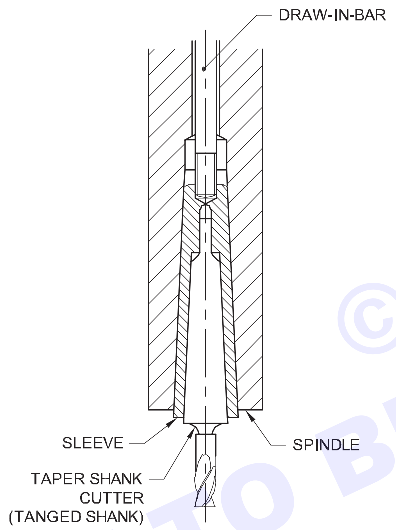


Fig 14

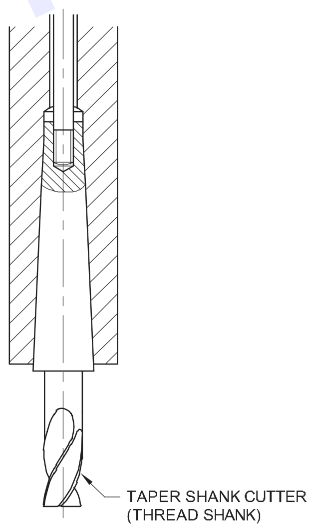


Fig 15

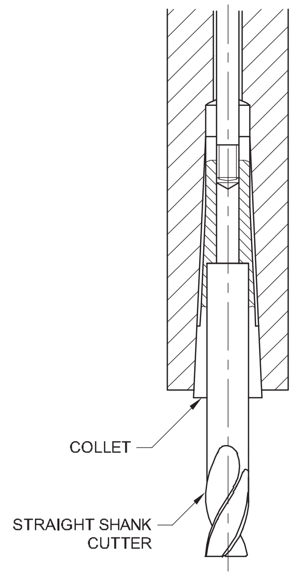
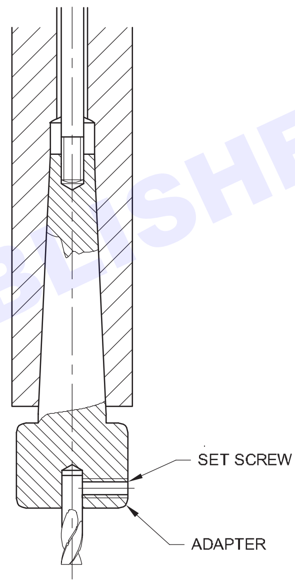


Fig 16



Cutting speed, feed and machining time calculation

Objectives: At the end of this lesson you shall be able to
- **calculate the revolution per minute for various cutters and materials**
- **select and calculate proper feeds for various cutters and materials**
- **explain the correct procedure for taking roughing and finishing cuts.**

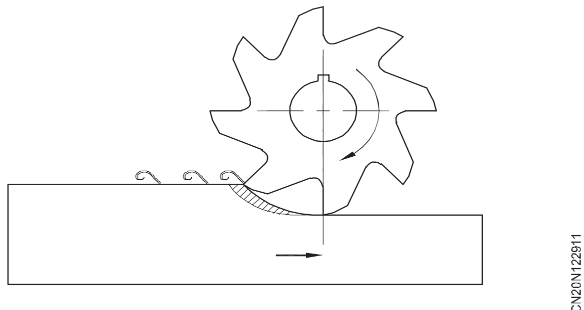
Cutting speeds, feeds and depth of cut: The efficiency of a milling operation depends upon the cutting speed, feed, and depth of cut.

If the cutter is run too slowly valuable time will be wasted, while excessive speed results in loss of time in replacing and regrinding cutters. Somewhere between these two extremes is the efficient cutting speed for the material being machined.

The rate at which the work is fed into the revolving cutters is important. If the work is fed too slowly time will be wasted and chatter may occur which shortens the tool life of the cutter. If the work is fed too fast, the cutter teeth can be broken. Much time will be wasted if several shallow cuts are taken instead of one deep or roughing cut. Therefore, speed, feed and depth of cut are the three important factors in any milling operation.

Cutting speed: The cutting speed for a milling cutter is the speed at which the cutting edge or tooth cuts into the workpiece. (Fig 1)

Fig 1



It is expressed in metres per minute.

The following important factors must be considered when determining the proper revolutions per minute at which to machine a metal.

- Type of work material
- Cutter material
- Diameter of the cutter
- Surface finish required
- Depth of cut being taken
- Rigidity of the machine and work set up

Since different types of metals vary in hardness, structure and machine ability, different cutting speeds must be used for each type of metal and for various cutter materials.

The cutting speeds for the more common metals for HSS milling cutters are shown in Table 1.

Table 1

Table for selecting cutting speeds for high speed steel milling cutters

Material to be machined	BHN hardness	Shell and Mill	End mill	S& F cutter	Cylind. cutter	Slot's cutter	Form cutter	In.tooth face mill
Mild steel	150	20-30	20-30	15-25	15-25	15-25	30-45	20-30
Medium carbon steel	200	15-25	15-20	15-20	20-30	15-20	15-25	15-25
High carbon steel	300	10-15	10-15	10-15	12-20	10-15	13-20	13-20
Stainless steel	200	22-30	22-30	15-25	15-25	20-30	15-25	20-30
Malleable iron	160	15-22	15-22	15-20	15-20	20-30	15-20	18-25
Soft cast iron	180 max	15-20	15-25	15-20	15-20	20-30	15-20	15-25
Hard cast iron	Over 180	13-17	10-15	10-15	10-15	10-25	10-15	13-17
Hard brass & hard bronze	-	40-60	40-60	30-45	30-45	70-90	30-45	50-60
Soft brass & soft bronze	-	40-60	40-60	25-35	25-35	70-90	25-35	40-50
Copper	-	30-45	30-45	30-45	30-45	70-90	25-35	50-60
Aluminium alloy	-	200-300	200-300	150-300	150-300	200-300	150-250	200-400

Carbide cutters are able to cut at a much higher speed than HSS cutters and they are made in a variety of grades. If you are going to use a carbide cutter, ask your instructor what cutting speed you should select, as he will have the values for the particular grade of carbide used in the cutter in your workshop.

Calculation

$$\text{Cutting speed}(V) = \frac{\pi DN}{1000} \text{m/min}$$

$$N(\text{rpm}) = \frac{V \times 1000}{3.1416 \times D}$$

Since only a few machines are equipped with a variable speed drive which allows them to be set to the exact calculated speed, a simplified formula can be used to calculate the revolution per minute.

The 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.

$$N(\text{r.p.m}) = \frac{V(m) \times 320}{D(mm)}$$

where 'D' is diameter of the cutter.

Example

Calculate the revolution per minute required for ϕ75 mm high speed steel cutter when cutting machine steel. (V = 30 m/min.)

$$\text{r.p.m} = \frac{30 \times 320}{75} = \frac{9600}{75} = 128$$

From the Table 2 the intersection of ∅ 75mm and cutter speed of 30 m/min. is in between 115 and 140 r.p.m. This can be taken as 128 r.p.m. as calculated.

Too fast a speed will shorten the cutter tool life; too slow a speed will waste time.

Milling feeds and depth of cut

The two other factors which affect the efficiency of a milling operation are the milling FEED or the rate at which the work is fed into the milling cutter and the DEPTH of CUT taken at each pass.

Feed

Feed is the rate at which the work moves into the revolving cutter. It is measured in milli metres per minute (mm/min.)

Feed rate is specified in mm/min.

The feed is expressed in milling machines by following three different methods.

Feed per tooth

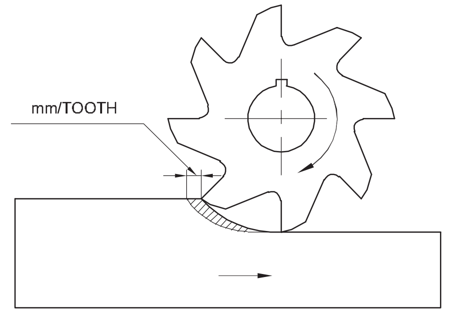
Feed per tooth is defined by the distance the work advances and the time between engagement by two successive teeth. It is expressed in mm/tooth of the cutter. (Fig 2)

Feed per cutter revolution

Feed per cutter revolution is the distance the work advances in the time when the cutter runs through one complete revolution. It is expressed in mm/revolution of the cutter.

Feed per minute

Fig 2



Feed per minute is defined by the distance the work advances in one minute. It is expressed in mm/ minute.

The rate of feed has an effect on the life of the cutter. An increase in feed, using the same cutting speed and depth of cut will reduce the amount of wear of the cutter.

In general, we can say that the

- cutting speed should be reduced when feed is increased (Fig 3)
- cutting speed should be increased when feed is reduced. (Fig 4)

Fig 3

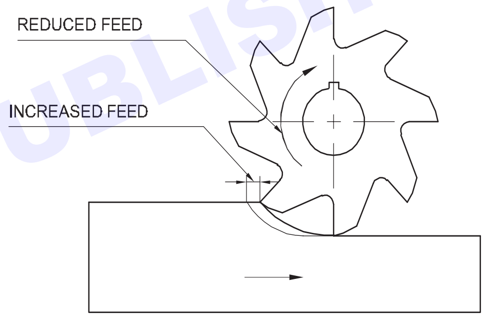
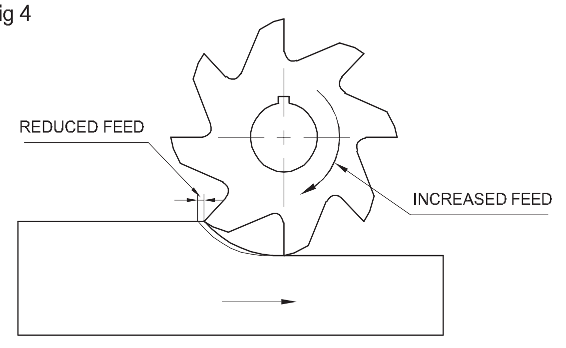


Fig 4



The feed rate on a milling machine depends on a variety of factors such as

- width and depth of cut
- type of cutter
- sharpness of the cutter
- workpiece material

- strength and uniformity of the workpiece
- type of finish and accuracy required
- power and rigidity of the machine.

Calculation

The formula used to find the work feed is

feed mm/min.(S) = N x Cpt x r.p.m.

where N = number of teeth in milling cutter

Cpt = chip per tooth for a particular cutter

r.p.m. = revolution per minute of the milling cutter.

Example 1

Calculate the feed in mm/min. for a ∅75, six-teeth helical carbide milling cutter when machining a cast iron workpiece (V = 60 and Cpt = 0.18).

First calculate the r.p.m. of the cutter

$$\text{rev/min} \quad \frac{60 \times 320}{75} = 256$$

Feed (mm/min.) = N x C.p.t x r.p.m.

= 6 x 0.18 x 256

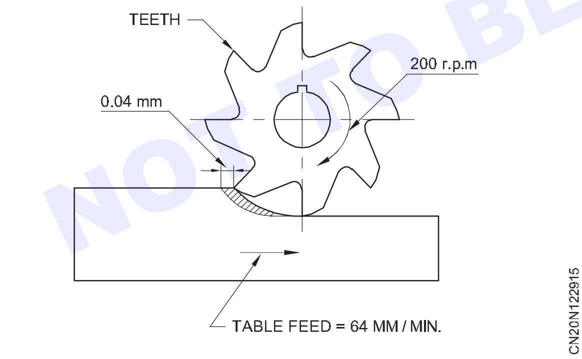
= 276.4

= 276 mm/min.

The spindle speed (revolution per minute) must always be calculated before the feed rate can be calculated.

Example 2 (Fig 5)

Fig 5



A cutter having 8 teeth is to have a feed of 0.04 mm/ tooth. The spindle speed is to be 200 r.p.m. What feed, in mm/min. should be set on the machine?

While rough milling, where the purpose is to remove surplus metal as quickly as possible and finish is not important, a heavy feed and low cutting speed are used. (Fig 6) However, the cutting speed should not be reduced too much as the cutter would then be operating under very heavy cutting forces.

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 7)

Fig 6

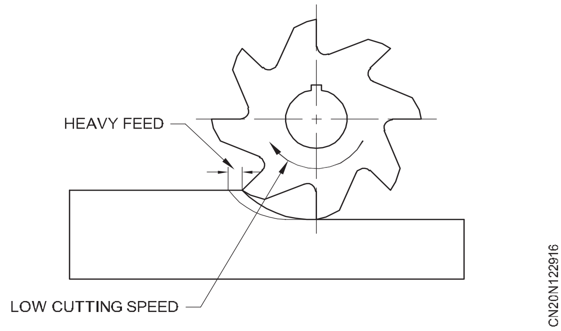
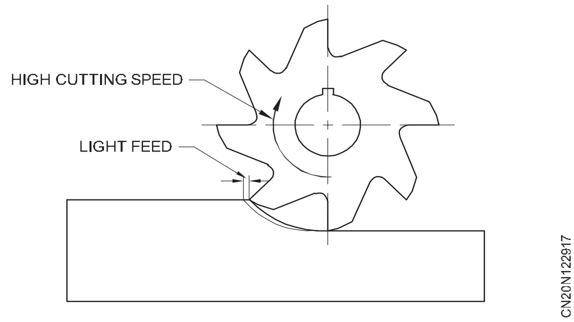


Fig 7



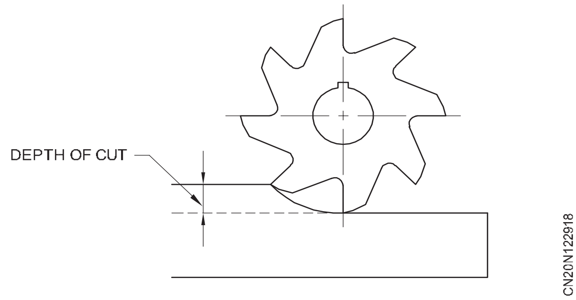
Cutting speed should be reduced when the feed is increased.

Cutting speed should be increased when the feed is reduced.

Depth of cut

The depth of cut is the depth to which the cutter penetrates the workpiece surface during a given cut. It is the perpendicular distance (Fig 8) measured between the original and the final surface of the workpiece.

Fig 8



Where a smooth and accurate finish is needed, it is a good practice to take roughing and finishing cuts. Roughing cuts should be deep with a feed as heavy as the work and machine will permit with low cutting speed. 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 is used in roughing cuts. The depth of cut should be atleast 0.4 mm. Light cuts and extremely fine feeds are not advisable, since the chip taken by each tooth will be thin and the cutter will often rub the surface of the work. When a fine finish is required, the feed should be reduced rather than the cutter speed; more cutters are dulled by high speeds than by high feeds.

Identification of defective milling cutters

Objectives: At the end of this lesson you shall be able to
- **findings the defects of milling cutters.**

Defective cutters can be identified by visual inspection and by performance.

A Visual inspection

1 Cracks and Breaks

Check on cutter body, teeth and shank.

2 Wear and Tear

Wear on the cutting edges teeth or bearing surfaces.

3 Rust or corrosion

Look out for rust or corrosion on the cutter body or shank.

4 Deformation

Deformation / bending of the cutter body or shank.

B Performance

1 Vibration

Any unusual patterns noticed on milled surfaces.

2 Noise

Unusual sound during milling operation.

3 Surface finish

Degradation noticed on milled surface finish.

Apart from the above type of defections some more technics used are like magnification of cutter through microscopes/borescopes, non-destructive test such as X-ray or ultrasonic testing for internal defects and tool life monitoring system.

Introduction to product design and development

Objectives: At the end of this lesson you shall be able to
- **understand the fundamentals of product design and development**
- **explore stages of product creation**
- **recognize the importance of collaboration.**

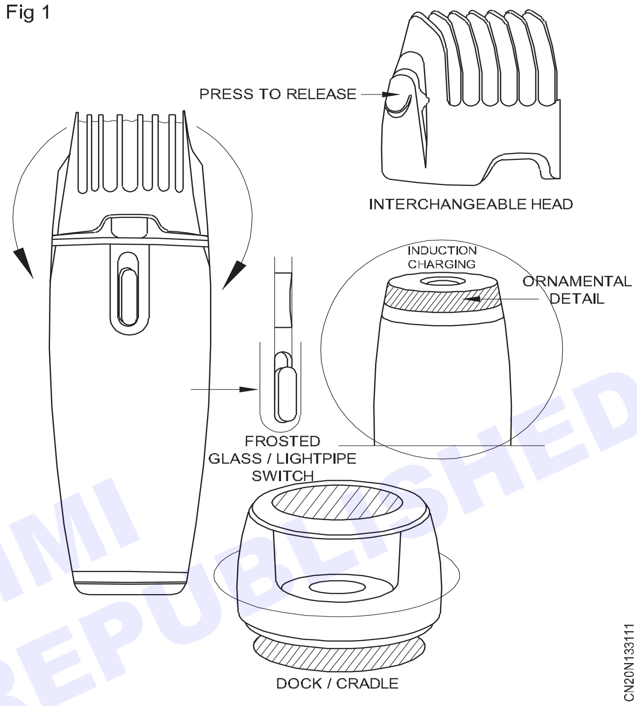
Product design

Product design and development is a multifaceted process that transforms an idea into a tangible product ready for the market. It combines creativity, engineering, and strategic planning to address user needs while meeting business objectives. The primary goal is to create innovative solutions that are functional, aesthetically appealing, and economically viable.

The product design process typically involves the following stages:

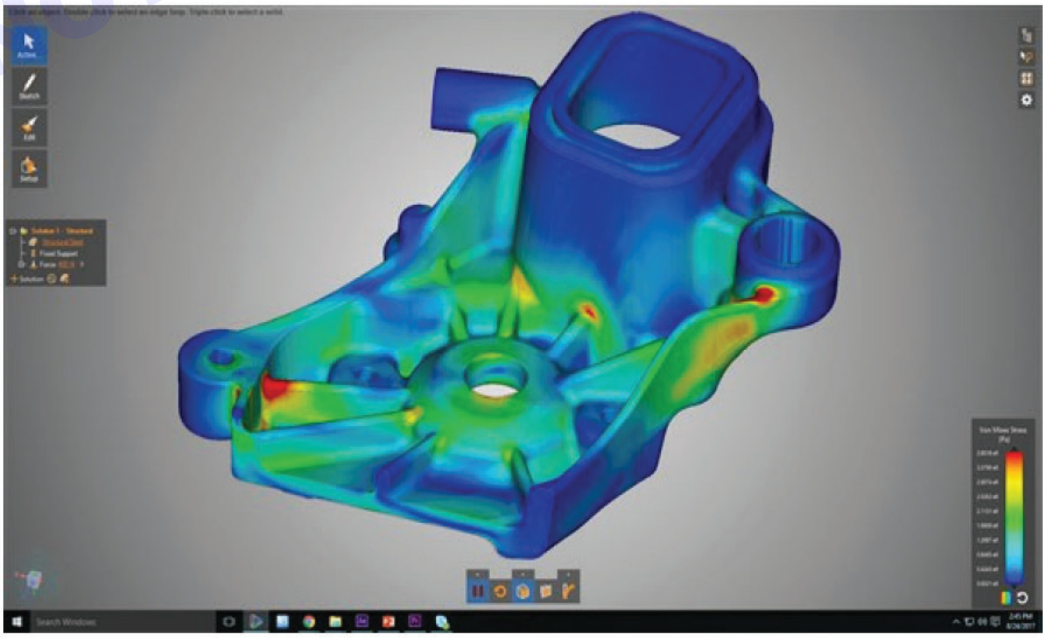
- **Defining the product vision:** Before the design process begins, the product team defines the product's vision and strategy.
- **Research:** The team conducts research to understand the market and identify gaps.
- **User analysis:** The team analyzes the user's needs, habits, behaviors, and frustrations.
- **Ideation:** The team generates ideas for the product.
- **Design:** The team translates the product features into a user experience.
- **Testing and validation:** The team tests and validates the product.
- **Post-launch activities:** The team continues to refine the customer experience and add new features after the product is launched.

Fig 1



A good product design can help a company build brand awareness and outcompete competitors. It can also lead to satisfied customers who are more likely to refer the product to others, which can boost sales and brand loyalty.

Fig 2



The journey of product design

The journey of product design begins with ideation, where concepts are generated based on comprehensive market research and user analysis. This phase emphasizes understanding the target audience's requirements and identifying gaps in the market. Once ideas are crystallized, the concept development stage refines these ideas into detailed plans, including sketches, digital models, and preliminary prototypes. This process allows designers to visualize the product and evaluate its potential feasibility.

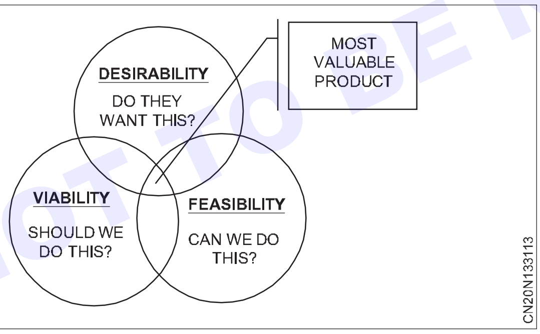
Feasibility analysis and prototyping

Feasibility analysis is a critical step where technical, financial, and market factors are assessed to determine the product's viability. This evaluation ensures that the concept aligns with both user expectations and organizational goals. Following this, prototyping brings the design to life through functional or visual models. Prototypes serve as a tool for testing, validation, and iterative improvements, ensuring that the final design meets all necessary criteria.

Scaling and market introduction

The production phase involves scaling the design for manufacturing. This stage requires meticulous planning to select suitable materials, production methods, and quality control processes. The result is a product ready for market introduction, supported by marketing strategies and distribution channels. Post-launch evaluation is equally important, as it gathers user feedback to identify areas for improvement and ensure the product's continued success.

Fig 3



Principles of successful product design

Central to successful product design is the adherence to key principles. User-centered design prioritizes the needs, preferences, and experiences of end-users. Functionality is paramount, ensuring the product performs its intended purpose effectively. Aesthetics play a significant role in creating visually appealing designs that attract and resonate with users. Sustainability has become increasingly crucial, with designers focusing on eco-friendly practices, such as using recyclable materials and minimizing waste. Scalability ensures that products can be efficiently produced and distributed as demand grows.

Advanced tools and techniques

Modern product design relies heavily on advanced tools and techniques. Computer-Aided Design (CAD) software, such as SolidWorks and AutoCAD, enables detailed digital modeling, while rapid prototyping technologies like 3D printing and CNC machining allow for quick and precise prototype creation. Simulation and testing tools help predict performance and address potential issues before production. Additionally, project management software streamlines collaboration among multidisciplinary teams.

Importance of collaboration

Collaboration is the cornerstone of product development, bringing together diverse expertise from engineering, design, marketing, and manufacturing. Effective communication and iterative processes ensure alignment across teams and refine concepts to achieve the desired outcome. Stakeholder feedback at every stage further enhances the product's relevance and effectiveness.

Emerging trends in product design

Emerging trends are reshaping the field of product design. Sustainable design practices are gaining prominence, focusing on reducing environmental impact throughout the product lifecycle. Smart products integrating Internet of Things (IoT) and Artificial Intelligence (AI) enhance functionality and user experience. Additive manufacturing, particularly 3D printing, expands possibilities for complex designs and rapid production. Human-centric design emphasizes accessibility and inclusivity, ensuring products cater to diverse user groups. Digital twins, virtual models that simulate real-world performance, are revolutionizing how products are optimized before production.

Applications across industries

Applications of product design and development span various industries. Consumer products, such as electronics and home appliances, prioritize convenience and usability. Industrial equipment focuses on functionality and durability for specific applications. Medical devices require precision and reliability to address healthcare challenges. Automotive and aerospace components demand innovation in performance and safety. Sustainable solutions align with environmental goals by minimizing waste and using renewable resources.

Path to excellence in product design

To excel in product design, professionals should adopt a user-focused approach, leveraging iterative processes to refine ideas. Staying abreast of technological advancements and collaborating across disciplines are essential. By integrating sustainability and scalability into their designs, designers can create products that meet contemporary needs while paving the way for future innovation. Ultimately, the synergy of creativity, technology, and strategic thinking defines the success of product design and development.

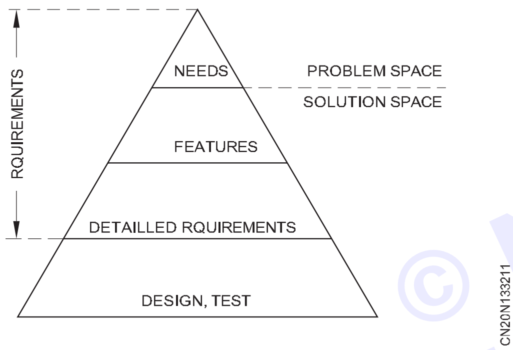
Customer's requirements, specification and importance of customer relationships

Objectives: At the end of this lesson you shall be able to
- **understand the importance of customer requirements**
- **identify different types of customer requirements**
- **recognize the importance of gathering customer needs.**

Importance of customer requirements

Customer requirements serve as the foundation of any successful engineering project. They represent the needs, preferences, and expectations of the target audience, guiding the design, development, and delivery processes. Accurately understanding these requirements ensures the product aligns with customer expectations, enhances user satisfaction, and gains a competitive edge in the market.

Fig 1



Types of customer requirements

1 **Functional requirements:** These specify what the product should do. They include features, capabilities, and tasks the product must perform to meet user needs.

2 **Non-functional requirements:** These refer to the quality attributes of the product, such as reliability, performance, usability, and security.

3 **Regulatory requirements:** These include compliance with industry standards, government regulations, and safety guidelines.

4 **Aesthetic requirements:** These pertain to the visual appeal and design elements that make the product attractive to users.

5 **Quality requirements:** Customers expect products to meet certain quality standards. This includes durability, consistency, and adherence to specified tolerances, ensuring the product performs as expected throughout its lifespan.

6 **On-time delivery requirements:** Timeliness is often a critical requirement. Customers expect the product to be delivered within the agreed-upon timeframe, ensuring minimal disruption to their operations or project timelines.

7 **Drawing requirements:** Clear and precise technical drawings are essential for manufacturing and assembly. Customers require these drawings to accurately represent the design, dimensions, materials, and other technical specifications, ensuring that the final product meets expectations.

Gathering customer requirements

The process of gathering customer requirements involves multiple methods to ensure comprehensive insights:

- **Interviews and surveys:** Direct engagement with potential customers to understand their preferences and pain points.
- **Focus groups:** Facilitating discussions among target users to gather qualitative feedback.
- **Market research:** Analyzing trends, competitors, and user behavior to identify unmet needs.
- **User observation:** Studying how customers interact with similar products to identify improvement opportunities.
- **Feedback and reviews:** Leveraging insights from existing product users to refine features and address issues.

Translating requirements into specifications

Specifications are the detailed and measurable descriptions derived from customer requirements. They guide the design and development teams by setting clear expectations. The process involves:

1 **Requirement analysis:** Categorizing and prioritizing requirements based on feasibility and importance.

2 **Defining metrics:** Establishing measurable parameters for requirements, such as weight, size, speed, and quality levels.

3 **Documentation:** Creating comprehensive specification documents to communicate expectations across teams, including details on quality checks, timelines, and drawing specifications.

Challenges in understanding customer requirements

Understanding customer requirements in engineering can be a complex task. Challenges often include:

- **Ambiguity:** Customers may express needs vaguely, making interpretation difficult.

- **Changing expectations:** Market dynamics and customer preferences can evolve, requiring adaptive strategies.
- **Conflicting requirements:** Different stakeholders may have diverging priorities that need reconciliation.
- **Technical constraints:** Translating customer needs into feasible technical solutions can be challenging.
- **Resource limitations:** Time, budget, and material constraints can impact the ability to fulfill all customer expectations.
- **Quality control:** Ensuring that all customer quality requirements are met within constraints.
- **On-time delivery:** Meeting the customer's expectations for delivery deadlines while maintaining quality standards.

Best practices for requirement gathering

- **Active listening:** Ensure clarity by asking questions and paraphrasing customer inputs.
- **Use of prototypes:** Visual representations help customers better articulate their needs.
- **Iterative feedback loops:** Regularly review and refine requirements through continuous interaction.
- **Collaborative tools:** Use software platforms to document and track requirements in real-time.
- **Scenario analysis:** Simulating various use cases to ensure all potential needs are covered.
- **Engaging cross-functional teams:** Involving engineers, designers, and project managers to provide diverse perspectives.

Real-world applications

- **Engineering product design:** Creating tools, machines, and systems that meet specific industrial needs.
- **Infrastructure projects:** Developing buildings, bridges, and facilities that adhere to customer specifications and safety standards.
- **Automotive engineering:** Designing vehicles with features tailored to performance, safety and user preferences.
- **Manufacturing systems:** Building efficient production lines based on precise customer requirements.
- **Renewable energy systems:** Developing wind turbines, solar panels, and other green technologies tailored to customer and environmental needs.
- **Software engineering:** Creating scalable, user-friendly applications with robust performance standards.

Understanding customer requirements and translating them into actionable specifications is a critical step in the engineering lifecycle. It ensures alignment between user needs and engineering solutions, driving innovation, customer satisfaction, and project success. By employing systematic methods and adhering to best practices, engineers can effectively meet and exceed customer expectations. Moreover, leveraging advancements in technology and fostering cross-disciplinary collaboration further enhances the ability to deliver innovative and reliable solutions tailored to customer demands. Special attention to quality, on-time delivery, and clear technical drawings further ensures that products meet both customer expectations and industry standards.

Importance of customer relationship

Customer relationships are the cornerstone of any business's success, influencing profitability, market position, and long-term sustainability. A customer relationship is not merely a transactional interaction; it involves creating a meaningful connection that fosters trust, satisfaction, and loyalty over time. As markets become more competitive and consumer expectations evolve, building strong and lasting relationships with customers has become even more critical.

Why customer relationships matter?

1 Retaining customers is cost-effective

Acquiring new customers involves significant investment in marketing, sales, and onboarding processes. In contrast, retaining existing customers requires fewer resources. Businesses that focus on nurturing existing relationships save costs and benefit from repeat business. Loyal customers are also less sensitive to price changes, often valuing the relationship and trust they have built with the company over time.

For example, subscription-based businesses like Netflix or Amazon Prime rely heavily on customer retention, ensuring they provide consistent value to maintain relationships with subscribers.

2 Building loyalty ensures stability

Strong customer relationships translate into loyalty. Loyal customers not only continue to purchase but also act as brand advocates, promoting the business through word-of-mouth recommendations. In markets where products and services are similar, loyalty is a key differentiator that provides a stable revenue stream.

Apple Inc., for instance, has cultivated a deeply loyal customer base by combining innovative products with exceptional customer experiences, ensuring customers return for upgrades and accessories.

3 Increased lifetime value

The lifetime value of a customer refers to the total revenue a business can generate from a single customer over their relationship with the company. Strong relationships encourage customers to engage with the business more frequently and make higher-value purchases. They may also explore additional products or services offered by the business, increasing their overall contribution to the company's growth.

For instance, a loyal customer at Starbucks might initially buy a coffee but later explore seasonal beverages, snacks, and loyalty card benefits, significantly enhancing their lifetime value.

4 Word-of-mouth and organic growth

Satisfied customers who trust and appreciate a brand often recommend it to others. This form of word-of-mouth marketing is invaluable, as potential customers are more likely to trust personal recommendations over traditional advertising. A positive reputation built through strong customer relationships can lead to organic growth, reducing dependency on costly marketing campaigns.

Brands like Tesla have grown significantly without heavy investment in traditional advertising, relying on their customers' enthusiasm and advocacy to spread the word.

5 Deep understanding of customer needs

Strong relationships provide businesses with insights into customer behavior, preferences, and expectations. Companies that actively engage with customers can adapt their products and services to better meet market demands. By fostering open communication, businesses can also identify and address pain points before they escalate, improving overall satisfaction.

For example, through feedback loops and surveys, companies like Airbnb consistently refine their offerings to align with user preferences and market trends.

6 Creating a competitive advantage

In today's crowded marketplace, where competitors often offer similar products or services, customer relationships can set a business apart. A company known for exceptional service and care can outshine rivals purely by the strength of its customer relationships.

Consider Zappos, an online shoe and clothing retailer, which has built a competitive edge by going above and beyond to delight customers with its service, including free returns and surprise upgrades.

7 Crisis management and resilience

During times of crisis - whether a product failure, a supply chain disruption, or negative publicity - businesses with strong customer relationships often face less backlash. Loyal customers who trust the brand are more likely to be understanding and forgiving. Furthermore, transparent communication during crises can strengthen trust and reinforce the relationship.

A notable example is Johnson & Johnson's handling of the Tylenol tampering crisis in the 1980s. The company's swift response and prioritization of customer safety preserved its reputation and customer trust.

How to strengthen customer relationships?

1 **Personalization:** Tailor experiences and offerings to individual customer needs. Personalization creates a sense of value and understanding, making customers feel unique and appreciated.

2 **Consistent communication:** Keep customers informed and engaged through various channels such as email updates, social media, and newsletters. Consistent communication fosters trust and keeps the brand top of mind.

3 **Excellent customer service:** Respond promptly and effectively to customer inquiries and complaints. Exceptional service creates memorable experiences, turning even dissatisfied customers into loyal advocates.

4 **Customer feedback and involvement:** Actively seek customer input and involve them in shaping products or services. Customers who feel heard are more likely to remain loyal.

5 **Reward loyalty:** Implement programs that recognize, and reward repeat customers, such as discounts, exclusive access, or free products. This reinforces their decision to stay with the brand.

6 **Emphasize trust and transparency:** Be honest and transparent in all dealings, especially when things go wrong. Trust is the foundation of any strong relationship.

Introduction to engineering drawing

Objectives: At the end of this lesson you shall be able to
- **explain use of engineering drawing**
- **understand basic components of engineering drawing.**

Introduction

Drawing is the Graphical means of expression of technical details without the barrier of a language. Engineering Drawing is the Universal Language for Engineers.

Communication in engineering is necessary for effectively transferring one's ideas to others. While communicating, we use our memory to remember objects, sense organs to perceive objects and mind to imagine objects. Our perception is coloured or modified by our past experiences.

We see things around us, perceive the objects and identify them by their names. Later, when we hear these names we can remember these items easily and imagine various features like the shape, size, color, functions etc, e.g. if I say that a particular object is having the shape of a cricket bat, it is easy for a high school student to imagine the shape of the object since he has seen a cricket bat and has perceived the object. If I say that a particular object looks like the Dendrite formed during solidification of a metal, then it is difficult for the same student to imagine, since he has not seen a dendrite. His question will "how do a dendrite look like?" when we say that dendrite means tree-like structure, his imagination can go to different types of trees, but still he may not have a clear picture of the dendrite. If we show him a picture of a dendrite, he can very easily perceive that object.

One picture/drawing is equivalent to several sentences. It is not easy for anyone to make another person understand somebody's face just by explaining the features. Even if several sentences are used to explain the features of the face, words it would be difficult for the listener to perceive the image of the face. However, if you show a sketch or a photograph of the person, all these sentences can be saved. i.e., We grasp information easily if it is illustrated with diagrams, sketches, pictures, etc.

Engineering drawing

Drawings help us in developing our thoughts and ideas into a final product.

Drawings are also necessary for engineering industries since they are required and are being used at various stages of development of an engineering product. Engineering drawing is completely different from artistic drawing, which are used to express aesthetic, philosophical, and abstract ideas. In an industry, these drawings help both the technical as well as commercial staffs at various stages like:

- conceptual stage
- design stage
- modification stage
- prototype development stage
- process and production planning
- production
- inspection etc.

What information should be available in an engineering drawing?

A perfect engineering drawing should have the following information:

- Shape of an object
- Exact Sizes and tolerances of various parts of the object
- The finish of the product
- The details of materials
- The company's name
- Catalogue no of the product
- Date on which the drawing was made
- The person who made the drawing

Layout of a drawing sheet

1 Borders

A minimum of 10 mm space left all around in between the trimmed edges of the sheet.

2 Filing margin

Minimum 20 mm space left on the left-hand side with border included. This provided for taking perforations.

3 Grid reference system

This is provided on all sizes of industrial drawing sheets for easy location of drawing within the frame.

4 Title block

This is located at the bottom right-hand corner of every sheet and provides the technical and administrative details of the drawing.

Title block (Figs 1 & 2)

The title box is divided into two zones.

Fig 1

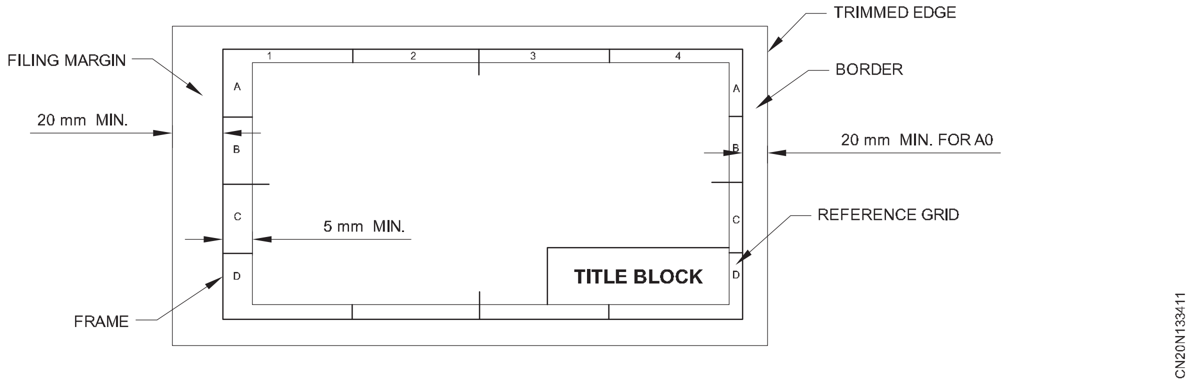
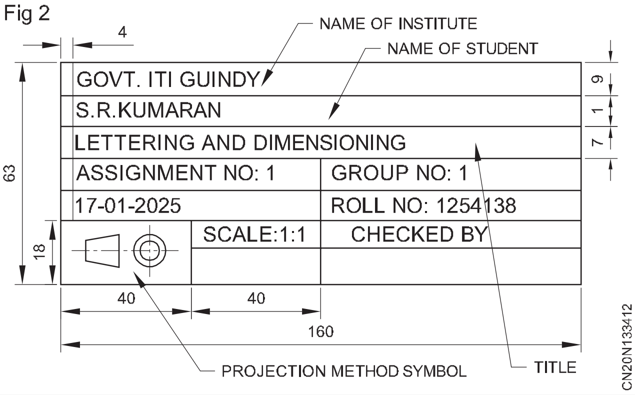


Fig 2



- **Identification zone**

  In this zone the details like the identification number or part number, Title of the drawing, legal owner of the drawing, etc. are to be mentioned.

- **Additional information zone**

  Here indicative items lime symbols indicting the system of projection, scale used, etc., the technical items line method of surface texture, tolerances, etc., and other administrative items are to be mentioned.

Basic components of an engineering drawing

1 **Different types of lines:** Not every line on an engineering drawing is equal. The different options make it possible to show both visible and hidden edges of a part, centre lines, etc.

Continuous lines

Continuous line, also known as a drawing line. This represents the physical boundaries of an object. Put simply, these lines are for drawing objects. The line thickness varies – the outer contour uses thicker lines and the inner lines are thinner.

Hidden lines

Hidden lines can show something that would not be otherwise visible on the drawings. For example, hidden lines may show the length of an internal step in a turned part without using a section or a cutout view (we explain both later).

Centre lines

Centre lines are used to show holes and the symmetric properties of parts. Showing symmetricity can reduce the number of dimensions and make the drawing more eye-pleasing, thus easier to read.

Extension lines

Extension lines annotate what is being measured. The dimension line has two arrowheads between the extension lines and the measurement on top (or inside, like in the image above) the line.

Break lines

Indicate that a view has been broken. If you have a part that is 3000 mm long and 10 mm wide with symmetric properties, using a break-out makes gives all the info without using as much space.

Illustration	Application
Thick ————————	Outlines, visible edges, surface boundaries of objects, margin lines
Continuous thin ————————	Dimension lines, extension lines, section lines leader or pointer lines, construction lines, boarder lines
Continuous thin wavy ∼∼∼∼	Short break lines or irregular boundary lines - drawn freehand

Continuous thin with zig-zag	Long break lines
Short dashes, gap 1, length 3 mm	Invisible or interior surfaces
Short dashes	Center lines, locus lines, alternate long and short dashes in a proportion of 6:1
Long chain thick at end and thin elsewhere	Cutting plane lines

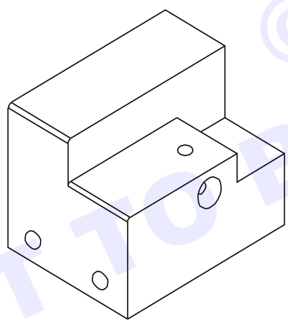
2 Types of views

Isometric view

Isometric drawings show parts as three-dimensional. All the vertical lines stay vertical (compared to the front view) and otherwise parallel lines are shown at a 30-degree angle.

The lines that are vertical and parallel are in their true length. This means you can use a ruler and the scaling of the drawing to easily measure the length straight from a paper drawing, for example. The same does not apply to angled lines. (Fig 3)

Fig 3



CN20N133413

Orthographic View

An orthographic view or orthographic projection is a way of representing a 3D object in 2 dimensions.

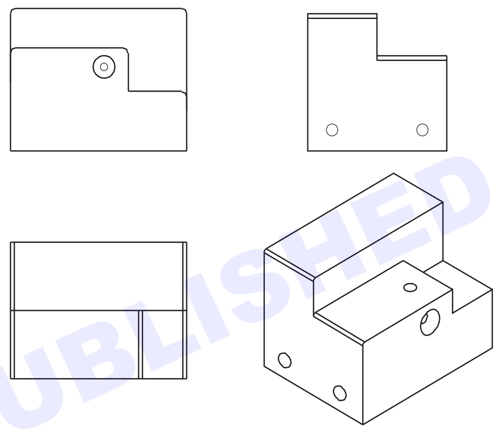
Thus, a 2D view has to convey everything necessary for part production. This kind of representation allows avoiding any kind of distortion of lengths. (Fig 4)

Orthographic projection (ISO standard) (Fig 5)

The most common way to communicate all the information is by using three different views in a multi view drawing:

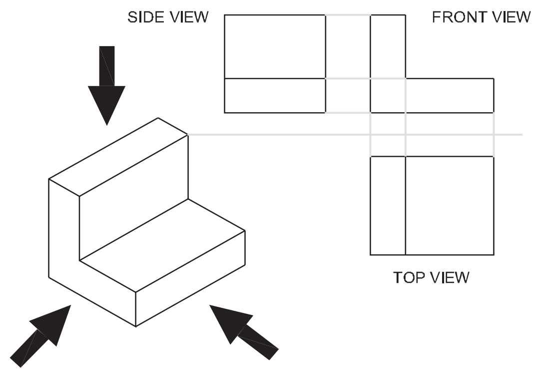
- Front view
- Top view
- Side view

Fig 4



CN20N133414

Fig 5



CN20N133415

3 Dimensioning

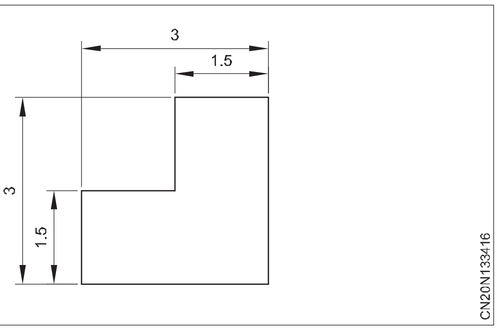
"Dimensioning" in engineering drawings are numerical values indicated graphically in a proper unit of measurement on engineering drawings with lines, symbols, and notes. These are indicated on the engineering drawing to define the size characteristics such as length, height, breadth, diameter, radius, angle, etc. Dimensioning Drawing provides complete shape description and furnishes information regarding the size and description. These are provided through the distances between the surfaces, location of holes, nature of surface finish, type of material, etc. You can create dimensions for existing entities by selecting them, or you can create dimensions by selecting points within a drawing.

Principles of Dimensioning

Some of the basic principles of dimensioning are:

- All dimensional information necessary to describe a component clearly and completely shall be written directly on a drawing.
- Each feature shall be dimensioned once only on a drawing, i.e., the dimension marked in one view need not be repeated in another view.
- Dimension should be placed on the view where the shape is best seen.
- As far as possible, dimensions should be expressed in one unit only preferably in millimeters, without showing the unit symbol (mm). (Fig 6)

Fig 6



- As far as possible dimensions should be placed outside the view.
- Dimensions should be taken from visible outlines rather than from hidden lines.
- No gap should be left between the feature and the start of the extension line.
- Crossing of center lines should be done by a long dash and not a short dash.

Fundamentals of limits, fits & tolerances & importance of interchangeability and ISO standards

Objectives: At the end of this lesson you shall be able to
- **state the necessity of limit system**
- **understand symbols used for limits & fits**
- **importance of ISO standards.**

Limits, fits and tolerances

Limits, fits, and tolerances are fundamental concepts in the field of engineering and manufacturing, serving as the anchor for ensuring the functionality and interchangeability of mechanical components. These principles define the acceptable variation in size and geometry of parts, ensuring that they can fit together precisely while still allowing for a certain degree of variability. With meticulous attention to these standards, engineers and manufacturers can achieve the delicate balance between tight clearances for accuracy and precision and the flexibility needed for efficient production, ultimately creating reliable, well-fitting, and interchangeable components that drive the machinery and devices shaping our modern world.

Necessity of limit system

It is practically impossible to machine components to an exact size, due to the varying skills of the operators, the condition of the machine tools, the quality of the cutting tools and the accuracy of the precision instrument used. Hence some permissible deviations to the exact size are accepted and given, and the operator is expected to produce the components within the limits, which, even though not necessarily equal to the exact size, will not affect the functioning of the components. This necessitates the introduction of the limit system.

Internationally accepted systems of limits and fits

- British Standard System of Limits and Fits (B.S.).
- International Standard Organization System of Limits and Fits. (I.S.O.)
- Bureau of Indian Standard System of Limits and Fits (B.I.S.)

Apart from the above most commonly used limit systems, various countries follow their own standards to manufacture components for some of their industries.

Advantages of the limit system

- Interchangeability is assured.
- Not necessary to employ highly skilled operators.
- Not necessary to use conventional measuring instruments.
- Time for the manufacture of components will be comparatively less.

Limits, fits and tolerance

The B.I.S. standard system of limits and fits is followed by the industries in our country as the standard. It is adopted from the I.S.O. and B.S. standards with modifications to suit our conditions and requirements. For the purpose of B.I.S. standard, the following definitions and symbols are followed.

Size

It is a number expressed in a particular unit in the measurement of length.

Basic size

It is the size based on which the dimensional deviations are given.

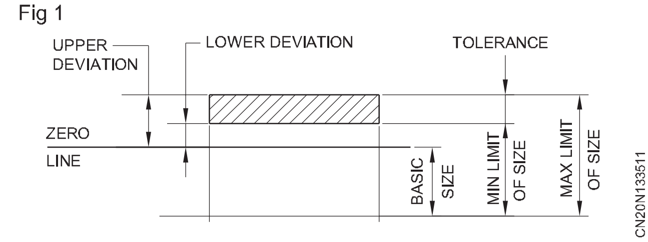
Maximum limit of size

It is the greater of the two limit sizes. (Fig 1)

Minimum limit of size

it is the smaller of the two limits of size. (Fig 1)

Fig 1



Actual size

It is the size of the component by actual measurement after it is manufactured, it should lie between the two limits of size if the component is to be accepted.

Limits of size

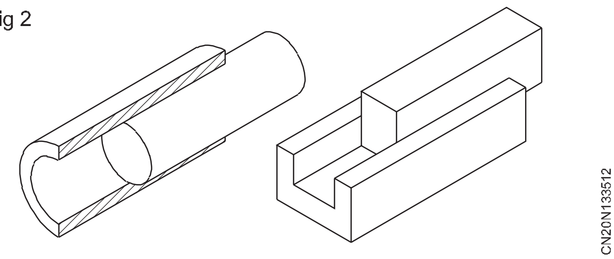
These are the extreme permissible sizes within which the operator is expected to make the component. (Maximum and minimum limits)

Hole

In the BIS system of limits & fits, all internal features of a component including those which are not cylindrical are designated as 'hole'. (Fig 2)

Shaft: In the BIS system of limits & fits, all external features of a component including those which are not cylindrical are designated as 'shaft'. (Fig 2)

Fig 2



Deviation: It is the algebraic difference between a size, to its corresponding basic size. It may be positive, negative or zero.

Upper deviation: It is the algebraic difference between the maximum limit of size and its corresponding basic size.

Lower deviation: It is the algebraic difference between the minimum limit of size and its corresponding basic size.

Upper deviation is the deviation which gives the maximum limit of size. Lower deviation is the deviation which gives the minimum limit of size.

Actual deviation

It is the algebraic difference between the actual size and its corresponding basic size.

Table 1

Sl.No.	Size of components	Upper deviation	Lower deviation	Max-limit of size	Min-limit of size
1	$20.0^{+0.008}_{-0.005}$	+0.008	-0.005	20.008	19.995
2	$20.0^{+0.028}_{+0.007}$	+0.028	+0.007	20.028	20.007
3	$20.0^{-0.012}_{-0.021}$	-0.012	-0.021	19.988	19.979

Tolerance

It is the difference between the maximum limit of size and the minimum limit of size. It is always positive and is expressed only as a number without a sign.

Zero line

In graphical representation of the above terms, the zero line represents the basic size. This line is also called as the line of zero deviation.

Fundamental deviation

There are 25 fundamental deviations in the BIS system represented by letter symbols (capital letters for holes and small letters for shafts), i.e. for holes - ABCD....Z excluding I,L,O,Q & W. For shafts, the same 25 letter symbols but in small letters are used.

Fundamental tolerance

This is also called as 'grade of tolerance'. In the Indian Standard System, there are 18 grades of tolerances represented by number symbols, both for hole and shaft, denoted as ITOI, ITO, ITI....t0 IT16. A high number gives a large tolerance zone.

The grade of tolerance refers to the accuracy of manufacture.

Toleranced size

This includes the basic size, the fundamental deviation and the grade of tolerance.

Example

25 H7 - toleranced size of a hole whose basic size is 25. The fundamental deviation is represented by the letter symbol H and the grade of tolerance is represented by the number symbol 7. (Fig 3)

25 e8 - is the toleranced size of a shaft whose basic size is 25. The fundamental deviation is represented by the letter symbol 'e' and the grade of tolerance is represented by the number 8. (Fig 4)

Fig 3

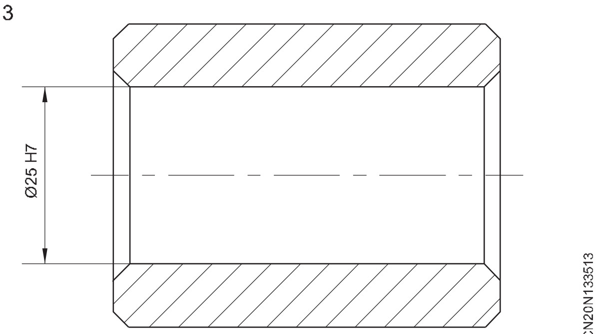
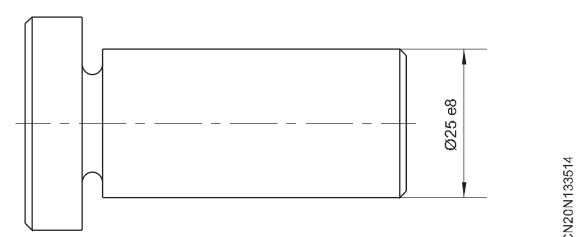


Fig 4

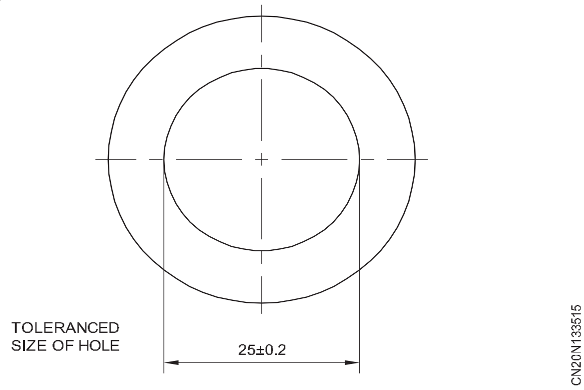


In Fig 6, a hole is shown as 25±0.2 which means that 25 mm is the basic dimension and ±0.2 is the deviation

As pointed out earlier, the permissible variation from the basic dimension is called 'DEVIATION' The deviation is mostly given on the drawing with the dimensions.

In the example 25 ±0.2, ± 0.2 is the deviation of the hole of 25 mm, diameter. (Fig 5) This means that the hole is of acceptable size if its dimension is between.

Fig 5



25 + 0.2 = 25.2 mm

or 25 - 0.2 = 24.8 mm.

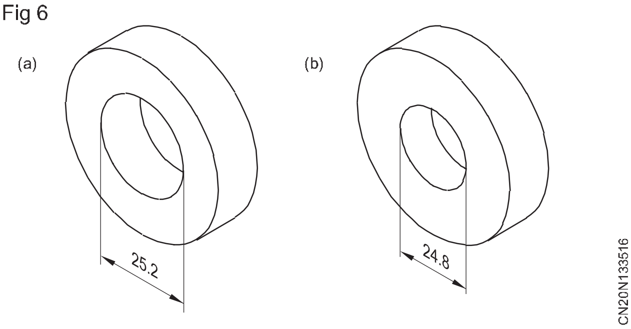
25.2 mm is known as the maximum limit. (Fig 6a)

24.8 mm is known as the minimum limit. (Fig 6b)

The difference between the maximum and minimum limits is the TOLERANCE.

Tolerance here is 0.4 mm

Fig 6



Unilateral & bilateral system

When the deviations given for a particular combination of the symbols are positive and negative so as to give the maximum limit more than the basic size and the minimum limit less than the basic size, then we call it bilateral tolerancing. If the deviations have only positive or negative values and have both the maximum limit and minimum limit more than the basic size or less than the basic size respectively, then it is called unilateral tolerancing.

Fits and their classification as per the Indian standard IS : 919

Objectives: At the end of this lesson you shall be able to
- **state the expression of fit**
- **explain the types of fits.**

Fit

It is the relationship that exists between two mating parts, a hole and a shaft, with respect to their dimensional differences before assembly.

Expression of a fit

A fit is expressed by writing the basic size of the fit first, (the basic size which is common to both the hole and the shaft,) followed by the symbol for the hole, and by the symbol for the shaft.

Example

30 H7/g6 or 30 H7 - g6 or 30 H7/g6

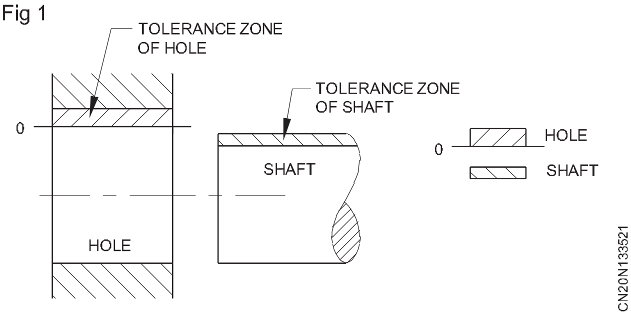
Clearance

In a fit the clearance is the different between the size of the hole and the size of the shaft which is always positive.

Clearance fit

It is a fit which always provides clearance. Here the tolerance zone of the hole will be above the tolerance zone of the shaft. (Fig 1)

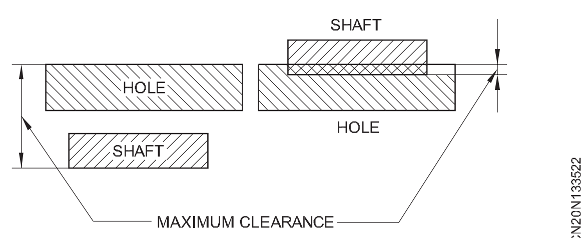
Fig 1



Maximum clearance

In a. clearance fit or transition fit, it is the difference between the maximum hole and minimum shaft.

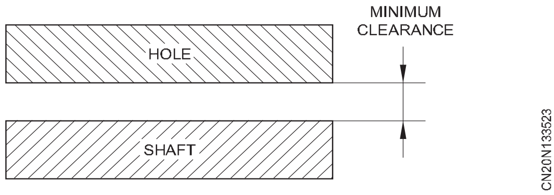
Fig 2



Minimum clearance

In a clearance fit, it is the difference between the minimum hole and the maximum shaft.

Fig 3



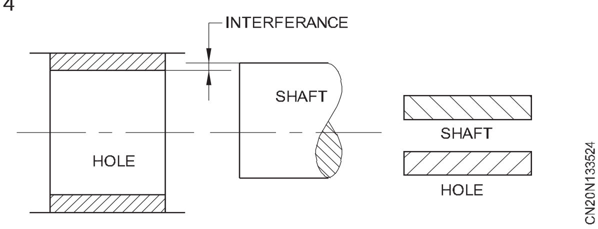
Interference

It is the difference between the size of the hole and the shaft before assembly, and this is negative. In this case, the shaft is always larger than the hole size.

Interference fit

It is a fit which always provides interference. Here the tolerance zone of the hole will be below the tolerance zone of the shaft.

Fig 4



Maximum interference

In an interference fit or transition fit, it is the algebraic difference between the minimum hole and the maximum shaft.

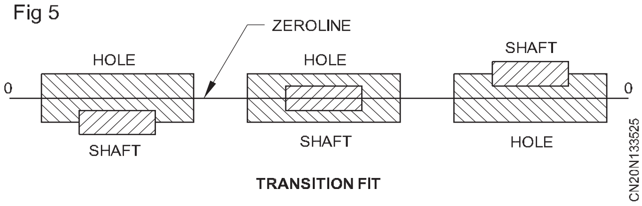
Minimum interference

In an interference fit, it is the algebraic difference between the maximum hole and the minimum shaft.

Transition fit

It is a fit which may sometimes provide clearance, and sometimes interference. When this class of fit is represented graphically, the tolerance zones of the hole and shaft will overlap each other. (Fig 5)

Fig 5



Interchangeability

Interchangeability is a foundational principle in engineering and manufacturing that allows components or parts to be replaced with identical units without requiring custom fitting or modifications. This concept is particularly vital in industries where precision, efficiency, and cost-effectiveness are paramount.

In earlier times, components were crafted individually by skilled artisans, making each piece unique. This meant that replacing a broken part required custom work, leading to high costs and delays. Interchangeability revolutionized this process by enabling mass production of standardized parts that fit together seamlessly.

Key benefits

1 **Ease of maintenance:** Interchangeability simplifies the maintenance process by ensuring that parts can be replaced easily. For example, if a machine component fails, it can be quickly swapped out with a standardized spare part, minimizing downtime. This is particularly important in industries like automotive manufacturing and aerospace, where even minor delays can have significant financial and operational impacts.

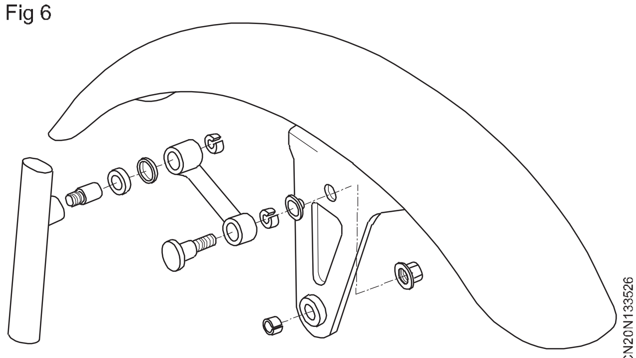
2 **Mass production:** The principle of interchangeability is integral to mass production systems. Factories can produce large quantities of parts to exact specifications, reducing the need for skilled labor and manual fitting. This not only enhances production efficiency but also lowers costs.

3 **Quality assurance:** Standardized parts improve product quality by ensuring consistency. Every component manufactured according to the same specifications performs identically, which is crucial in applications where reliability and safety are non-negotiable, such as medical devices or industrial machinery.

4 **Global collaboration:** Interchangeability fosters global collaboration in manufacturing. A company in one country can produce parts that are used by a manufacturer in another, thanks to standardized dimensions and tolerances. This level of compatibility supports international trade and supply chains.

5 **Cost reduction:** By eliminating the need for custom fitting or reworking parts, interchangeability reduces production and maintenance costs. This makes products more affordable and accessible to a wider market. (Fig 6)

Fig 6



Importance of ISO standards

The International Organization for Standardization (ISO) develops globally recognized standards that ensure consistency, safety, and quality in engineering, manufacturing, and trade. These standards provide a common framework for organizations to produce and manage products efficiently.

Key contributions of ISO standards

1 **Uniformity and consistency:** ISO standards define precise tolerances, fits, and measurements that manufacturers across the world can adhere to. This ensures that parts produced by different manufacturers are consistent in quality and dimension, enabling seamless integration into larger systems.

2 **Interoperability:** The global adoption of ISO standards allows components from different manufacturers or regions to work together without compatibility issues. This is particularly significant in complex systems like airplanes, automobiles, or electronic devices, where parts come from diverse suppliers.

3 **Improved quality:** Adhering to ISO standards ensures a high level of quality in products and processes. For instance, ISO 9001 sets benchmarks for quality management systems, promoting continual improvement and customer satisfaction.

4 **Reduced errors and rework:** ISO standards minimize errors in production by providing clear guidelines and specifications. Manufacturers can avoid costly rework or rejection of parts, thereby improving efficiency and reducing waste.

5 **Ease of communication:** ISO standards establish a universal language for engineering and manufacturing. Designers, engineers, and manufacturers from different backgrounds can use standardized terminologies and measurements to collaborate effectively.

6 **Environmental and safety standards:** ISO also sets guidelines for environmental sustainability and workplace safety. For instance, ISO 14001 focuses on environmental management systems, ensuring that manufacturing processes minimize their ecological impact.

Role of ISO standards in interchangeability

ISO standards and interchangeability are intrinsically linked. The universal system of tolerances and fits defined by ISO ensures that parts produced by different manufacturers are interchangeable without any modification.

1 **Dimensional tolerances and fits:** Standards like ISO 286 specify tolerances and fits for components, ensuring that parts manufactured to these standards fit together precisely. For example, an ISO-defined H7/g6 fit guarantees a clearance fit that works consistently across manufacturers.

2 **Global compatibility:** By adhering to ISO standards, manufacturers can create components that are compatible worldwide. This is especially valuable in industries like automotive and electronics, where supply chains span multiple countries.

3 **Reduced inventory:** Standardization through ISO reduces the need to stock diverse, non-interchangeable parts. Companies can maintain streamlined inventories, saving storage costs and improving logistics.

4 **Streamlined supply chains:** ISO standards facilitate smoother supply chain operations by ensuring that all components conform to the same specifications. This reduces delays caused by compatibility issues and enhances overall efficiency.

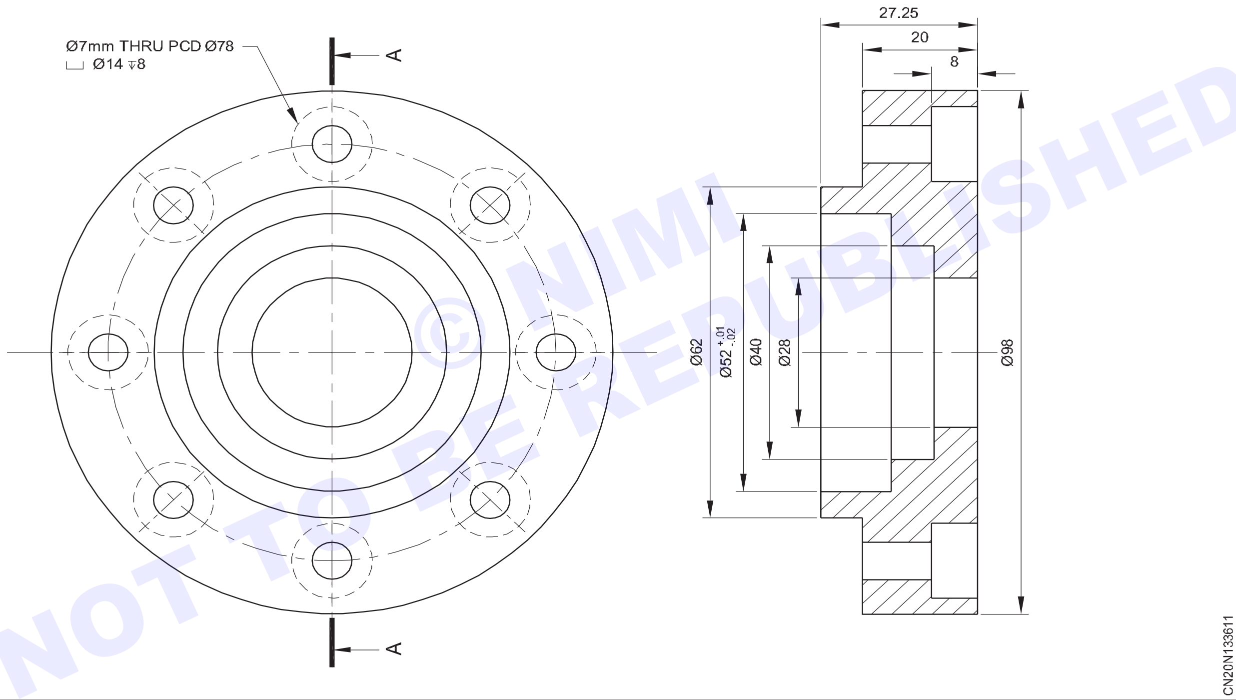
The importance of interchangeability and ISO standards cannot be overstated in modern engineering and manufacturing. Interchangeability streamlines maintenance, supports mass production, and ensures quality, while ISO standards provide the framework for global consistency and compatibility. Together, they enable industries to innovate, reduce costs, and deliver high-quality products efficiently, making them indispensable in today's interconnected world.

Industrial engineering special characteristic symbol

Objectives: At the end of this lesson you shall be able to
- **enhance product quality and reliability**
- **understand engineering special characteristics symbol**
- **understand customer- specific standards**
- **understand geometrical dimensions & tolerances.**

Industrial engineering special characteristics symbols: Special characteristics symbols in industrial engineering highlight critical features of a product or process that directly impact its performance, safety, or compliance. These symbols are often standardized or customer-specific. For example, Critical Characteristics (CC) are denoted by a diamond or "C" and signify attributes that are essential for meeting safety or regulatory requirements. Similarly, Significant Characteristics (SC), represented by a triangle or "S," indicate features important for the product's functionality and reliability. Key Characteristics (KCs), marked with a "K," identify elements critical for assembly or operation. Recognizing and addressing these symbols helps prioritize quality and minimize risks in manufacturing.

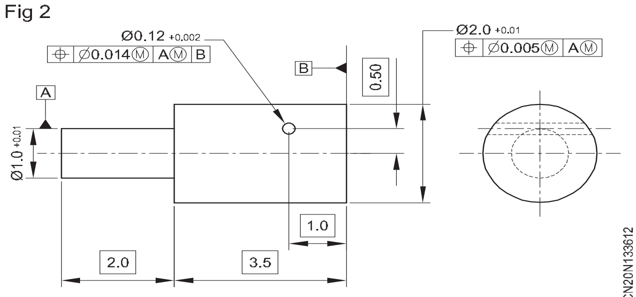
Fig 1



- Maximum material condition MMC
Ⓜ - Regardless feature size RFS
Ⓛ - Least material condition LMC
Ⓟ - Projected tolerance zone
ø - Diametrical (Cylindrical) tolerance zone or feature
0.500 - Basic or exact, dimension
- Datum feature symbol
⌖ Ø0.003Ⓜ A - Feature control frame

Customer-specific standards (CSS): Customer-specific standards (CSS) are tailored requirements defined by individual clients to ensure products meet unique operational or design needs. These standards often specify proprietary drawing formats, distinct symbols for tolerances, and unique surface finish or welding notations. CSS may also include particular inspection methods or reporting styles, ensuring consistency and alignment with the customer's expectations. By adhering to CSS, manufacturers can enhance customer satisfaction and build trust, as well as maintain competitive advantages in industries where precision and customization are paramount.

Fig 2



Geometrical dimensions & tolerances

Objectives: At the end of this lesson you shall be able to
- **define geometrical tolerance**
- **different characteristic used in engineering drawing**
- **state the necessity of using geometrical tolerances**
- **Identify the recommended symbols for tolerance under three graphs of form, attitude and location**
- **geometrical tolerances for the control of angularity.**

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

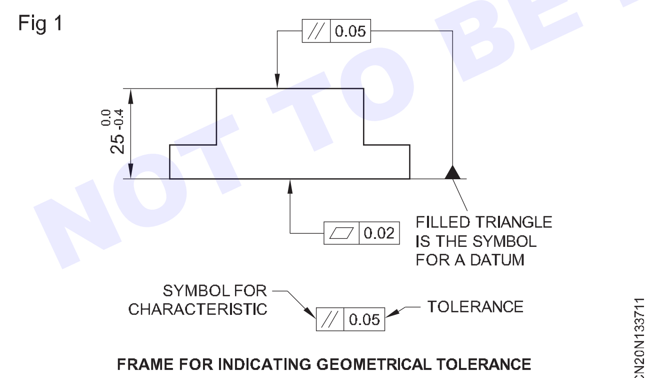
this will help the operator to produce the components, particularly those parts that must fit together precisely.

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. (Fig1)

Fig 1

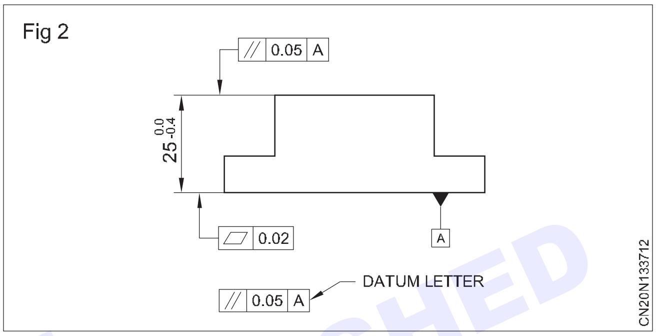


You will notice that from each of the frames a leader is drawn so that it is normal, i.e. at 90° to the relevant face and ending with an arrowhead against the face.

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 tolerance.

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

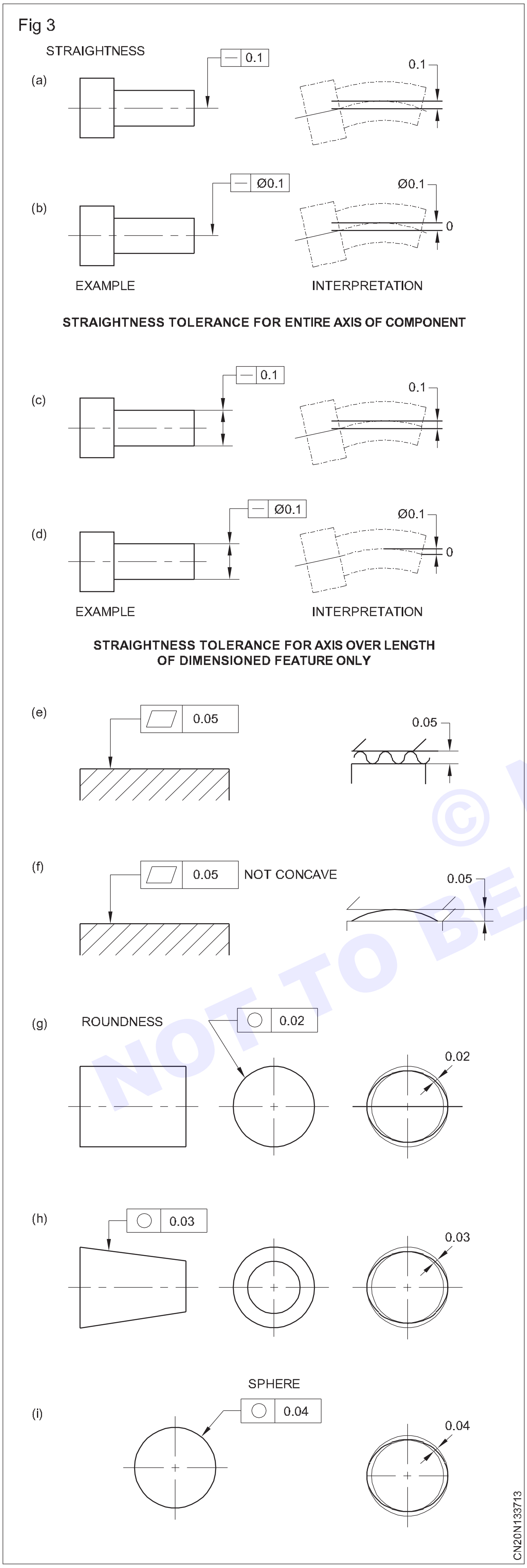
Parallelism, square-ness and angularity.

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 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 r 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 prohibited, then this stated in a note form against the tolerance frame. e.g. 'Not concave'.

The geometrical tolerance controlling the roundness of a part 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.

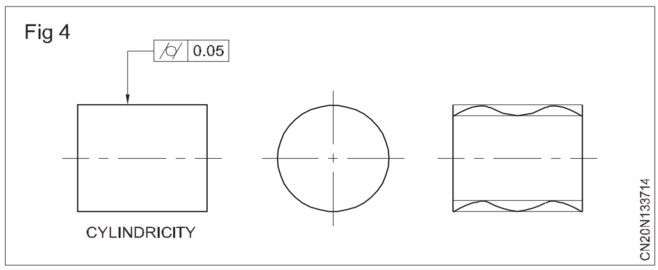


Fig 5 shows the method of applying a geometrical tolerance to a curved surface. The symbol followed by the tolerance 0.05, which means that the actual surface must lie between two surfaces enveloping a succession of sphere 0.05 diameter whose center lies on the theoretical surface.

In Fig 6, the geometrical tolerance applied to linear dimensions controlling the profile. The rectangular 'boxes' around the 250 center 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 center lies on the theoretical profile.

Fig 5

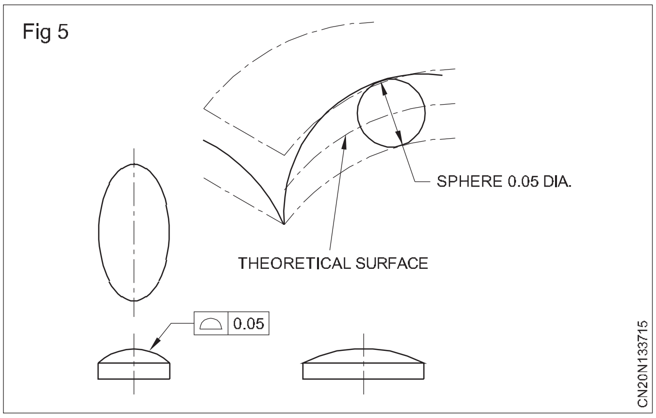
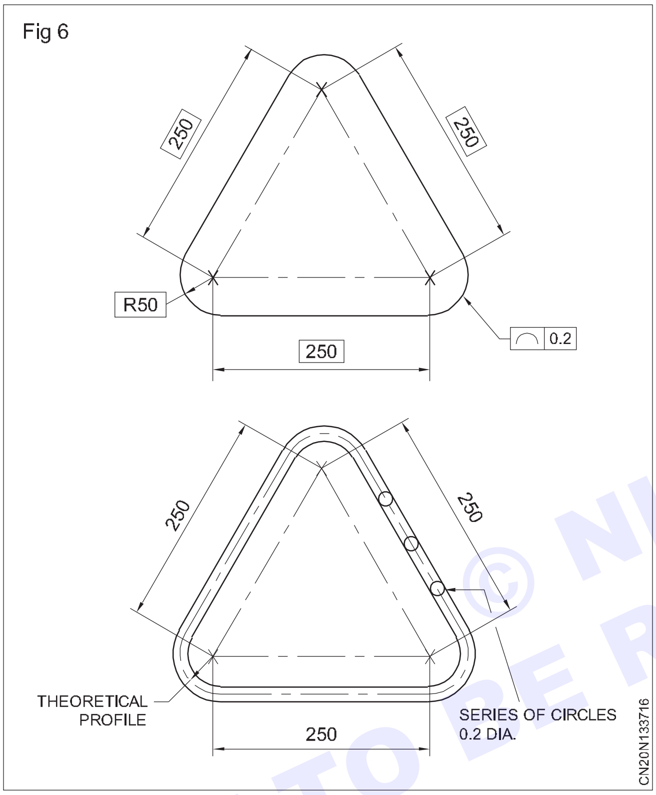


Fig 6



Attitude

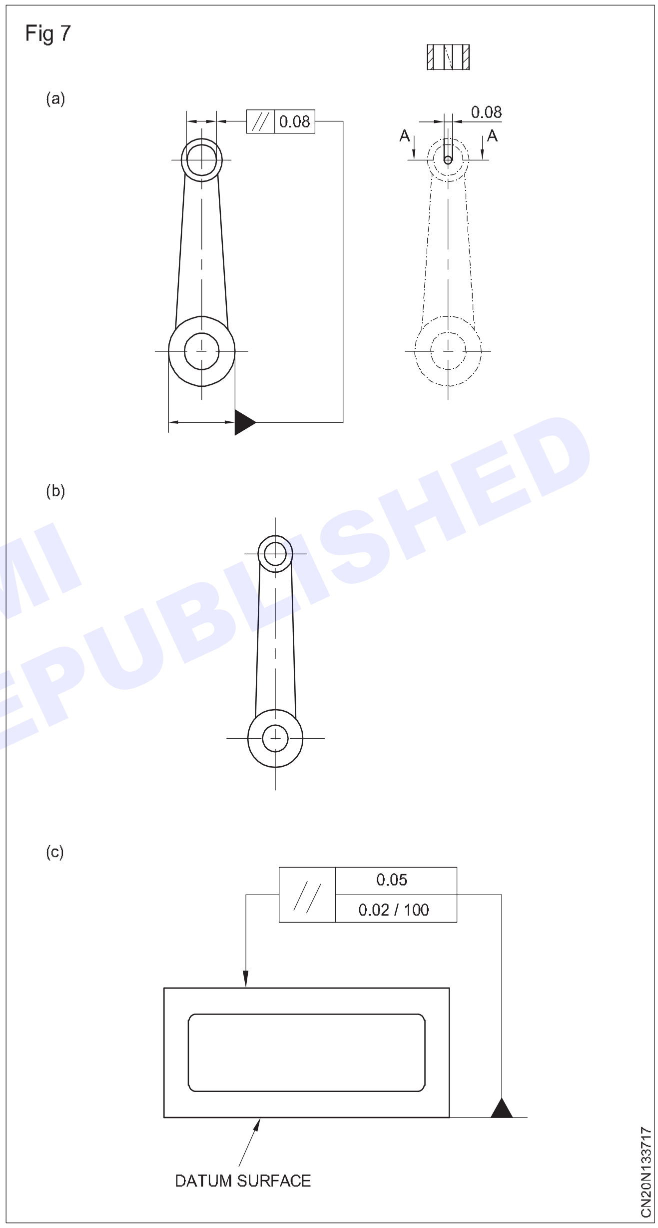
Tolerances of attitude are identified and indicated by the use of symbols for the following.

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 center lies on the theoretical profile.

Characteristics	Symbol
Parallelism	//
Squareness	⊥
Angularity	∠

A typical application of tolerances for these three characteristics 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

Fig 7



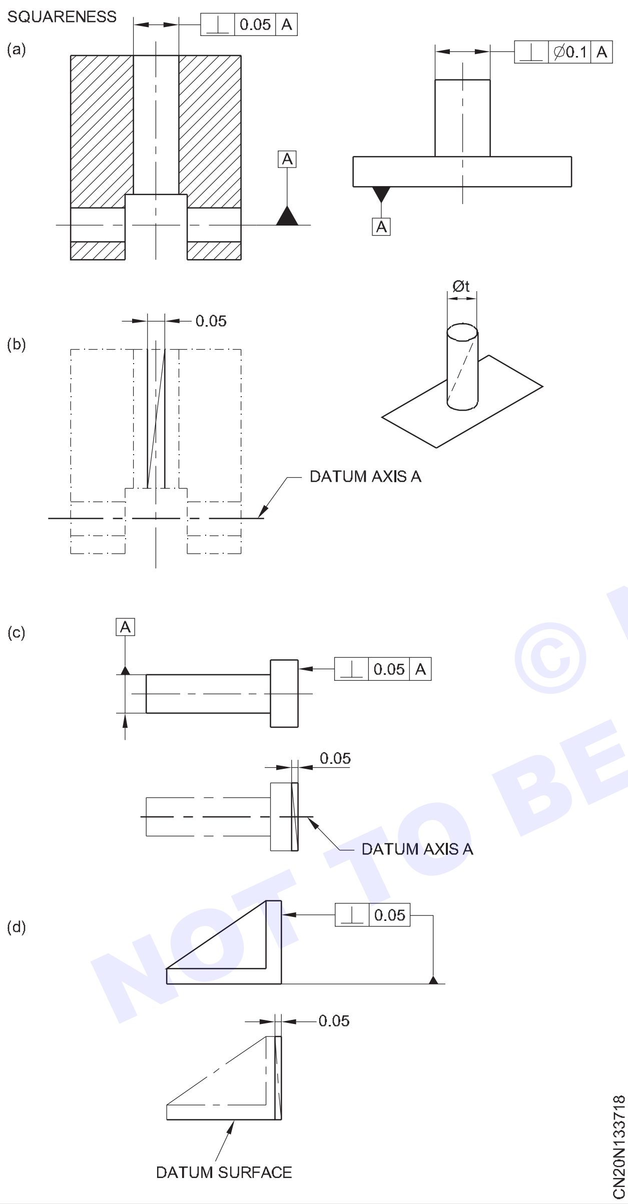
0.05 diameter with its axis parallel with the axis of the Datum hole 'A'. Fig (7c) shows a component whose upper surface must be between two parallel planes 0.05 apart, parallel with the bottom datum surface. While the overall tolerance zone is 0.05 as shown in the upper section of the frame, the fig in the lower section of the frame stipulate that over any length of 100 the parallelism tolerance 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



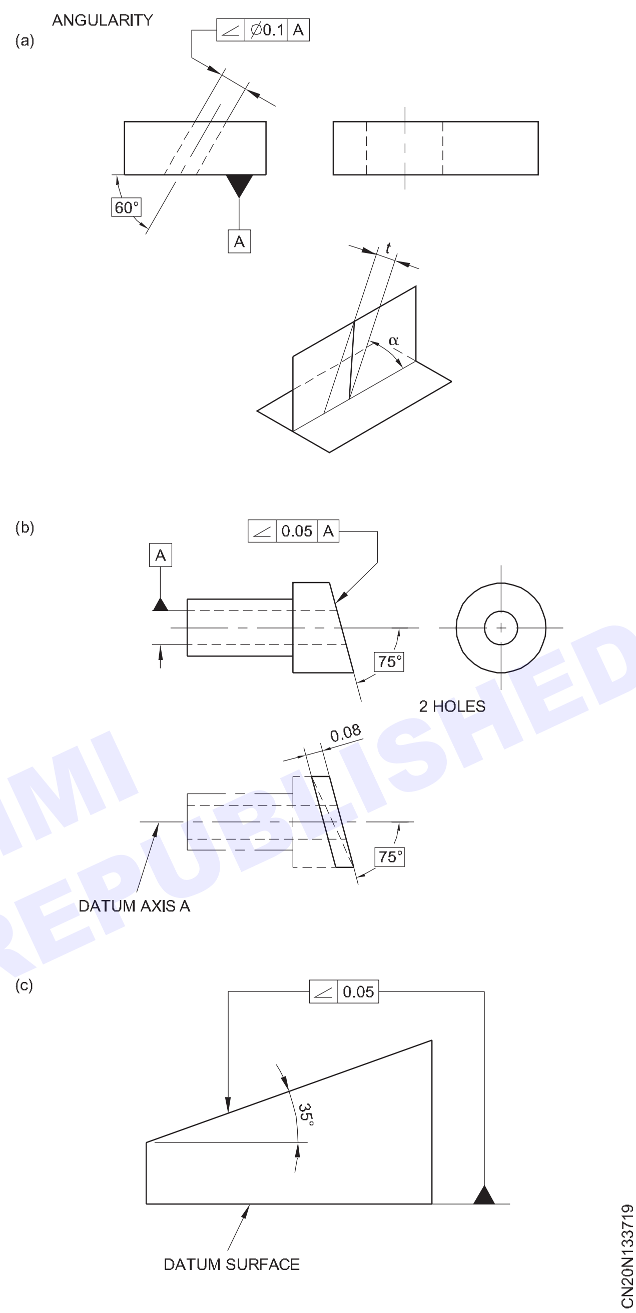
The axis of the upper cylindrical portion must lie within a cylindrical tolerance zone of 0.1 diameter, the axis of which is perpendicular to the datum axis 'A'. (8b)

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 a leader from the frame indicates the datum surface. 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 shown in Fig (9a), (9b) and (9c).

Fig 9



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 in box.

Location

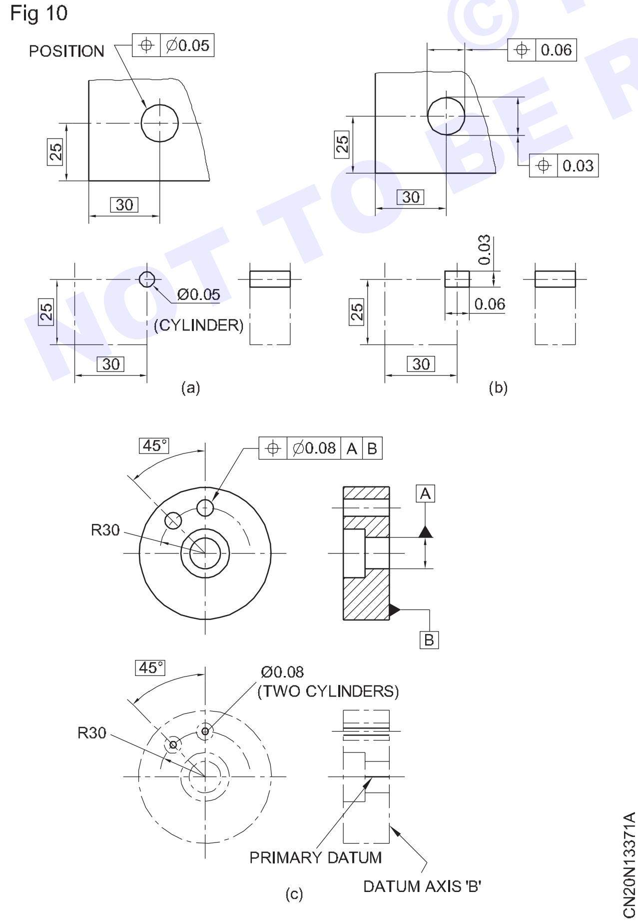
Tolerances of location are identified and indicated by the use of symbols for the following characteristics.

Characteristics	Symbol
Position	⌖
Concentricity	◎
Symmetry	⌯

Fig (10), (11), (12) show typical examples of these characteristics and symbols. In Fig (10a) the hole. Center dimensions of 25 and 30 are boxed to show that these are the theoretical dimensions. The geometrical tolerance requires that the hole center 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 shown with their true position spaced at 45° on a 30 mm pitch circle radius.

Fig 10

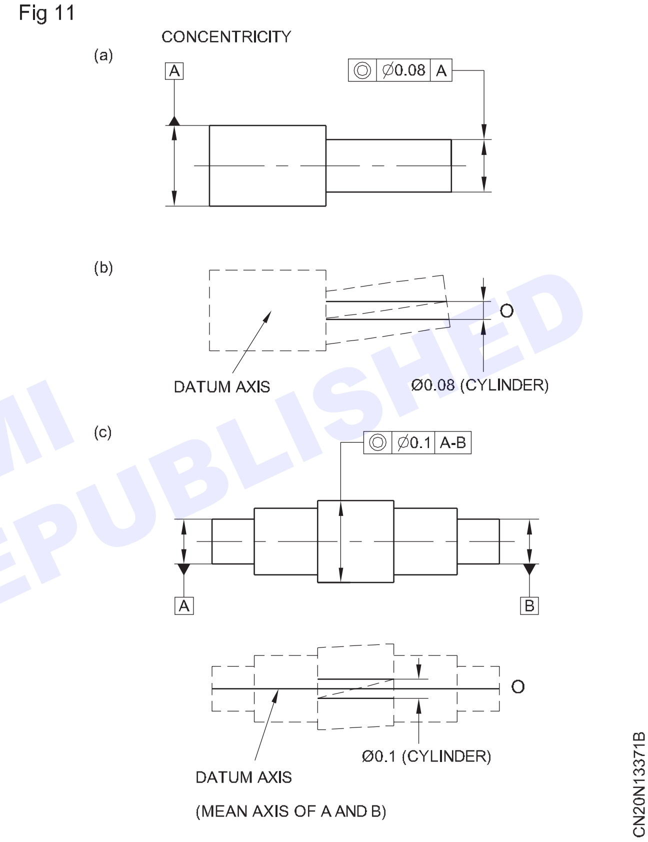


The Geometrical tolerance shows that each actual hole center. Must lie within a cylindrical zone 0.08 diameter whose axis lies at the theoretically exact true center position. The tolerance cylinders are disposed relative to the two datum features, namely the axis of the smaller bore and the right hand end face. The datum letters are included in the tolerance frame.

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

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'.

Fig 11



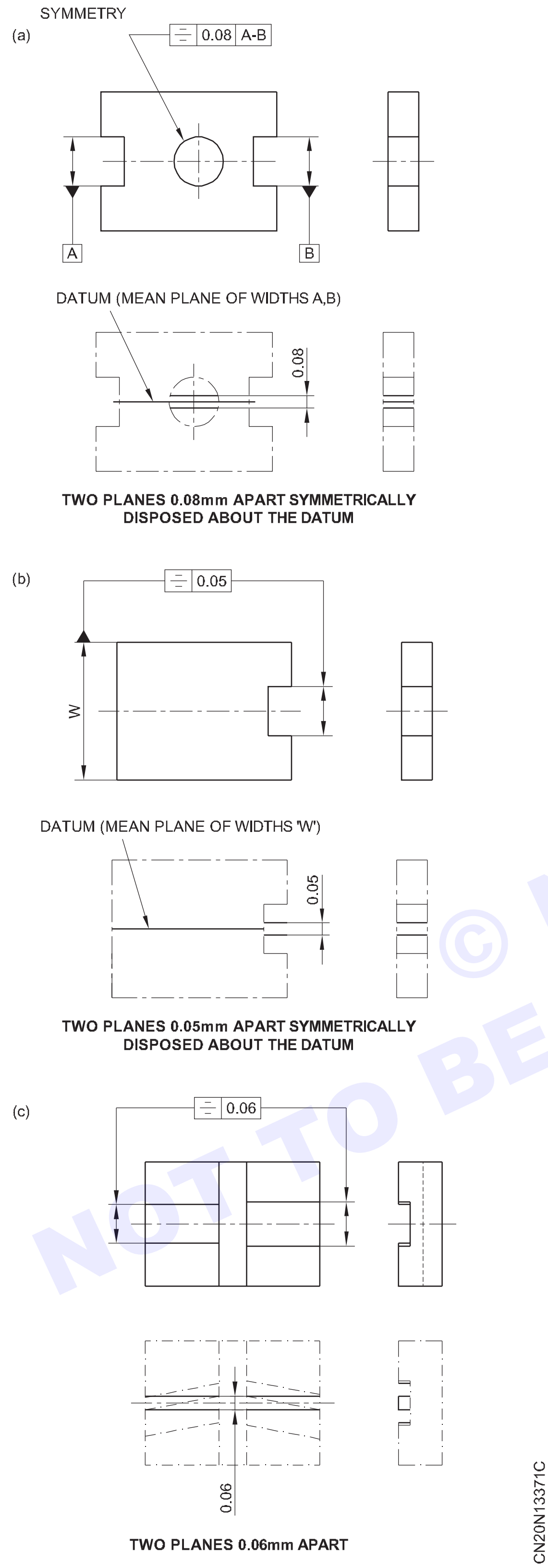
In Fig (11b), the axis of two end portions must lie within a cylindrical tolerance zone 0.08 dia.

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' 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'

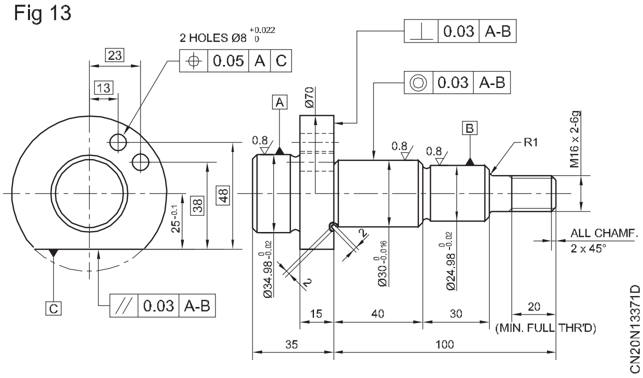
Fig 12



- 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 13



Machining procedure from raw material to finished product

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**
- **analyze the production drawing**
- **explain the process chart, flow chart**
- **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.

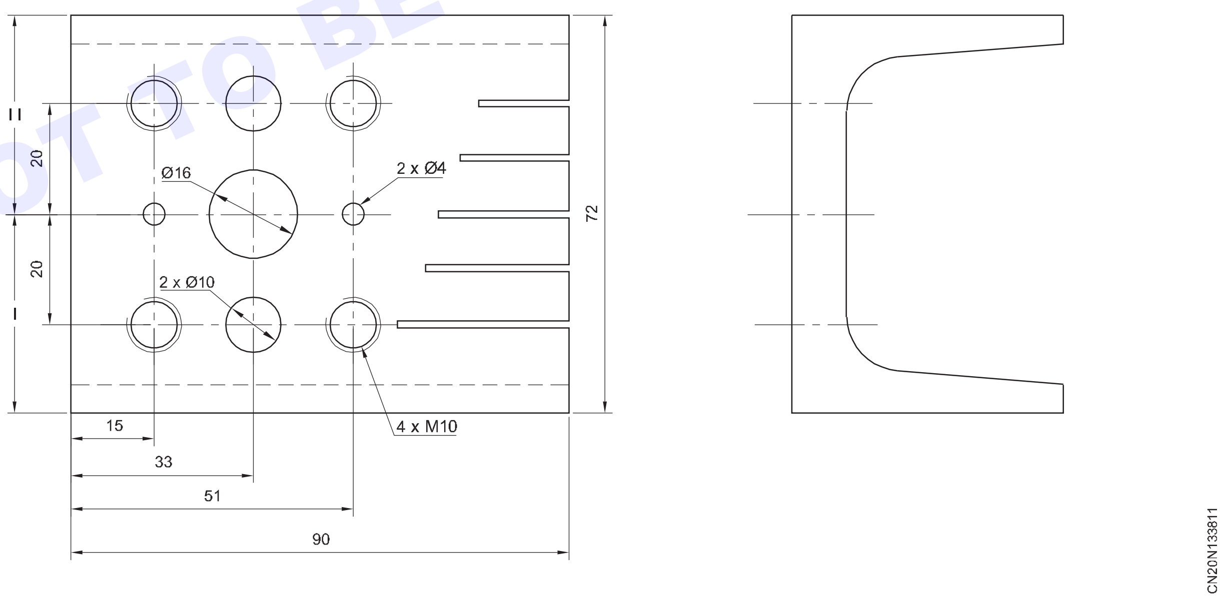
Product drawing

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.

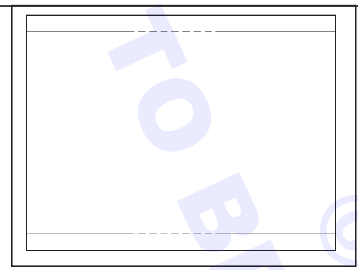
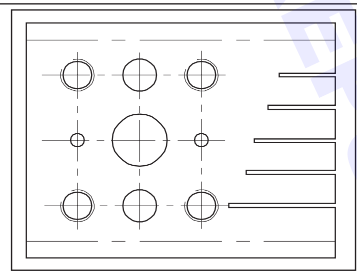
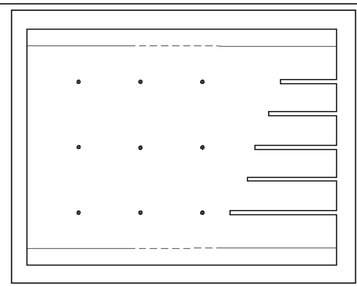
Fig 1

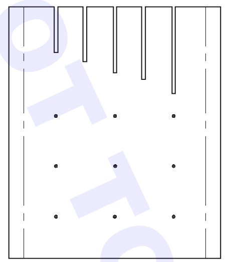
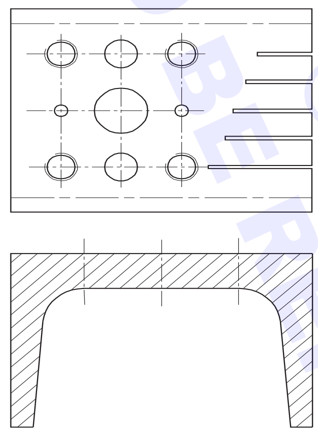
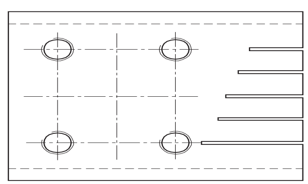


Process Chart

Product detail **Name fitting exercise** **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	Maintaining overall size of channel 90x72		Filing	Vice 300mm Bastard file 300mm second cure file	Steel rule		
010	Surface plate	Apply marking media		Marking block	Cellulose			
015		Mark hole centre and slot as per drawing		Marking	Vernier height gauge			
020	Work bench	Punching hole center and slot line		Punching	Punch steel plate center punch brick punch ball peen hammer			

OPN No. centre	Work description	Operation safety	Operation sketch	Operation	Tools	Instrument	Precaution	Remarks
025	Fitting	Swing slot to the required depth		Sawing	Vice machine frame	Steel rule		
030	Bench drill	Drill overall size of channel 90x72		Drilling	Drill file 300mm second cut file	Vernier caliper 150mm		
035	Fitting	Tapping 4 holes to M10		Tapping	Vice, tap and tap wrench	Tap M10		
040		Deburring		Deburring	File 150mm smooth			Check the tap hole by M10 bolt
045	Inspection table center and per drawing	Check all dimension as		Inspection	Try square	Vernier caliper 150mm		
050		Apply oil		Preserving	oil	Tap M10		To protect from rusting

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 in accordance with batch processing record manufacturing organization

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 ____________		Operation 1 ____________ Operation 2 ____________ Operation 3 ____________ Operation 4____________
2	Sizing of material Operation 1 ____________ Operation 2 ____________		

Flow chart: Induction monitoring and stages of production activity will be record 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	◯ → ▭ D ▽		
Machining	◯ → ▭ D ▽		
Filing	◯ → ▭ D ▽		
Inspection of finished job	◯ → ▭ D ▽		
To stores	◯ → ▭ D ▽		
	◯ → ▭ D ▽		

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

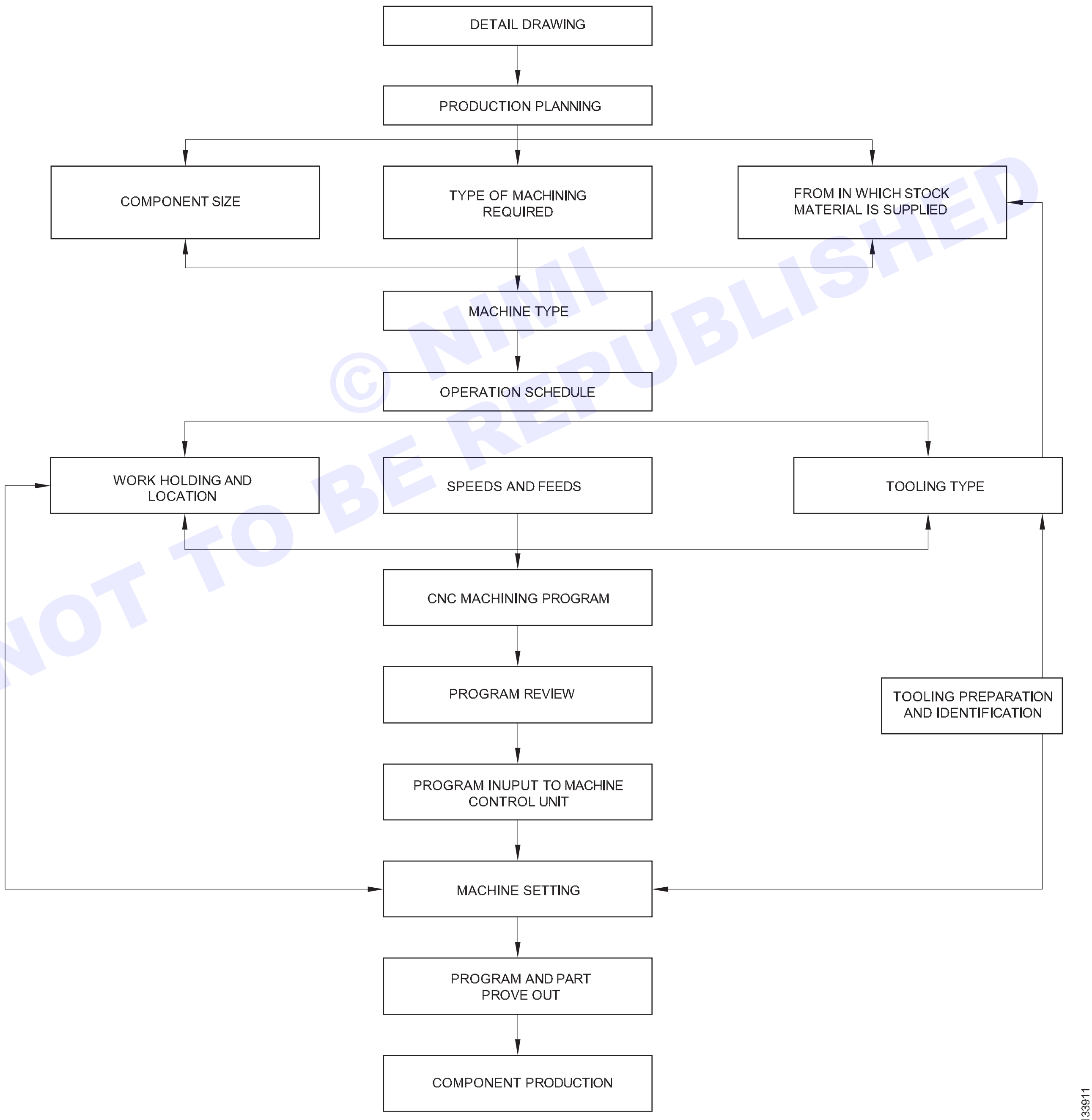
Concept of process flow of machining operation

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 backbone 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 workstation.

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 inspe ction are assembled, packaged, labeled and shipped to customer.

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

Types of process planning

The Types are

1 Manual process planning

2 Automated process planning

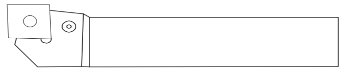
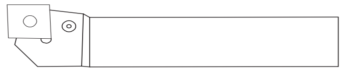
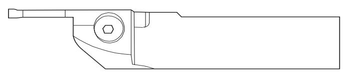
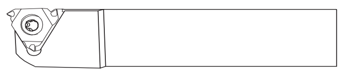
3 Generative process planning.

Machining sequence

Machining sequence defines the order of machining operations. Technical skill and machine shop experience does help in program planning, but some common sense approach is equally important. The sequence of machining must have a logical order - for example, drilling must be programmed before tapping, roughing operations before finishing, first operation, before second etc. Within 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 unnecessary 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

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 12404-Insert		180	1	0.2
3	OD grooving	Groove tool holder LH		150	3	0.15
4	OD threading	LH thread holder 25x25x150mm length DEG., DEPTH 3.0, LH		100	0.2	Pitch 1.5

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

Typical turning - sequence (facing, rough turning, 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

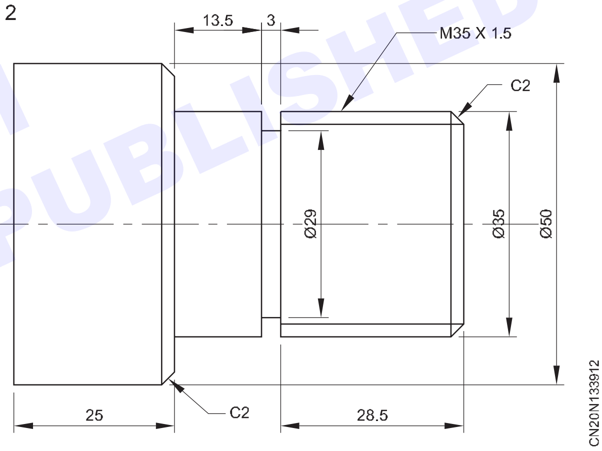
The tool layout for a job constitutes the predetermined plan for machining operation of a particular component the layout is dependent upon the number of pieces to be manufactured in lot size.

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



Concept work-piece holding

Objectives: At the end of this lesson you shall be able to
- **understanding different types of holding devices**
- **understanding working of holding devices.**
- **working of hydraulic chucks and their parts.**
- **different types of chucks used in CNC machines for holding work-piece.**

The Three- jaw chuck (Fig 1)

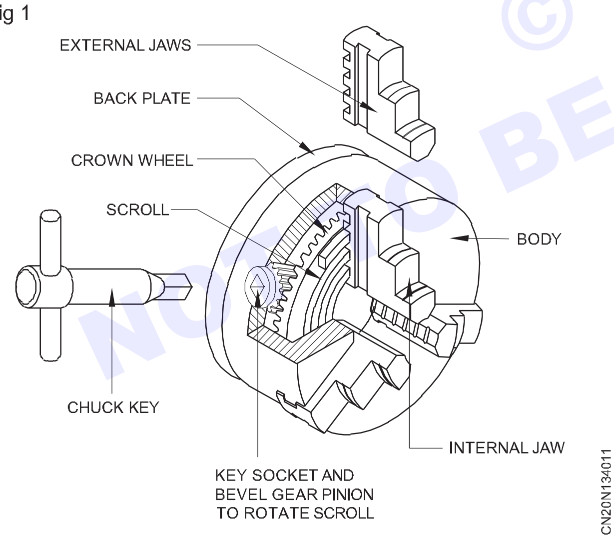
The three-jaw chuck is also known as self-centering chuck. The majority of the chucks have two sets of jaws for holding internal and external diameters. Only perfectly round work, or work with equally spaced flats, divisible by three, should be held in a three-jaw chuck.

The construction of a three-jaw chuck shows that the scroll not only clamps a component in place but also locates the component. This is fundamentally a bad practice, since any wear in the scroll and / or the jaws impairs the accuracy of location. Further, there is no means of adjustment possible to compensate for this wear.

The jaws of this type of chuck are not reversible, and separate internal and external jaws have to be used.

Parts of a three jaw chuck (Fig 1)

Fig 1



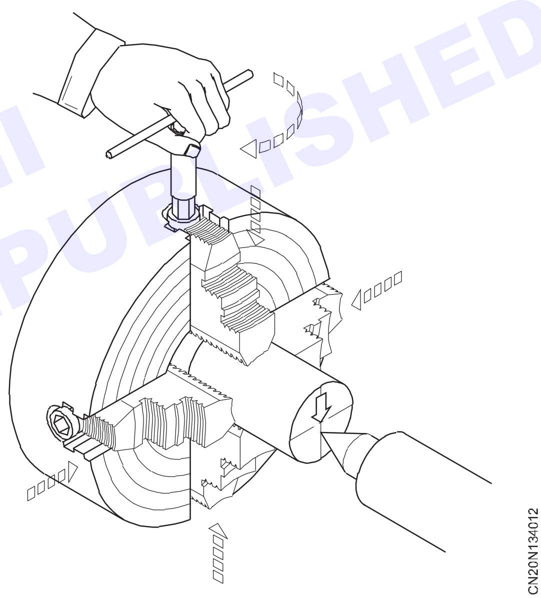
- Back plate
- Body
- Jaws
- Crown wheel
- Pinion

Four-jaw chuck (Fig 2)

The four-jaw chuck is also known as an independent chuck, since each jaw can be adjusted independently; a work can be trued to within 0.001" or 0.02 mm accuracy, using this chuck.

This type of chuck is much more heavily constructed than the self-centering chuck, and has much greater holding power. A square thread screw moves each jaw independently. The jaws are reversible for holding large diameter jobs. The independent four-jaw chuck has four jaws each working independently of the others in its own slot in the chuck body and actuated by its own separate square threaded screw. By suitable adjustment of the jaws, a work piece can be set to run either true or eccentric with the machine center

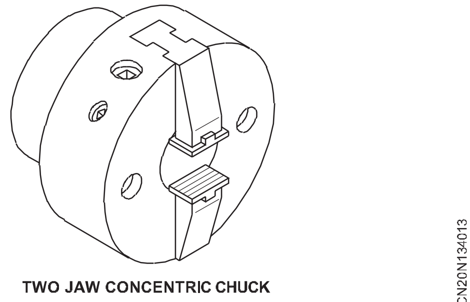
Fig 2



Two jaw concentric chuck (Fig 3)

The constructional features of this chuck are similar to those of three jaw and Four jaw chucks.

Fig 3



TWO JAW CONCENTRIC CHUCK

Each jaw is an adjustable jaw which can be operated independently. In addition to this feature, both jaws may be operated concentric to the center. Irregular shaped works can be held. The jaws may be specially machined to hold a particular type of job.

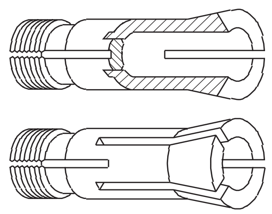
Collet chuck (Fig 4)

A collet is a hardened steel sleeve having slits cut partly along its length. It is held by a draw-bar which can be drawn in or out in the lathe spindle. The collet is guided in the collet sleeve, and held with the nose cap. It is possible to change the collet for different cross-sections depending on the cross-section of the raw material.

There are three most commonly used types of collet chucks.

- Push-out chucks
- Draw-in chucks
- Dead length bar chucks

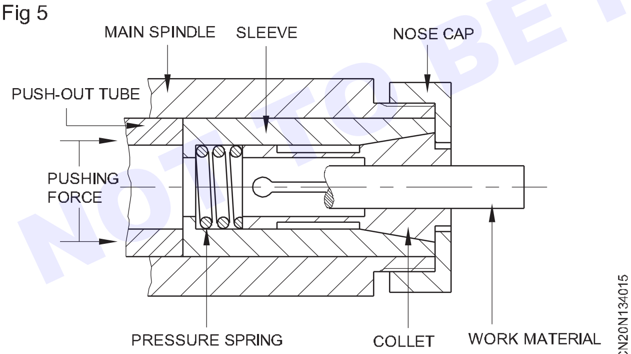
Fig 4



Push-out chucks (Fig 5)

The collet closes on the work-piece in a forward direction and consequently an end-wise movement of the work results. The cutting pressure tends to reduce the grip of the collet on the work-piece.

Fig 5



Draw-in chuck (Fig 6)

The collet closes on the work-piece in a backward direction and movement of the work. Take special care to avoid errors of length due to this fault. The cutting pressure increases the grip of the collet on the work-piece.

Magnetic Chuck (Fig 7)

This chuck designed to hold the job by means of magnetic force. The face of the chuck may be magnetized by inserting a key in the chuck and turning it to 180°. The amount of magnetic force may be controlled by reducing the angle of the key. The truing is done with a light magnetic force, and then the job is held firmly by using the full magnetic force.

Fig 6

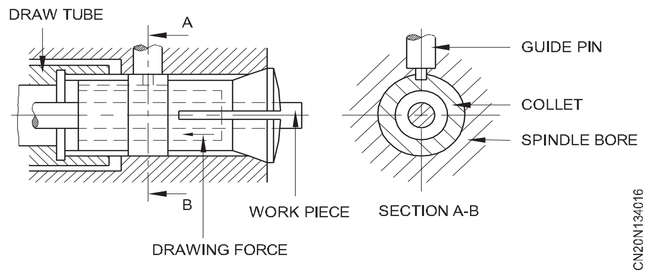
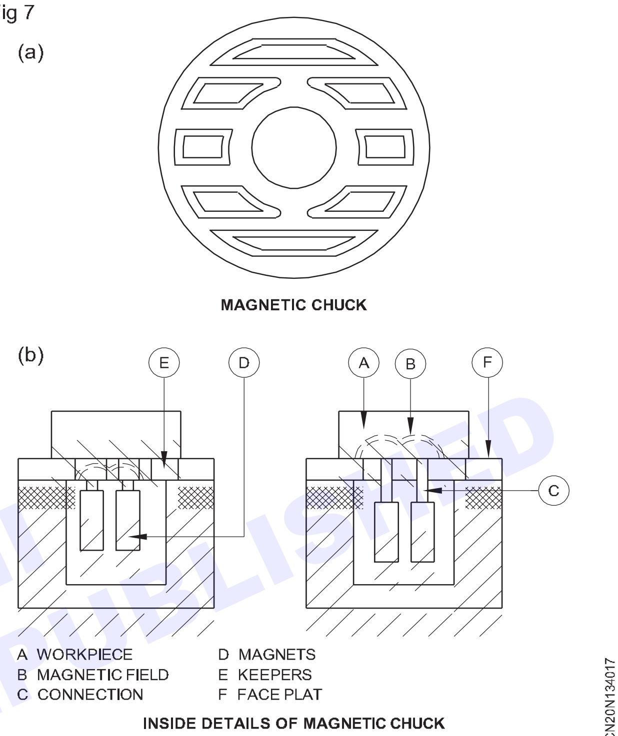
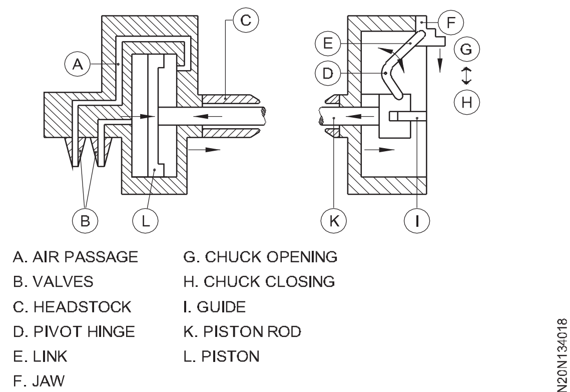


Fig 7



Hydraulic chuck or air-operated chuck (Fig 8)

Fig 8



These chucks are mainly used for getting a very effective grip over the job. This mechanism consists of a hydraulic or an air cylinder which is mounted at the rear end of the headstock spindle, rotating along with it. In the case of a hydraulically operated chuck the fluid pressure is transmitted to the cylinder by operating the valves. This mechanism may be operated manually or by power. The movement of the piston is transmitted to the jaws by means of connecting rods and links which enable them to provide a grip on the job.

Multi-stage engineering drawings

Objectives: At the end of this lesson you shall be able to
- **understand the multi-stage engineering drawings**
- **key aspects of multi-stage engineering drawings.**

Multi-stage engineering drawings

Multi-stage engineering drawings are detailed technical documents used to illustrate the various stages of a complex engineering project. These drawings are essential for visualizing, planning, and executing the design and manufacturing processes. Here is an overview of their key aspects

1 Sequential detailing

Multi-stage drawings divide a project into distinct phases, each concentrating on a specific stage involve in development of component or assembly. This step-by-step detailing aids in comprehending the evolution from the initial concept to the final product. For instance, if a part, such as a two- wheeler alloy wheel, is first manufactured through casting and then through CNC machining, production drawings for both stages will be created. These drawings will clearly outline the manufacturing criteria for each process

2 Component and assembly views

These drawings include detailed views of individual components as well as their assembly. This ensures that each part is accurately represented and can be correctly manufactured and assembled. Due to this there are several types of assembly drawings, including exploded view drawings, detailed assembly drawings, and schematic assembly drawings. Each type serves a specific purpose and provides diverse levels of detail and information about the assembly.

3 Dimensional accuracy

Precise dimensions and tolerances are specified for each stage to ensure that all parts fit together. correctly, maintaining the integrity and functionality of the final product. Components manufactured through various stages or processes will have varying geometric dimensions and tolerances. Consequently, separate drawings are released for each stage, and the machines or processes are configured accordingly. Tight tolerances, achievable in CNC machining but not in casting, necessitate different drawing details for each stage. Therefore, components passing through these stages will have distinct drawings released separately.

4 Material specifications

Information about the materials to be used at each stage is provided. This includes details on material properties, grades, and any special treatments required. Many time component which are manufactured through different stages will have raw material of different shape and tolerances, therefore at every stage the size and allowance of the raw material should be checked precisely before entering into next manufacturing stage.

5 Manufacturing instructions

Multi-stage drawings often include instructions for manufacturing processes, such as machining, welding, or assembly techniques. This ensures that the production team follows the correct procedures. Manufacturing instructions in multi-stage engineering drawings provide detailed guidance on the specific processes, tools, and sequences required at each stage of production. They ensure that each component is manufactured accurately and efficiently, specifying quality control measures, safety protocols, and material handling guidelines. These instructions help maintain consistency, achieve tight tolerances, and ensure that the final product meets all design and quality.

6 Quality control

Quality control drawings in multi-stage engineering projects are essential for ensuring that each component meets the required standards and specifications. These drawings provide detailed guidelines for inspections, including dimensions, tolerances, and material properties. They serve as a reference for quality control teams to verify that each stage of production adheres to the design requirements, helping to identify and correct any discrepancies early in the process. This ensures consistency, precision, and overall product quality throughout the manufacturing stages.

7 Iterative design process

The iterative nature of multi-stage engineering drawings enables continuous refinement and optimization throughout the manufacturing process. At each stage, feedback is gathered on the design, dimensions, and functionality of the components. This feedback is then used to make necessary adjustments and improvements before moving on to the next stage. By incorporating these iterative cycles, any issues or discrepancies can be identified and corrected early, ensuring that the final product meets all specifications and quality standards. This approach enhances efficiency, reduces errors, and leads to a more robust and well-designed final product.

By providing a comprehensive and detailed roadmap, multi-stage engineering drawings facilitate efficient project management, reduce errors, and ensure that the final product meets all design and quality requirements.

Benefits of multi-stage drawings

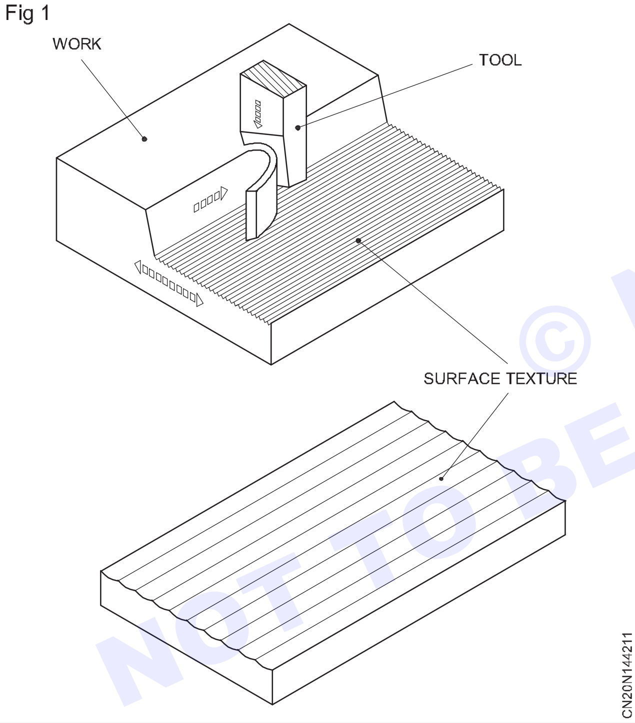
- Provides clear guidance to machinists and fabricators at each production step. Reduces confusion and errors in the production process.
- Facilities communication between design, manufacturing and quality teams. Helps vendors and subcontractors understand specific manufacturing steps.
- Enables efficient planning of operations and resource allocation. Reduces rework by ensuring accurate production at every stage.
- Supports stage-wise inspection and validation, reducing the likelihood of defects. Identifies potential issues early in the production process.

Surface finish and its importance

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

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

Fig 1



The components of surface texture

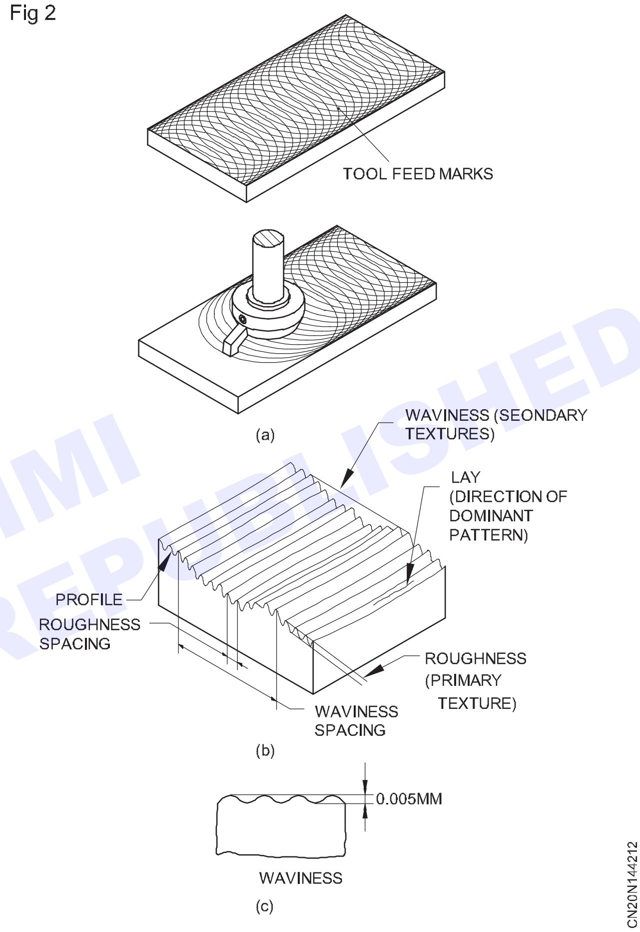
Roughness (Primary texture)

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

Waviness (Fig 2b & 2c)

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

Fig 2



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

Examples

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

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

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

Fig 3

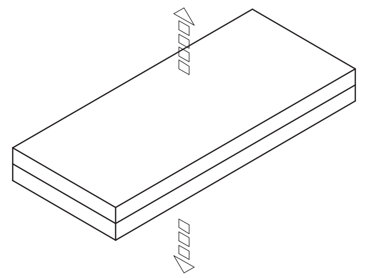
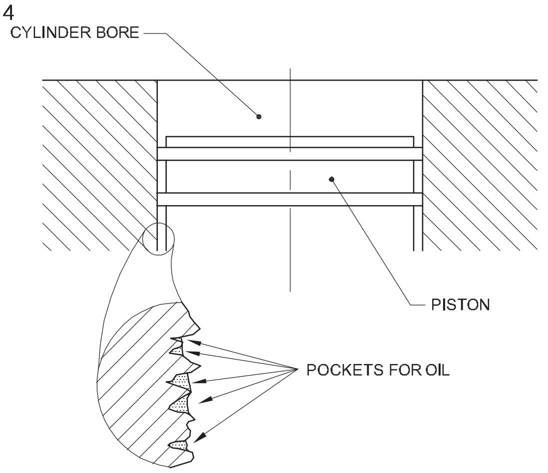
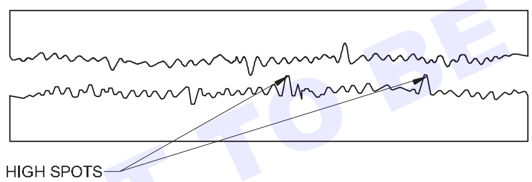


Fig 4



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

Fig 5



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

The surface texture quality can be expressed and assessed numerically.

'Ra' Values

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

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

Fig 6

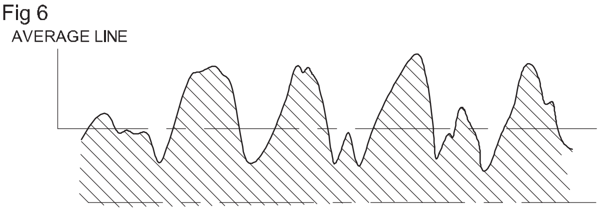
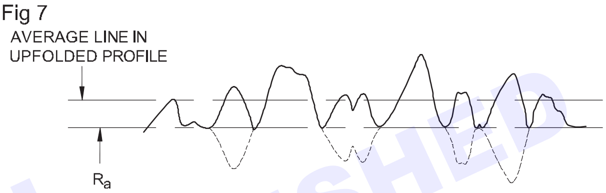


Fig 7



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

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

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

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

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

Surface texture measuring instruments

Objectives: At the end of this lesson you shall be able to
- **distinguish the features of mechanical and electronic surface indicators**
- **name the parts of a mechanical surface indicator**
- **identify the features of electronic surface indicators (talysurf)**
- **state the functions of the different features of electronic surface indicators.**

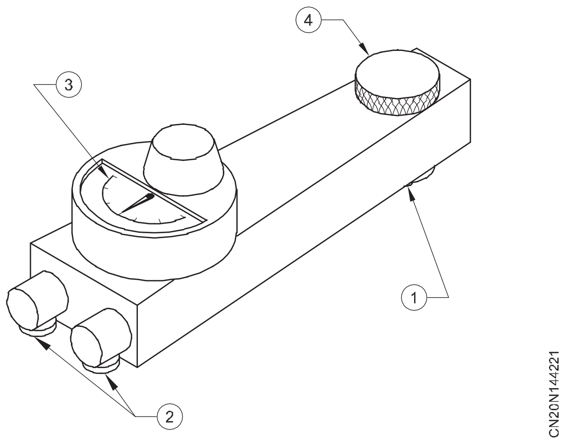
The use of surface finish standards which we have seen earlier is only a method of comparing and determining the quality of surface. The result of such measurement very much depends on the sense of touch and cannot be used when a higher degree of accuracy is needed.

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

Mechanical surface indicator

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

Fig 1



1 Measuring stylus

2 Skids

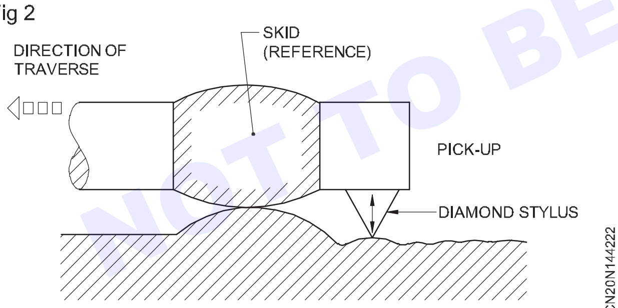
3 Indicator scale

4 Adjustment screw

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

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

Fig 2



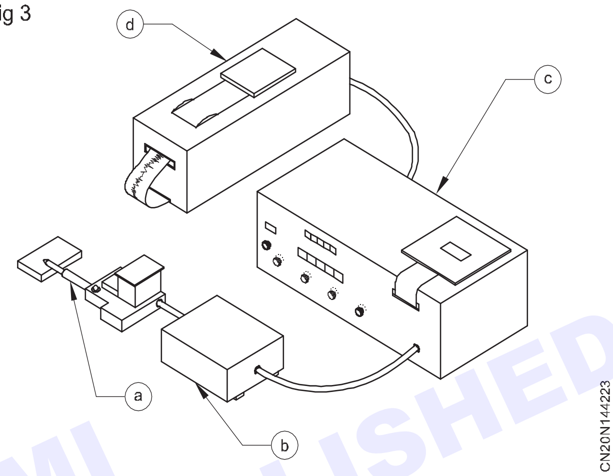
When using a mechanical surface indicator, measurement must be read as it is moved over the surface, and then a profile curve is drawn manually to compute the mean value.

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

Taly-surf (Electronic surface indicator)

This is an electronic instrument for measuring surface texture. This instrument can be used for factory and laboratory use. (Fig 3)

Fig 3



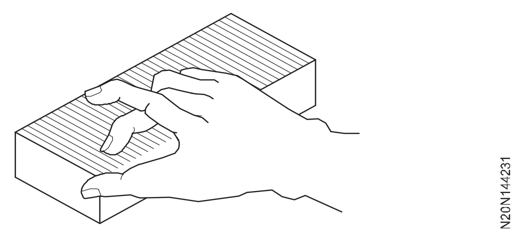
The measuring head of this unit consists of a stylus (a) and a motor race (b) which controls the movement of the instrument head across the surface. The movement of the stylus is converted to electrical signals. These signals are amplified in the surface analyser/amplifier (c) which calculates the surface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

Surface quality

Objectives: At the end of this lesson you shall be able to
- **identify the surface quality symbols.**

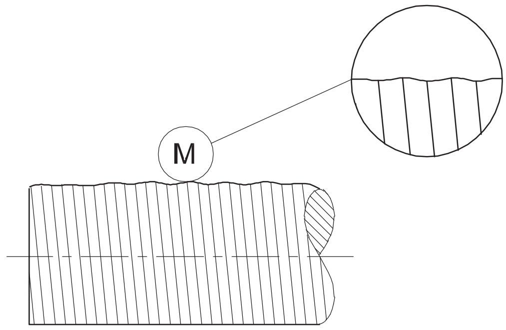
Various components are manufactured by different machining processes. The surfaces of the components differ in their appearance as well as `feel' when we move our hand over the surface. (Fig 1)

Fig 1



The surface will have ups and downs. These ups and downs are due to the tool marks. The pattern of these tool marks depends on the machining processes. The irregular patterns of tool marks depend on the feed, speed, tool angles, depth of cut etc. So all the machined surfaces are rough due to the inherent tool marks left in the machining processes. The surface appearance of components is shown in Figs 2 to 4.

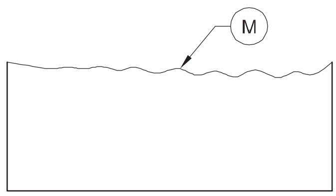
Fig 2



TOOL MARKS IN TURNING

CN20N144232

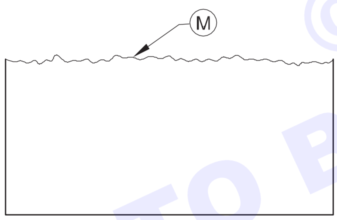
Fig 3



TOOL MARKS IN MILLING

CN20N144233

Fig 4



TOOL MARKS IN SHAPING. PLANING, SLOTTING

CN20N144234

In other words, the selection process and setting of machining parameters are dictated by the type of surfaces quality demanded in the drawing of the part.

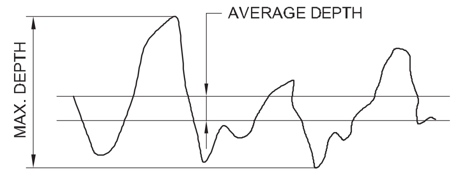
Surface roughness measurement

To control the roughness of a surface precisely, we need to define and establish a measuring system for it.

Roughness is defined as the average height or depth from the hill to the valley of a surface pattern (Fig 5) and it is possible to measure this by instruments specially designed for this purpose.

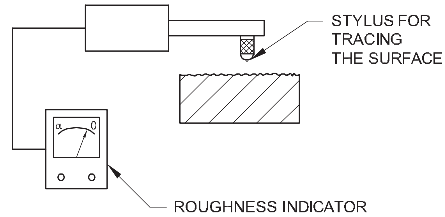
This instrument has a very sharp stylus. (Fig 6) This stylus is moved across the surface to be measured mechanically over a short distance and during this time the instrument calculates the average depth and displays the value as a roughness number.

Fig 5



CN20N144235

Fig 6

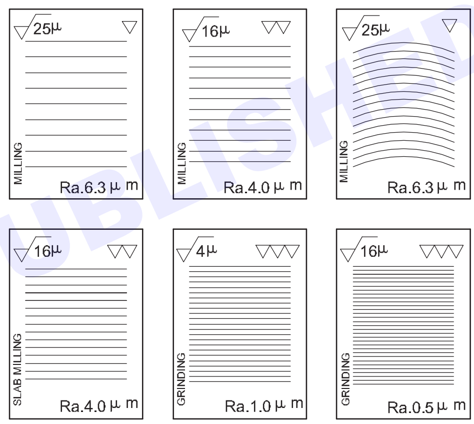


CN20N144236

Surface finish standard

One method of determining the surface roughness is by using a surface finish standard. (Fig 7) This is a box which consists of 20 blocks of a specific surface finish obtained by a specific machining operation.

Fig 7



CN20N144237

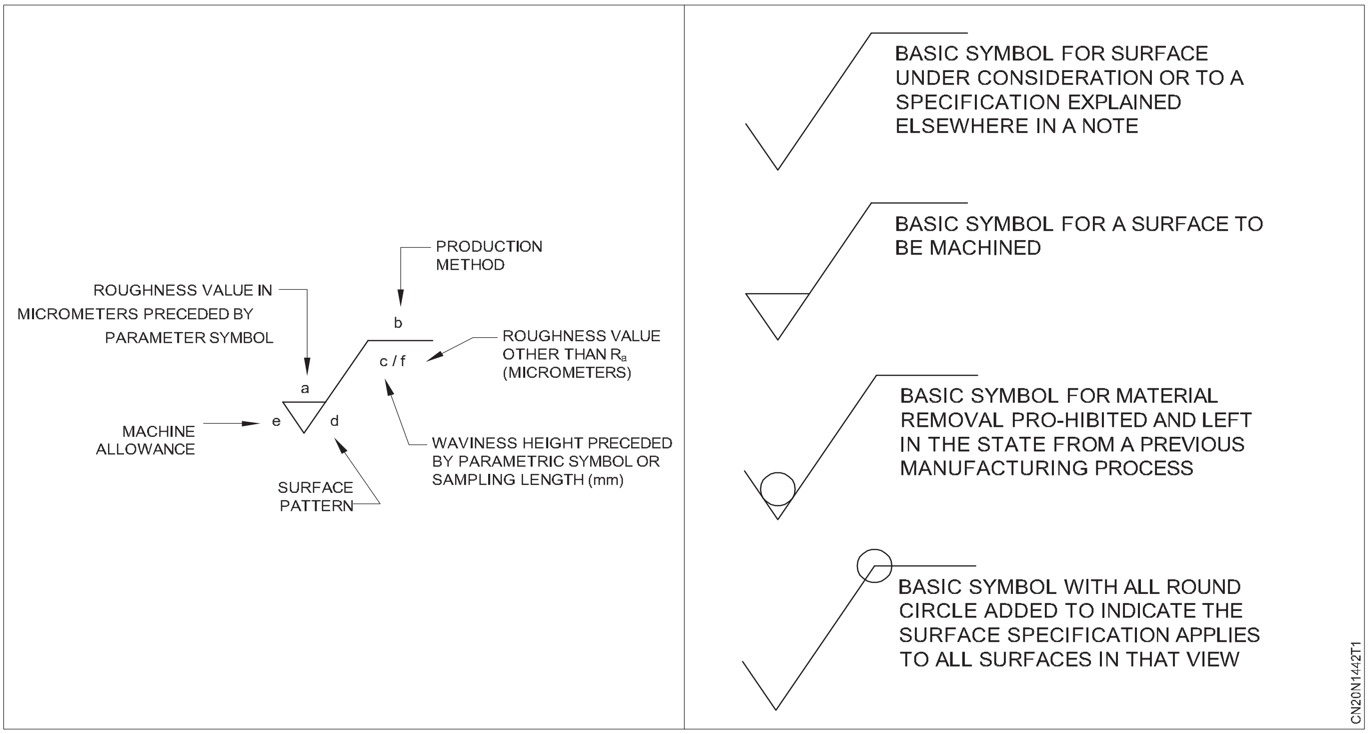
The type of machining operation is marked on each block together with the surface roughness number for height and width. Using the surface finish standard, we can make comparisons between the machined surface and the standard surface using our sense of touch.

However, this method is sometimes not accurate enough and the individual must be very sensitive to the different surface roughness.

If the degree of accuracy of checking is high, then the application of a sensitive instrument is inevitable.

In order to obtain the required surface quality, it is necessary to choose the appropriate manufacturing process. Table-1 appended here gives an idea about the different processes and range of surface quality attainable.

For more detailed information on surface texture, symbols and their representations refer to IS:10719.


PRODUCTION
METHOD
ROUGHNESS VALUE IN
MICROMETERS PRECEDED BY
PARAMETER SYMBOL
b
ROUGHNESS VALUE
OTHER THAN R_a
(MICROMETERS)
c / f
a
MACHINE
ALLOWANCE
e
d
WAVINESS HEIGHT PRECEDED
BY PARAMETRIC SYMBOL OR
SAMPLING LENGTH (mm)
SURFACE
PATTERN
BASIC SYMBOL FOR SURFACE
UNDER CONSIDERATION OR TO A
SPECIFICATION EXPLAINED
ELSEWHERE IN A NOTE
BASIC SYMBOL FOR A SURFACE TO
BE MACHINED
BASIC SYMBOL FOR MATERIAL
REMOVAL PRO-HIBITED AND LEFT
IN THE STATE FROM A PREVIOUS
MANUFACTURING PROCESS
BASIC SYMBOL WITH ALL ROUND
CIRCLE ADDED TO INDICATE THE
SURFACE SPECIFICATION APPLIES
TO ALL SURFACES IN THAT VIEW
CN20N1442T1

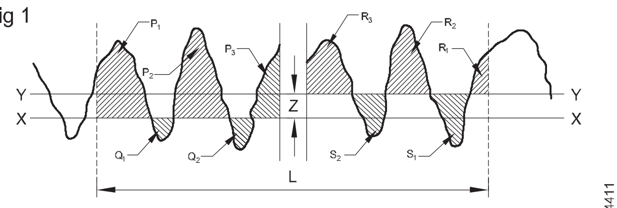
Surface finish calculation

Objectives: At the end of this lesson you shall be able to
- **find the Ra value using centre line average method**
- **evaluate using root mean square method**
- **calculate Rz value using peak-to-valley average method.**

Centre line average method (Fig 1)

The 'M-system' also known as center line average method (CLA) or mean line system, expresses the arithmetical average (AA) departure of the actual surface both above and below a mean line, within a specified length. According to Indian standard, the surface roughness is specified in terms of CLA values denoted by R_a, meaning average roughness height, in microns (μm).

Fig 1



TO DETERMINE Ra VALUES BY CENTRAL LINE AVERAGE METHOD (M-SYSTEM)

The central line is defined a s the line parallel to the general direction of the profile for which the areas embraced by the profile above and below the line are equal. The center line and the mean line may be considered to be equivalent for practical purposes.

a To determine the R_a values, it is necessary to first determine the mean line for a given profile. Any straight line XX is drawn (parallel to the general direction of the surface profile over the sampling the length L in Fig 1. The sum of all the areas p1, p2 etc. are then determined either by measuring ordinates or by the use of a planimeter, the line XX is then shifted by a distance Z such that

$$Z = \frac{\text{sum of area p - sum of the area q}}{L}$$

Now the line yy being required center line, sum of the areas r_1, r_2 etc. will be equal to sum of the areas s_1, s_2 etc. The arithmetic average is given by

$$R_a = \frac{\text{sum of the area r - sum of the area s}}{L} \times 1000/M$$

Where areas are expressed in sq.mm, L in mm and M, the vertical magnification of the record.

b If in a given profile of sampling length L, a center line, say AB is located such that the sum of the areas A_1, A_3 etc, above the line is equal to the areas A_2, A_4 etc.m below this line, then

$$R_a = \frac{A_1 + A_2 + A_3}{L} = \sum A/L$$

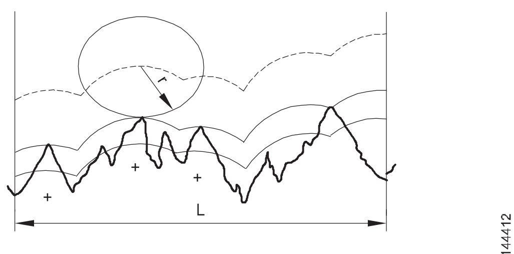
c If n measurement are made vertically without considering plus or minus from the center line on the profile in the Fig 2 and are called y_i, then

$$R_a = \frac{y_1 + y_2 + y_3 + \ldots\ldots + y_n}{n} = \frac{\sum_1^n |y_i|}{n}$$

$$\text{Or,} \quad R_a = \frac{1}{l}\int_0^l |y| dx$$

d The 'E-system' expresses the arithmetical average departure of a surface both above and below a mean curve. This is developed from what is known as a ' contacting envelope' by displacing it to a position where the areas enclosed by the profile above and below the mean curve are equal. This envelop or curve is obtained by rolling a circle of radius 'r', which is normally 25 mm, across the surface. It touches the peaks of the surfaces and is parallel to the locus by the centre of this circle as it rolls over the surface as illustrated in Fig 2.

Fig 2



TO DETERMINE R_a VALUES BY 'CONTACTING ENVELOP' METHOD (E-SYSTEM)

Root mean square method

In this method, the rms average value is obtained by setting many equidistant ordinates on the mean line as in Fig 3b and taking the root of the mean of the squared ordinates as

$$rms = \sqrt{\frac{y_1^2 + y_2^2 + y_n^2}{n}} = \left[\frac{\Sigma\left|y_i^2\right|}{n}\right]^{1/2}$$

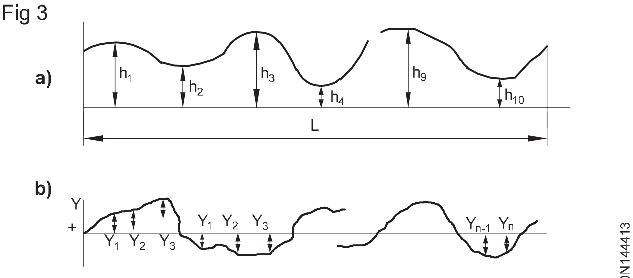
For a given profile of the same sampling length, the rms values are about 11% greater than R_a values.

Peak-to-valley average method

The PVA value is obtained as the average deference between (say) five highest peaks and equal number of deepest valleys over the sampling length measured parallel to the mean line and not crossing the profile as shown in Fig 3a. This is denoted by R_z

$$R_Z = \frac{h_1 + h_3 + ...+) - (h_2 + h_4 + ... + h_{10}}{5}\ \mu m$$

Fig 3



TO DETERMINE Ra VALUES BY (a) PVA METHOD, (b) RMS METHOD

Roughness values

The preferred values for R_d and R_z shall be selected from the following:

R_d 0.025, 0.05, 0.1, 0.2, 0.4, 0.8, 1.6, 3.2, 6.3, 12.5, 25

R_z 0.05, 0.1, 0.2, 0.4, 0.8, 1.6, 3.2, 6.3, 12.5, 25, 50, 100

Surface roughness statements

If a single R_a value is stated it means that R_a value is from zero to that is stated. When both minimum and maximum R_a values needed to specify these shall be expressed as :

$R_{a\ 16.0}^{\ 8.0}$ or alternatively R_a 8.0-16.0

The sampling length shall be indicated in parenthesis following the roughness value as : R_a 8.0(2.5)

Sampling length

This is also called the 'Instrument cut-off' in regard to measuring instruments. The value of sampling length L shall be selected from the following series

0.25, 0.8, 2.5, 8, 10 and 25 mm.

For machining process 0.25, 0.8 and 2.5mm and for non-machined surfaces 0.8 and 2.5 mm lengths are suitable.

Post process manufacturing operation to improve surface finish quality

Objectives: At the end of this lesson you shall be able to
- **state the introduction to improve surface finish quality**
- **state the features of lapping**
- **state the features of honing.**

Introduction to improve surface finish quality

In a manufacturing plant, a product may be shaped, turned, milled or drilled and left in that condition as being satisfactory for use. However, if a better finish is desired, for looks, for accuracy, for wearing qualities or for any other reasons, one of the micro finishes that include lapping, honing, super finishing, polishing, buffing may be employed. In some cases, other operations are done only to get durable finishes.

Post process manufacturing operations

- Scrapping
- Lapping
- Honing
- Reaming

Scrapping: Scrapping is used to produce a high degree of fit between two flat or two curved surfaces particularly where the surfaces can be rub together in use.

After a surface is filed or machined as accurately as possible, it can be further improved by rough scraping after which finish scrapping is employed. Accurate scrapping is used only to remove minute amounts of material.

It is essential that the cutting edge of a scraper be carefully tempered, uniform and free from defects. In addition, the cutting edge must be sharp, hard and durable. This requires careful grinding followed by honing on a suitable abrasive stone.

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

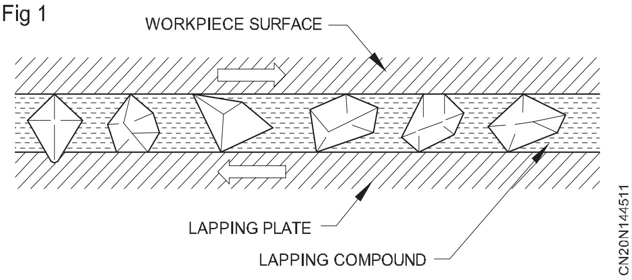
Purpose: This process

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

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

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

Fig 1



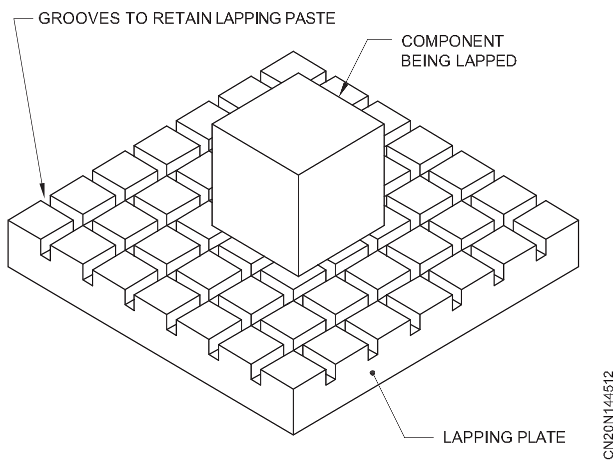
The lapping compound which is introduced between the workpiece and the lap chips away the material from the workpiece. Light pressure is applied when both are moved against each other. The lapping can be carried out manually or by machine.

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

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

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

Fig 2



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

This is a process by which the abrasive particles are embedded on to the surfaces of the laps which are comparatively softer than the component being lapped.

For charging the cast iron lap, apply a thin coating of the abrasive compound over the surface of the lapping plate.

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

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

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

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

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

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

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

Honing

Honing is a super finishing process carried out using abrasive sticks for the removal of stock from metallic and non-metallic surfaces.

This process:

- produces high surface finish
- corrects the profiles of cylindrical surfaces
- removes taper.

Working principle

The honing tool with abrasives mounted on it is held on the spindle of a machine which can be rotated in its axis.

As the spindle rotates, a reciprocating motion is also given to the tool. The surface produced will have a cross hatched pattern. (Figs 1 & 2) This pattern of the surface texture provides better lubrication in cylindrical bores.

Application

Honing is used for finishing of bores in ferrous and non-ferrous materials.

Honing can be done in hardened or un-hardened state.

Bores of any size, length, blind or through, tandem or interrupted surfaces can also be honed.

Honing can be carried out on drilling or other machines which have arrangement for rotary and reciprocating motion simultaneously.

A rotary motion can be given by the spindle and the reciprocating motion can be either manual or by power depending on the type of machine used.

For mass production special honing machines are used.

Methods of honing

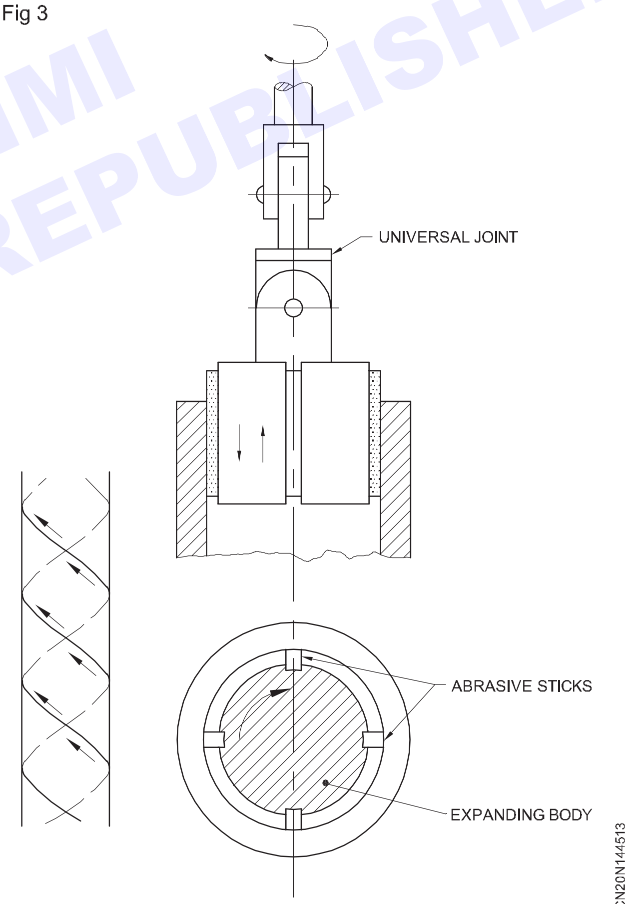
Manual stroking/Power stroking

Manual stroking is preferred for large quantities when tolerances are extremely close.

Many operators prefer this because of the flexibility in operation.

This eliminates the use of expensive fixtures to hold the work. (Fig 3)

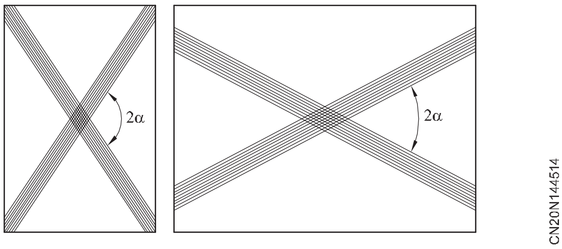
Fig 3



Jobs can be quickly changed from one type to another.

Jobs can be reversed from end to end for accurate honing and correction. The stroke length can be altered depending on the actual requirement of the individual workpiece. (Fig 4)

Fig 4



Power stroking is used for honing all types of workpieces. Power stroking may prove to be economical particularly in the case of small parts.

Reaming

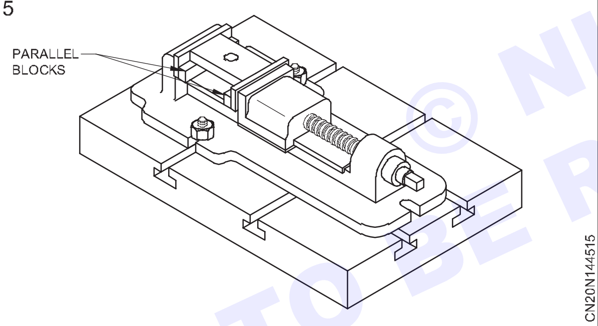
Reaming is the operation of finishing and sizing a hole which has been previously drilled, bored, casted holes. The tool used is called a learner, which has multiple cutting edges. Manually it is held in a tap wrench and reamed. Machine reamer are used in drilling machine using sleeves (or) socked. Normally the speed for reaming will be 1/3rd speed of drilling.

Hand reaming

Drill holes for reaming as per the sizes determined.

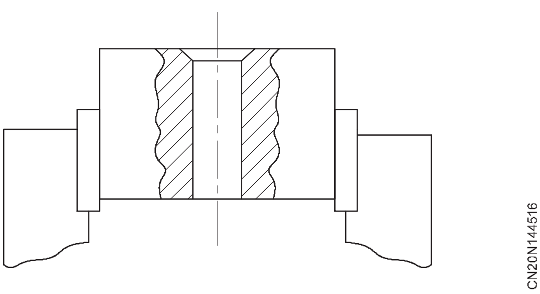
Place the work on parallels while setting on the machine vice. (Fig 5)

Fig 5



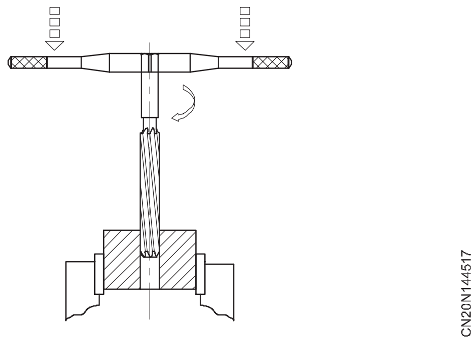
Chamfer the hole ends slightly. This removes burrs and will also help to align the reamer vertically. Fix the work in the bench vice. Use vice clamps to protect the finished surfaces. Ensure that the job is horizontal. (Fig 6)

Fig 6



Fix the tap wrench on the square end and place the reamer vertically in the hole. Check the alignment with a try square. Make corrections, if necessary. Turn the tap wrench in a clockwise direction applying a slight downward pressure at the same time. Apply pressure evenly at both ends of the tap wrench. (Fig 7)

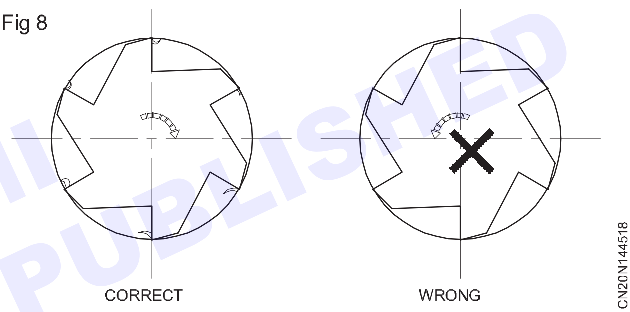
Fig 7



Apply cutting force: Turn the tap wrench steadily and slowly, maintaining the downward pressure.

Do not turn in reverse direction it will scratch the reamed hole. (Fig 8)

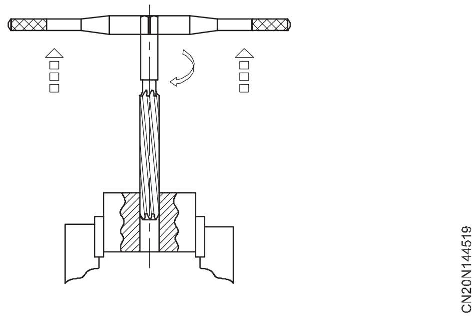
Fig 8



Ream the hole through, ensure that the taper lead length of the reamer comes out well and clear from the bottom of the work. Do not allow the end of the reamer to strike on the vice.

Remove the reamer with an upward pull until the reamer is clear of the hole. (Fig 9)

Fig 9



Remove the burrs from the bottom of the reamed hole.

Clean the hole. Check the accuracy with the cylindrical pins supplied.

Introduction to quality of a product

Objectives: At the end of this lesson you shall be able to
- **state what is quality of a product**
- **define and explain the product quality and its important**
- **explain the concept of quality control and quality assurance of product**
- **brief the purpose of inspection of instruments and type of inspection instruments**
- **list the importance calibration of inspection instruments**
- **explain standard guideline to minimize the human error in instruments handling**
- **state the properties of gauges.**

Quality of a product

Quality is a fundamental concept in manufacturing, service industries, and everyday life. It refers to the degree to which a product or service satisfies customer requirements, expectations, and specifications. The quality of a product is a key determinant of customer satisfaction, brand reputation, and market competitiveness.

Definition of product quality

Product quality can be defined as

1 "The totality of features and characteristics of a product that bear on its ability to satisfy stated or implied needs." - ISO 9000.

2 "Fitness for use" - Joseph Juran.

3 "Conformance to requirements" - Philip Crosby.

Aspects of product quality

1 **Performance:** How well the product performs its intended function.

2 **Reliability:** The consistency of performance over time without failure.

3 **Durability:** The product's lifespan under normal use.

4 **Aesthetics:** The product's appearance, design, or sensory appeal.

5 **Features:** Additional attributes or capabilities beyond basic functionality.

6 **Conformance:** Adherence to specifications, standards, or regulations.

7 **Serviceability:** Ease of repair or maintenance when required.

8 **Perceived quality:** The customer's perception of the product, often influenced by branding or reviews.

Importance of product quality

1 **Customer satisfaction:** High-quality products meet or exceed customer expectations, leading to greater satisfaction and loyalty.

2 **Market competitiveness:** Quality differentiates a product from competitors and builds a strong market presence.

3 **Brand reputation:** A reputation for quality strengthens trust and enhances a company's image.

4 **Cost efficiency:** Reducing defects, waste, and rework lowers production costs and improves profitability.

5 **Compliance:** High-quality products meet regulatory and safety standards, avoiding legal and financial penalties.

6 **Innovation:** A focus on quality drives continuous improvement and innovation.

Concept of quality control and quality assurance of product

The concepts of Quality Control (QC) and Quality Assurance (QA) are integral to ensuring products or services meet specified standards and fulfill customer expectations. While they are closely related, they focus on different aspects of the quality process. Here's a breakdown:

Quality Control (QC)

Definition

Quality Control is a reactive process that focuses on identifying and correcting defects in a product or service. It ensures that the output meets predefined quality standards through inspection, testing, and validation.

Features

- **Focus:** Detecting defects in the final product or during production.
- **Objective:** To ensure the product is free from defects and meets quality standards.
- **Methods:** Includes inspections, testing, sampling, and statistical quality control techniques.
- **Responsibility:** Typically performed by a dedicated QC team or department.

- **Examples**
  - Inspecting products on an assembly line for physical defects.
  - Testing software for bugs or errors before release.
  - Conducting batch sampling in pharmaceutical production.

Quality Assurance (QA)

Definition

Quality Assurance is a proactive, process-oriented approach aimed at preventing defects by improving the processes used to design and produce a product.

Features

- **Focus:** Building systems and processes to ensure quality.
- **Objective:** To prevent defects and ensure consistency in the production process.
- **Methods:** Includes process audits, training, documentation, and implementation of standards like ISO 9001 or Six Sigma.
- **Responsibility:** Usually involves all levels of the organization, especially management.
- **Examples**
  - Developing standard operating procedures (SOPs).
  - Training employees on quality standards.
  - Regular process audits and reviews to ensure compliance.

Introduction to inspection instruments

Inspection instruments are tools, devices, or equipment used to measure, examine, and evaluate the quality and specifications of a product during the manufacturing or quality control process. These instruments play a critical role in ensuring that products meet specified standards, tolerances, and customer requirements.

Purpose of inspection instruments

1 **Ensure product quality:** Verify that products meet design specifications, dimensions, and tolerances.

2 **Prevent defects:** Detect deviations or irregularities early in the production process.

3 **Compliance:** Ensure adherence to industry standards, certifications, or regulatory requirements.

4 **Improve efficiency:** Streamline the inspection process to maintain consistent quality.

Types of inspection instruments

Inspection instruments can be broadly categorized based on their application

1 Dimensional Inspection Instruments: Used to measure the physical dimensions of a product (e.g., length, width, height, diameter, etc.).

2 **Surface inspection instruments:** Used to evaluate the surface finish, texture, or flatness of a product.

3 **Optical inspection instruments:** Used to visually inspect or measure components using optical principles.

4 **Material inspection instruments:** Used to assess the physical, chemical, or mechanical properties of materials.

5 **Non-Destructive Testing (NDT) instruments:** Used to inspect components without damaging the product.

6 **Electrical inspection instruments:** Used to test the electrical properties of components or systems.

Selection criteria for inspection instruments

Choosing the right inspection instrument depends on several factors:

1 **Type of measurement:** Dimensional, surface, material, or electrical.

2 **Accuracy and Precision:** The level of detail required for the inspection.

3 **Application environment:** Laboratory, factory floor, or fieldwork.

4 **Ease of use:** Operator skill level and training.

5 **Cost and maintenance:** Budget constraints and instrument durability.

Importance of calibration of inspection instruments

Calibration is the process of comparing the measurements from an instrument to a known standard to ensure accuracy and reliability. Regular calibration of inspection instruments is crucial for maintaining consistent quality in production processes and ensuring that products meet required standards and specifications.

Importance of calibration

1 **Ensures measurement accuracy**

- Calibration verifies that the inspection instrument provides accurate readings within acceptable limits.
- It helps minimize errors in measurements, ensuring product specifications and tolerances are met.

2 **Maintains product quality**

- Accurate instruments lead to consistent inspection results, which ensure that manufactured products conform to quality standards.
- Reduces the risk of defective or non-compliant products reaching the customer.

3 **Compliance with standards and regulations**

- Many industries, such as aerospace, automotive, medical devices, and pharmaceuticals, have strict regulatory requirements for instrument calibration (e.g., ISO, ANSI, or FDA standards).

- Regular calibration ensures compliance with these standards and avoids potential legal or certification issues.

4 **Reduces downtime and costs**

- Calibrated instruments help detect deviations early, preventing large-scale production defects or rework.
- Reducing defective products saves material and labor costs, and ensures smooth operations.

5 **Increases reliability and trust**

- Properly calibrated instruments increase confidence in the inspection results.
- Customers and auditors trust the reliability of measurements, enhancing the organization's reputation.

6 **Enhances decision-making**

- Reliable measurements provide accurate data for quality assurance, troubleshooting, and process optimization.
- Decisions based on incorrect measurements can lead to flawed conclusions and ineffective actions.

7 **Prevents instrument drift**

- Over time, instruments may experience wear and tear or drift due to environmental factors, such as temperature or humidity.
- Calibration detects and corrects these deviations, maintaining the instrument's performance.

8 **Facilitates traceability**

- Calibration establishes a documented link between inspection measurements and international or national standards.
- Traceability ensures measurements can be independently verified and audited.

9 **Promotes safety**

- In industries where precision is critical (e.g., aviation, construction, healthcare), inaccurate measurements can lead to safety risks.
- Calibration ensures that inspection instruments provide dependable results, promoting operational safety.

10 **Supports continuous improvement**

- Calibration records help identify trends in instrument performance over time.
- This data can be used to improve processes and determine when instruments require repair or replacement.

Handling standard guidelines to minimize human error: Minimizing human error is critical in ensuring quality, safety, and efficiency in various processes. By implementing and adhering to standardized guidelines, organizations can systematically reduce the likelihood of mistakes. Below are strategies and practices based on standard guidelines to minimize human error

1 **Establish clear standard operating procedures (SOPs)**

- **Purpose:** SOPs provide step-by-step instructions for tasks to ensure consistency.
- **Implementation**
  - Write SOPs in simple, unambiguous language.
  - Use visual aids like flowcharts, diagrams, or pictures.
  - Regularly review and update SOPs to reflect process changes.
- **Benefit:** Reduces confusion and ensures that employees follow a consistent process.

2 **Conduct comprehensive training programs**

- **Purpose:** Equipe employees with the knowledge and skills needed to perform tasks correctly.
- **Implementation**
  - Provide hands-on and theoretical training for new employees.
  - Conduct refresher training sessions periodically.
  - Use simulations to practice handling critical scenarios.
- **Benefit:** Reduces errors caused by lack of knowledge or misunderstanding.

3 **Implement checklists and work instructions**

- **Purpose:** Checklists serve as reminders to ensure all steps are completed accurately.
- **Implementation**
  - Create task-specific checklists for complex or repetitive tasks.
  - Require employees to physically mark completed steps.
  - Digitize checklists for real-time tracking and accountability.
- **Benefit:** Prevents skipped steps and omissions.

4 **Design user-friendly work environments**

- **Purpose:** Optimize the workplace to reduce physical and mental strain on employees.
- **Implementation**
  - Arrange tools and materials ergonomically to reduce effort.
  - Minimize distractions, such as noise and clutter.
  - Use color coding and labeling for easy identification.

- **Benefit:** Reduces cognitive overload and fatigue, lowering the risk of mistakes.

5 **Automate where possible**

- **Purpose:** Automation reduces reliance on human intervention for repetitive or error-prone tasks.
- **Implementation**
  - Use technology like sensors, software, and robotics for precision tasks.
  - Incorporate automatic data collection and error detection systems.
- **Benefit:** Reduces human involvement in error-prone activities.

6 **Encourage a culture of accountability**

- **Purpose:** Empower employees to take ownership of their tasks and responsibilities.
- **Implementation**
  - Promote open communication where employees feel comfortable reporting issues.
  - Implement error-reporting systems without fear of punishment.
  - Recognize and reward adherence to guidelines.
- **Benefit:** Fosters a sense of responsibility and reduces intentional negligence.

7 **Use standardized tools and Equipment**

- **Purpose:** Ensure uniformity and compatibility across processes.
- **Implementation**
  - Use calibrated and certified tools regularly.
  - Establish guidelines for tool usage and maintenance.
  - Restrict unauthorized modifications to equipment.
- **Benefit:** Reduces variability caused by tool-related errors.

8 **Implement error-proofing techniques (Poka-yoke)**

- **Purpose**: Prevent errors by designing processes and systems to make mistakes impossible or detectable.
- **Implementation**
  - Use fixtures, jigs, or templates to ensure correct assembly.
  - Implement validation systems to check for incorrect inputs.
  - Introduce alarms or warnings for potential errors.
- **Benefit:** Catches and corrects errors before they escalate.

9 **Conduct regular audits and process reviews**

- **Purpose:** Identify gaps and weaknesses in processes that may lead to human error.
- **Implementation**
  - Perform internal and external audits.
  - Review error logs and employee feedback.
  - Use statistical tools to analyze process performance.
- **Benefit:** Identifies root causes of errors and provides data for continuous improvement.

10 **Manage workload and prevent fatigue**

- **Purpose:** Ensure employees are not overburdened, as fatigue increases error rates.
- **Implementation**
  - Limit working hours and provide adequate breaks.
  - Rotate tasks to avoid monotony.
  - Monitor workload to avoid overwhelming employees.
- **Benefit:** Enhances focus and reduces errors caused by tiredness.

11 **Apply visual management systems**

- **Purpose:** Provide clear and immediate visual cues for tasks and processes.
- **Implementation**
  - Use signs, labels, and color coding to guide actions.
  - Display dashboards for performance and error tracking.
  - Provide real-time feedback using screens or indicators.
- **Benefit:** Makes it easier for employees to follow correct procedures.

12 **Foster a learning organization**

- **Purpose:** Create an environment where continuous improvement is part of the culture.
- **Implementation**
  - Conduct root cause analysis (RCA) after errors to learn from mistakes.
  - Share lessons learned across teams.
  - Encourage innovation to improve processes and reduce complexity.
- **Benefit:** Builds resilience and reduces recurrence of errors.

Concept of inspection instruments

Inspection instruments are tools or devices used to measure, assess, or evaluate the physical properties, dimensions, or quality of a product to ensure it meets specified requirements. They are vital in quality control and manufacturing processes to maintain consistency, minimize defects, and ensure compliance with standards.

Properties of gauges

Gauges are a subset of inspection instruments specifically designed for measuring and assessing dimensions or tolerances. The properties of gauges determine their effectiveness in delivering accurate and reliable measurements. following are the key properties of gauges

1 **Accuracy**

- **Definition:** The degree to which a gauge's measurement matches the true or accepted standard value.

2 **Precision (Repeatability)**

- **Definition:** The ability of the gauge to consistently provide the same measurement under the same conditions.

3 **Sensitivity**

- **Definition:** The smallest change in the measured quantity that a gauge can detect.

4 **Range**

- **Definition:** The maximum and minimum limits within which a gauge can measure.

5 **Resolution**

- **Definition:** The smallest unit or increment that a gauge can display.

6 **Durability**

- **Definition:** The ability of the gauge to maintain its performance over time under working conditions.

7 **Stability**

- **Definition:** The ability of the gauge to maintain consistent performance over time without significant drift.

8 **Calibration**

- **Definition:** The process of aligning the gauge with a known standard to maintain accuracy.

9 **Ease of use**

- **Definition:** The simplicity and intuitiveness with which the gauge can be used.

10 **Material compatibility**

- **Definition:** The suitability of the gauge material for the component being inspected.

11 **Versatility**

- **Definition:** The ability of the gauge to perform multiple measurements or adapt to different applications.

12 **Traceability**

- **Definition:** The ability to trace measurements back to national or international standards.

13 **Environmental resistance**

- **Definition:** The gauge's ability to withstand adverse environmental conditions like temperature, moisture, or corrosion.

CNC toolings, holders & its types

Objectives: At the end of this lesson you shall be able to
- **knowledge on CNC cutting tools**
- **brief the other cutting tool materials**
- **describe the various shapes of inserts**
- **list the commonly used turning tools in CNC turning.**

Inserted carbide tooling is the preferred tooling for many CNC applications. For the full utilization of CNC machines, it is essential to pay due attention to the selection and usage of tooling,

Carbide tool

Carbide tools, especially indexable carbide tools, are the leading products of CNC machining tools. The varieties of integral and indexable carbide tools or inserts have been extended to various cutting tool fields, among which indexable carbide tools have been expanded from simple turning tools and face milling cutters to various precision, complex and forming tools

Types of carbide tools they are

- CVD coated carbide tipped tools
- PVD coated carbide tipped tool
- Cemented carbide tipped tools

CVD coated tipped tools

CVD stands for Chemical Vapour Deposition. The CVD coating is generated by chemical reactions at temperatures of 700-1050°C.

CVD coatings have high wear resistance and excellent adhesion to cemented carbide.

The first CVD coated cemented carbide was the single layer titanium carbide coating (TiC).

Alumina coatings (Al2O3) and Titanium Nitride (TiN) coatings were introduced later.

More recently, the modern titanium carbonitride coatings (MT-Ti(C,N) or MT-TiCN, also called MT-CVD) were developed to improve grade properties through their ability to keep the cemented carbide interface intact.

Applications

CVD coated grades are the first choice in a wide range of applications where wear resistance is important. Such applications are found in general turning and boring of steel, with crater wear resistance offered by the thick CVD coatings; general turning of stainless steels and for milling grades in ISO P, ISO M, ISO K. For drilling, CVD grades are usually used in the peripheral insert.

PVD coated tipped tools

Physical Vapour Deposition (PVD) coatings are formed at relatively low temperatures (400-600°C). The process involves the evaporation of a metal which reacts with, for example, nitrogen to form a hard nitride coating on the cutting tool surface.

The main PVD-coating constituents are described below. Modern coatings are combinations of these constituents in sequenced layers and/or lamellar coatings. Lamellar coatings have numerous thin layers, in the nanometer range, which make the coating even harder.

PVD-TiN - Titanium nitride was the first PVD coating. It has all-round properties and a golden color.

PVD-Ti(C,N) - Titanium carbonitride is harder than TiN and adds flank wear resistance.

PVD-(Ti,Al)N - Titanium aluminium nitride has high hardness in combination with oxidation resistance, which improves overall wear resistance.

PVD-oxide - Is used for its chemical inertness and enhanced crater wear resistance.

Applications: PVD coated grades are recommended for tough, yet sharp, cutting edges, as well as in smearing materials. Such applications are widespread and include all solid end mills and drills, and a majority of grades for grooving, threading and milling. PVD-coated grades are also extensively used for finishing applications.

Uncoated cemented carbide cutting tool material

Uncoated cemented carbide grades represent a very small proportion of the total cutting tool assortment. These grades are either straight WC/Co or have a high volume of cubic carbonitrides.

Applications: Typical applications of this cutting tool material are machining of HRSA (heat resistant super alloys) or titanium alloys and turning hardened materials at low speed.

The wear rate of uncoated cemented carbide grades is rapid yet controlled, with a self-sharpening action.

Cermet cutting tool material

Cermet is a cemented carbide with titanium-based hard particles. The name cermet combines the words ceramic and metal. Originally, cermet was a composite of TiC and nickel. Modern cermets are nickel-free and have a designed structure of titanium carbonitride Ti(C,N) core particles, a second hard phase of (Ti,Nb,W)(C,N) and a W-rich cobalt binder.

Ti(C,N) adds wear resistance to the grade, the second hard phase increases the plastic deformation resistance, and the amount of cobalt controls the toughness.

Applications

Cermet grades are used in smearing applications where built-up edge is a problem. Its self-sharpening wear pattern keeps cutting forces low even after long periods in cut. In finishing operations, this enables a long tool life and close tolerances, and results in shiny surfaces.

Typical applications are finishing in stainless steels, nodular cast irons, low carbon steels and ferritic steels.

Ceramic cutting tool material

All ceramic cutting tools have excellent wear resistance at high cutting speeds.

There are a range of ceramic grades available for a variety of applications.

Sialon (SiAlON) grades combine the strength of a self-reinforced silicon nitride network with enhanced chemical stability. Sialon grades are ideal for machining heat resistant super alloys (HRSA).

Applications

Ceramic grades can be applied in a broad range of applications and materials, most often in high speed turning operations but also in grooving and milling operations. The specific properties of each ceramic grade enable high productivity when applied correctly. Knowledge of when and how to use ceramic grades is important for success.

Polycrystalline cubic boron nitride cutting tool material

Polycrystalline cubic boron nitride, CBN, is a cutting tool material with excellent hot hardness that can be used at very high cutting speeds. It also exhibits good toughness and thermal shock resistance.

Modern CBN grades are ceramic composites with a CBN content of 40-65%. The ceramic binder adds wear resistance to the CBN, which is otherwise prone to chemical wear. Another group of grades are the high content CBN grades, with 85% to almost 100% CBN. These grades may have a metallic binder to improve their toughness.

Applications

CBN grades are largely used for finish turning of hardened steels, with a hardness over 45 HRc. Above 55 HRc, CBN is the only cutting tool which can replace traditionally used grinding methods. Softer steels, below 45 HRc, contain a higher amount of ferrite, which has a negative effect on the wear resistance of CBN. CBN can also be used for high speed roughing of grey cast irons in both turning and milling operations.

Polycrystalline diamond cutting tool material

PCD is a composite of diamond particles sintered together with a metallic binder. Diamond is the hardest, and therefore the most abrasion resistant, of all materials. As a cutting tool material, it has good wear resistance but it lacks chemical stability at high temperatures and dissolves easily in iron.

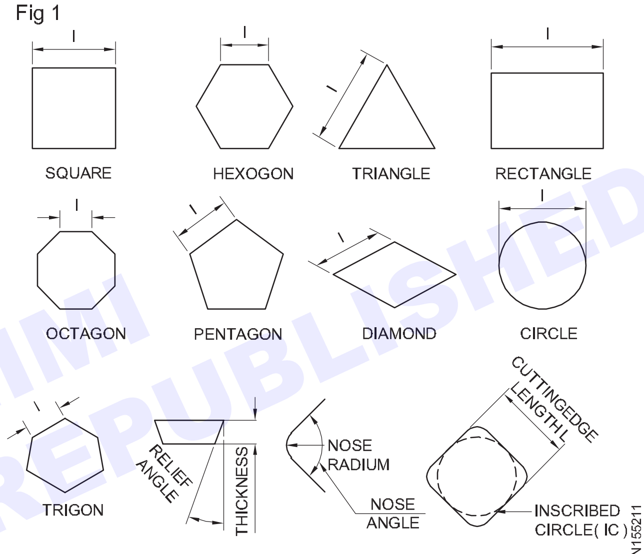
Applications

PCD tools are limited to non-ferrous materials, such as high-silicon aluminium, metal matrix composites (MMC) and carbon fibre reinforced plastics (CFRP). PCD with flood coolant can also be used in titanium super- finishing applications.

Tool inserts shapes

Inserts are available in various shapes such as triangle, square, rectangle, pentagon, hexagon, octagon, diamond shaped and circle. They cannot be resharpened, but they have a number of cutting edges. (Fig 1)

Fig 1



INDEXABLE TOOL INSERTS

Inserts are produced in various sizes and thicknesses. Smallest possible size is chosen to produce the desired depth of cut. Thickness of an insert affects its strength. Hence, for a large depth of cut and feed, a thicker insert is chosen.

ISO standard is commonly followed for specifying inserts. An example is CNMG120408. The first letter, C in this case, indicates the shape of the insert. The common types are

Symbol	Shape
S	Square
T	Triangular
H	Hexagonal
O	Octagonal
P	pentagonal
L	rectangular
R	round
A, B, K	parallelogram (nose angles 85°, 82° and 55° respectively)

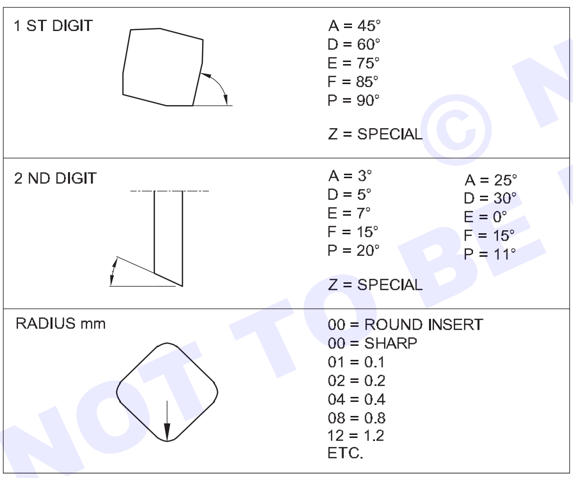
Symbol	Shape
C, D, E, F, M, V	Diamond shaped or rhombic (nose angles 80°, 55°, 75°, 50°, 86°, 35 ° respectively)
W	Trigon (nose angle 80°)

The second letter specifies the relief angles

Symbol	Relief angle
N	0°
A	3°
B	5°
C	7°
P	11°
D	15°
E	20°
F	25°
G	30°

The third letter specifies tolerances on various dimensions (Fig 2) (e.g., thickness) of the insert. The different tolerance classes are A, F, C, H, E, G (absolute values) and J, K, L, M, N, U (tolerance values depend on the diameter of the inscribed circle of the insert).

Fig 2



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 center. 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 3)**

F for sharp,

T for chambered, E for honed and

S for chambered and honed.

This information, however, is non-obligatory.

- **Cutting direction (Fig 4)**

L for machining with left - rotated (CCW) spindle (M04), R for machining with right - rotated (CW) spindle (M03) and N for both left-and right - rotated.

The appropriate character designations are appended to the right, after the radius specification.

Fig 3

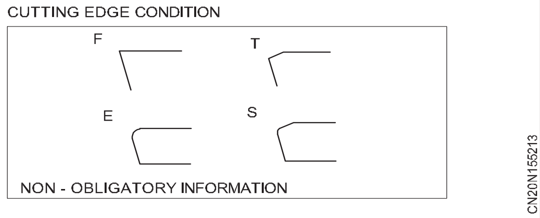
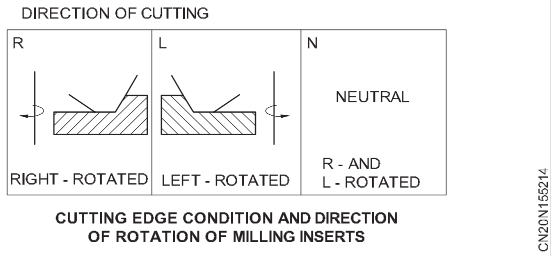


Fig 4



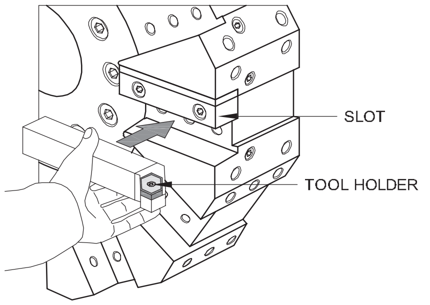
CUTTING EDGE CONDITION AND DIRECTION OF ROTATION OF MILLING INSERTS

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 T Q		cylindrical with 40°-60° counter sink	without
			single-sided
B		cylindrical with 40°-60° double countersink	without
H			double-sided
C		cylindrical with 70°-90° counter-sink	without
J			single-sided
X		cylindrical with 70°-90° double countersink	without double-sided
		special shape	

CNc tool holders (Turning): A normal tool is lamped by inserting it in a slot in the turret and tightening screws to hold it firmly (Fig 5). While doing this, care must be taken to seat the holder properly against available butting surfaces. Finally, the tool offsets must be taken by touching the part or taking skin cuts.

Rectangular or square type shank tools (Fig 6) are fitted in the tool holders (Fig 7) and the tool holders are fitted in turret.

Fig 5



NORMAL TOOLING

Fig 6

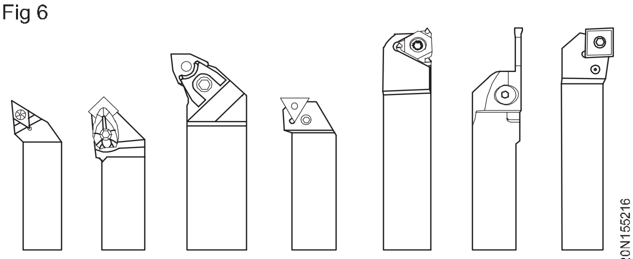


Fig 7

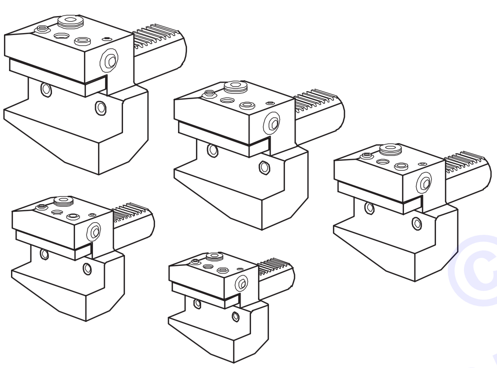
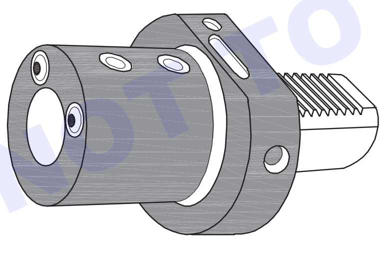


Fig 8



Whereas the round shank tools are fitted in round shank tool holders as shown in Fig 8 and the tool holders along with the tools are fitted in turret as shown in Fig 9.

Quick change tool holder systems on CNC lathes (Figs 10 & 11)

A quick-change tool holder, or quick-change tooling, is a modular system that enables tools to be changed very fast on the machine during machining.

The system reduces machine downtime for tool change during setup changes, and for changing worn out inserts during machining.

With a quick-change tool holder, you do not clamp the tool directly in the turret. The tool is replaced by an interchangeable cutting unit. You insert the cutting unit into a standardized locking unit. There are different locking units for internal and external tools, and the locking units remain on the turret forever - there is no need to keep removing them for every part. The interface between the units is such that the cutting unit will only sit in a particular position in the locking unit and will sit accurately every time.

Fig 9

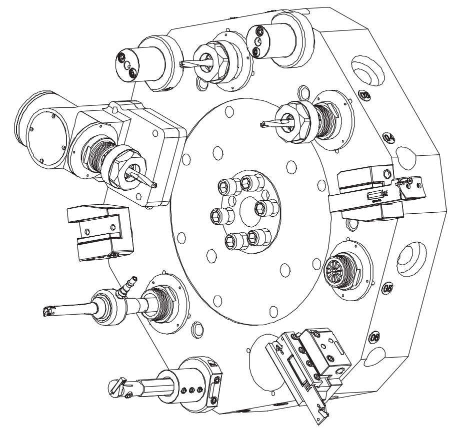


Fig 10

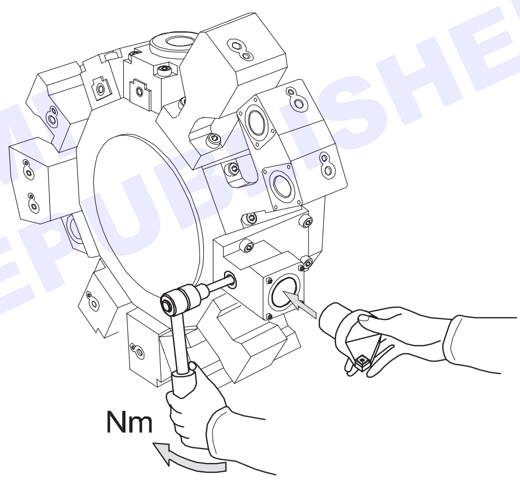
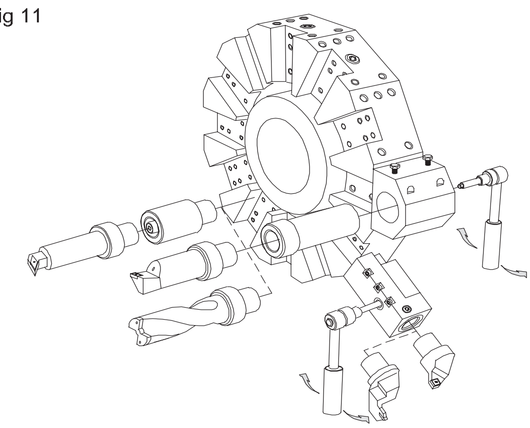


Fig 11



When doing the setup for a new part, you just quickly insert and clamp a new set of cutting units into the appropriate locking units. The tool offsets are pre-determined for each cutting unit and remain the same every time that you put in the unit.

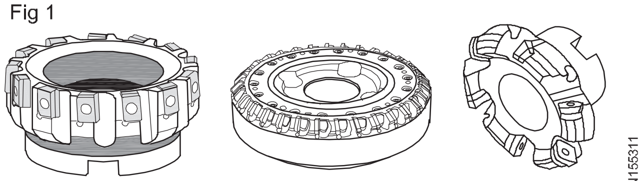
Nomenclature of cutting tools & Selection of cutting tools, its machining process parameter

Objectives: At the end of this lesson you shall be able to
- **state what's face mill, types and uses**
- **list and brief the various types of milling cuttings**
- **explain the construction and uses of 'U' drills and ball nose hose end mills**
- **describe the milling inserts design out on system**
- **explain the tool holding devices in VMC**

Types of face mills

Face mills may be of solid construction or with holders and inserts. They can be used with a number of end or tip geometry options, including square end, ball nose, radius tip, and chamfer tip. (Fig 1)

Fig 1



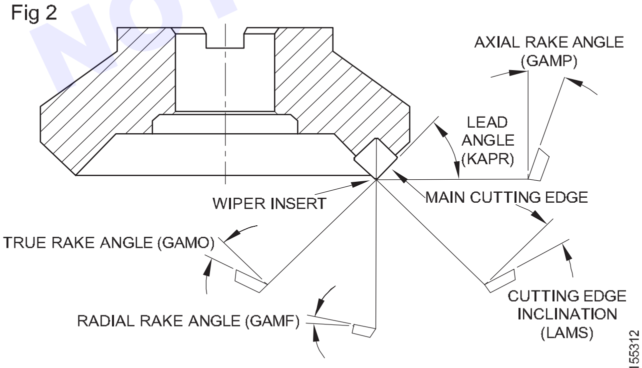
GENERAL PURPOSE FACE MILL CUTTER

CN20N155311

Specifications

When selecting a face mill, consideration must be made to the desired finish type. These are typically two finish options: roughing/hogging and finishing. Roughing/hogging mills are designed so that machine geometry, flutes, and materials can be used for rapid and heavy materials removal. They are typically used to machine workpiece close to the desired finishing dimensions, where a finishing face mill take over and produces closer tolerances and higher-quality surface finish. (Fig 2)

Fig 2



EACH CUTTING EDGE ANGLE IN FACE MILLING

CN20N155312

Other considerations for face mills include cutter size construction criteria. Size considerations for face mills relate to the:

- Cutting diameter
- Shank or arbor diameter
- Flute or cutting edge length
- Overall tool length
- Radius dimension and angle

Construction options for face mills include the number of flutes of cutting edges. This number can vary with the cutter diameter, milling material, and other factors. Two-flute face mills are often used with ductile materials that produce long chips. Face mills using a higher-number of flutes can be used to minimum chip load and vibration.

Materials

The materials of the face mills is important for understanding the level of cutting the machine can handle. Materials like carbide, cobalt, and diamond are hard and can be used in high-speed applications, whereas materials like steel are used for general metal machining. Other material options for face mills include micro grain carbide, which is used most-often in surface finishing applications, and ceramic.

Coatings for face mills are important considerations as well, as they can provide additional protection against corrosion and abrasion, increase the tool's hardness, provide lubrication and smoothness assistance, and improve the overall lifetime of the tool. other considerations and options for face milling may also be available depending on the manufacturer.

Types of CNC milling cutters

1 Face milling cutter

2 Shoulder milling cutter

3 END mill

   a Insert type end mill

   b Solid carbide end mill

   c Ball nose end mill

4 Drills

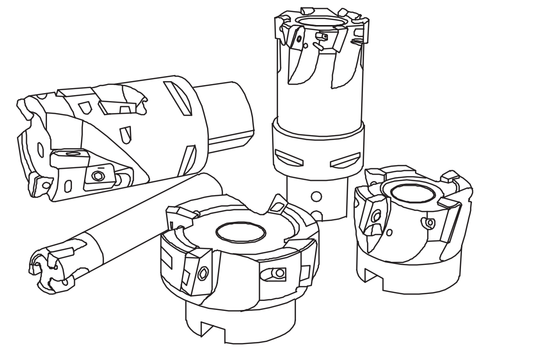
   a Solid carbide drill

   b 'U' Drill

Shoulder milling cutter (Fig 3)

Shoulder milling cutter is used to create a plane and shoulder surface at a time on the work surface.

Fig 3



End mill cutter (Fig 4)

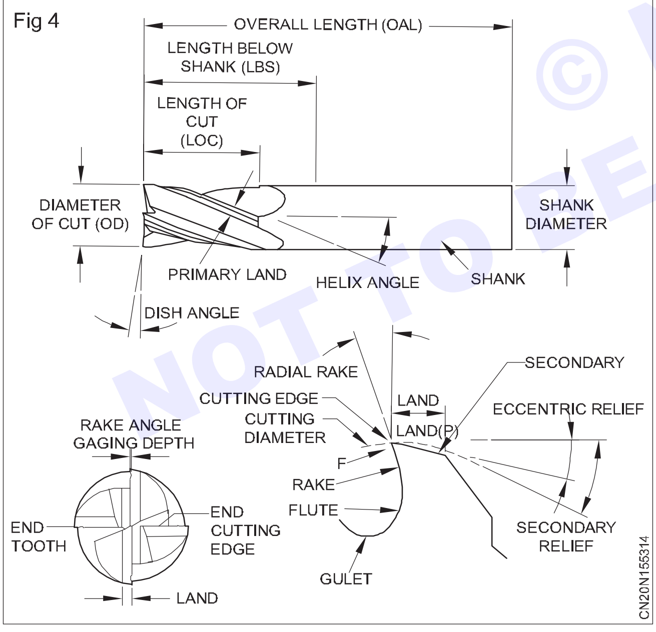
End mill is one kind of milling cutter to do the process of removing metal by CNC milling machines.

There are various diameters, flutes, lengths, and shapes to choose from. Basically, user choose them according to the material of the work piece and the surface finish required for the work piece.

Recommended cutting parameters - Square shoulder milling cutter

Work piece material		Hardness HB	Insert grade	Cutting parameters	
				V (m/min)	f (mm/z)
P	Low-carbon steel, Soft steel	< 180	YBG202 YBG205	190 (140-250)	0.08 (0.04-0.15)
	High-carbon steel, Alloy steel	180-280	YBG202 YBG205	170 (130-250)	0.08 (0.04-0.15)
	Alloy tool steel	280-350	YBG202 YBG205	150 (110-240)	0.08 (0.04-0.15)
M	Stainless steel	< 270	YBG202 YBG205	120 (180-190)	0.08 (0.04-0.15)
K	Cast iron	180-250	YBG202 YBG205	120 (80-210)	0.08 (0.04-0.15)

Fig 4



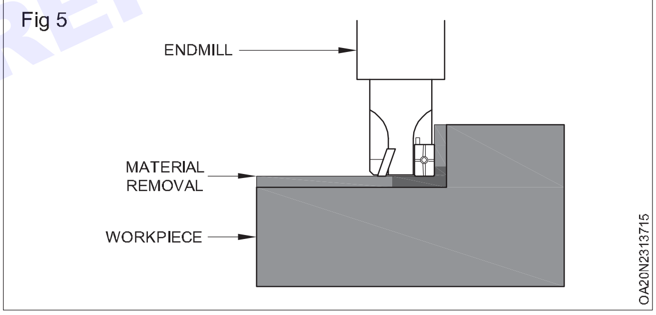
Insert type end mill cutter (Fig 5)

Milling is used essentially for the rapid removal of metal and is particularly suitable for the production of flat surfaces or a combination of surfaces. It is also possible to produce contoured surfaces by using form cutters. Holes may also be drilled and bored. Generally, tolerances of ± 0.025 mm may be held by milling, although 0.075mm is generally more practical.

The choice of cutter type and its size will upon several factors.

- Type of work being performed
- Power of machine available

Fig 5



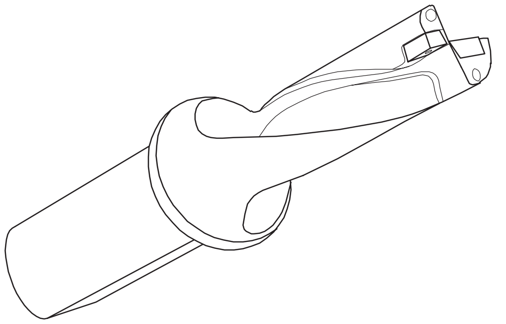
Generally, use of a cutter incorporating carbide inserts will reduce power requirement on the machine or allow use of increased feed and cutting speeds.

This cutter can be used produce slots, ramps, shoulders etc.

'U' Drill - specification (Figs 6 &7)

U- Drills are a kind of drilling instruments that include carbide inserts. These inserts are easy to replace and cost effectively. Insert type U-drills are used for the process of embedding heat pipe because of their widespread employment in CNC machining centers and high-speed machining applications.

Fig 6



L1 - distance between cutting edge to shoulder L2 - distance between cutting edge to neck

L - distance between cutting edge to end.

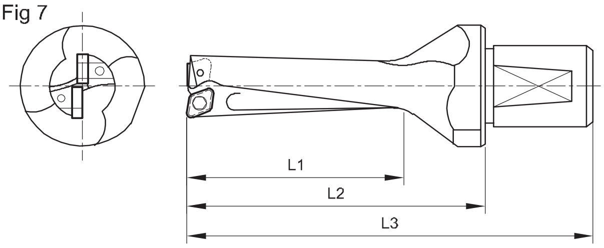
Ball Nose End mill

A Ball Nose End mill is a type of tool that is used to mill various curved surfaces and arc grooves on a VMC OR HMC machine with a cutter that looks like a ball head.

The ball nose end mill has the best impact on cutting curved surfaces, however because of the short chip retaining groove at the tip, chip discharge is low.

Ball nose end mills also called full radius end mills or ball mills. Ball nose cutter is cutting tools with a nose radius equal to half the tool's diameter. This results in a profile with a single radius (or ball) at the tool end and so straight edges (sharp corners). They're utilized for anything from contouring and profiling to slotting and corner selecting in milling. Their principal use in 3D semi-finishing and manufacture part shapes more effectively.

Fig 7



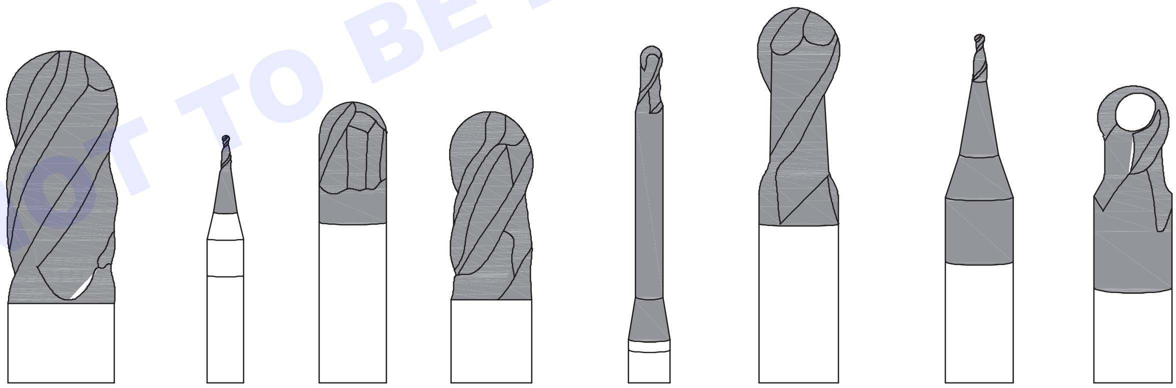
- Ball nose end mill 2 flutes
- Ball nose end mill 4 flutes
- Corner Radius end mill

Ball nose end mills have a radius at bottom, resulting in a finer surface finish in your work piece and less labour for you because the piece will not need to be finished further.

The edges of this tool are center cutting, and the end is ground with a full radius equal to half of the tool diameter. They can be used to mill big corner radius, full radius grooving, and contour or profile milling. Engraving can be done on the smaller diameters.

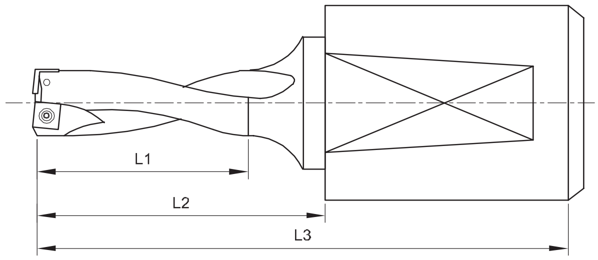
They're used in applications including contour milling, shallow slotting, pocketing, and contouring. Because they are less prone to chipping and produce a beautiful, rounded edge, ball nose mills are great for 3D contouring. (Fig 8)

Fig 8



BALL NOSE END MILL

U Drill cutting parameters



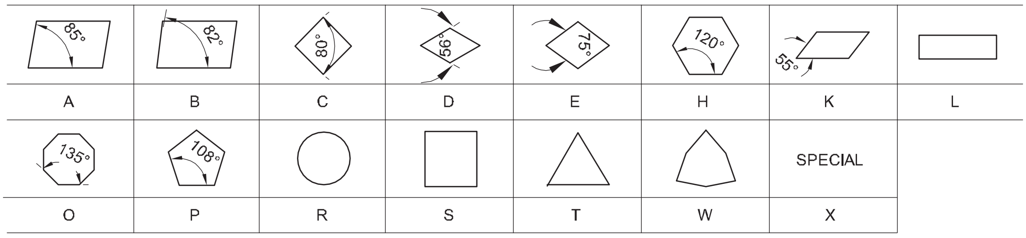
Designation	D	L	1.1	1.2	4	Insert	Screw	Key
17WCMX-25-3D	17	133	51	77	25	WC00030200	TS2506	FT08
18WCMX-25-3D	18	137	54	81				
19WCMX-25-3D	19	140	57	84				
20WCMX-32-3D	20	158	60	89	32	WC00040200		
21WCMX-32-3D	21	161	63	92				
22WCMX-32-3D	22	164	66	95				
23WCMX-32-3D	23	167	69	99				
24WCMX-32-3D	24	170	72	103				
25WCMX-32-3D	25	173	75	107		WC00050200	TS3007	FT10
26WCMX-32-3D	26	176	78	110				
27WCMX-32-3D	27	179	81	113				
28WCMX-32-3D	28	182	84	117				
29WCMX-32-3D	29	185	87	120				
30WCMX-32-3D	30	188	90	125				
31WCMX-32-3D	31	191	93	129		WC0006T300	TS3511	FT15
32WCMX-32-3D	32	194	96	132				
33WCMX-32-3D	33	197	99	136				
34WCMX-32-3D	34	200	102	139				
35WCMX-32-3D	35	203	105	143				
36WCMX-32-3D	36	206	108	146				
37WCMX-32-3D	37	210	111	150				
38WCMX-32-3D	38	214	114	154				
39WCMX-32-3D	39	217	117	157				
40WCMX-32-3D	40	231	120	161				
41WCMX-32-3D	41	235	123	165				
42WCMX-32-3D	42	238	126	168	40	WC00080400	TS4012	FT15
43WCMX-00-3D	43	241	129	171				
44WCMX-00-3D	44	245	132	175				
45WCMX-00-3D	45	248	135	178				

Milling insert designation system (Figs 9 & 10)

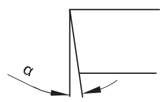
Fig 9

S	P	K	N
1	2	3	4

1 INSERT SHAPE



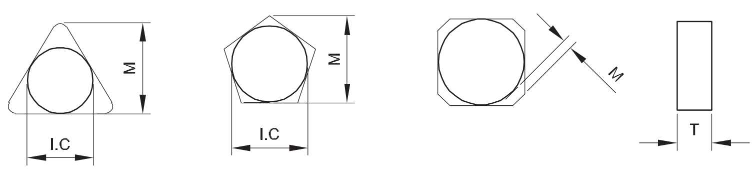
2 CLEARANCE ANGEL



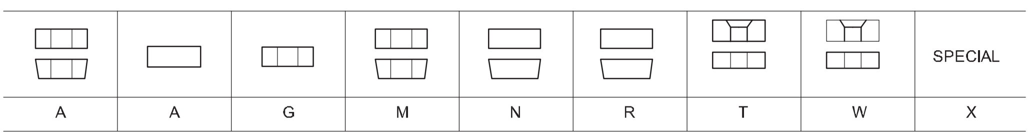
5°	7°	15°	20°	25°	30°	0°	11°
B	C	D	E	F	G	N	P

3 TOLERANCE

	TOLERANCE			I.C.SIZE					
	M	T	I.C	6.35	9.525	12.7	15.875	19.05	25.4
A	±0.005	±0.025	±0.025	●	●	●	●	●	●
C	±0.013	±0.025	±0.025	●	●	●	●	●	●
E	±0.025	±0.025	±0.025	●	●	●	●	●	●
F	±0.005	±0.025	±0.013	●	●	●	●	●	●
G	±0.025	±0.013	±0.025	●	●	●	●	●	●
H	±0.013	±0.025	±0.013	●	●	●	●	●	●
K	±0.013	±0.025	±0.05	●	●				
			±0.08			●			
			±0.010				●	●	
			±0.013						●
M	±0.08	±0.13	±0.05	●	●				
	±0.013		±0.08			●			
	±0.015		±0.10				●	●	
	±0.018		±0.13						●



4 CROSS SECTION SHAPE



CN20N155319

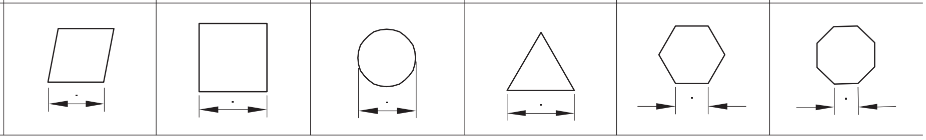
Fig 10

12	03	ED	T	R	CHIP BREAKER
5	6	7	8	9	10

5

CUTTING EDGE LENGTH

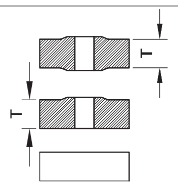
	C	S	R	T	H	O
I.C. SIZE						



5.56	05	05	05	09		
6.35	06	06	06	11		
7.94	08	08	08	13		
9.525	09	09	09	16		
12.7	12	12	12	22	05	05
15.875	16	16	16	27	09	06
19.05	19	19	19	33	10	
25.4	25	25	25	44		

6

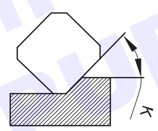
THICKNESS

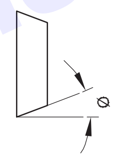


SYMBOL(t)	mm
02	2.38
03	3.18
T3	3.97
04	4.76
06	6.35
07	7.94
09	9.52

7

LEAD ANGEL &RELIEF ANGLE OF MINOR CUTTING EDGE



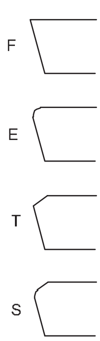


LEAD ANGEL	
A	45°
D	60°
E	75°
F	85°
P	90°
Z	SPECIAL

RELIEF ANGLE OF MINOR CUTTING EDGE	
B	5°
C	7°
D	15°
E	20°
F	25°
G	30°
N	0°
P	11°
Z	SPECIAL

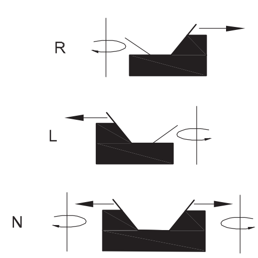
8

EDGE PREPARATION



9

CUTTING DIRECTION



10

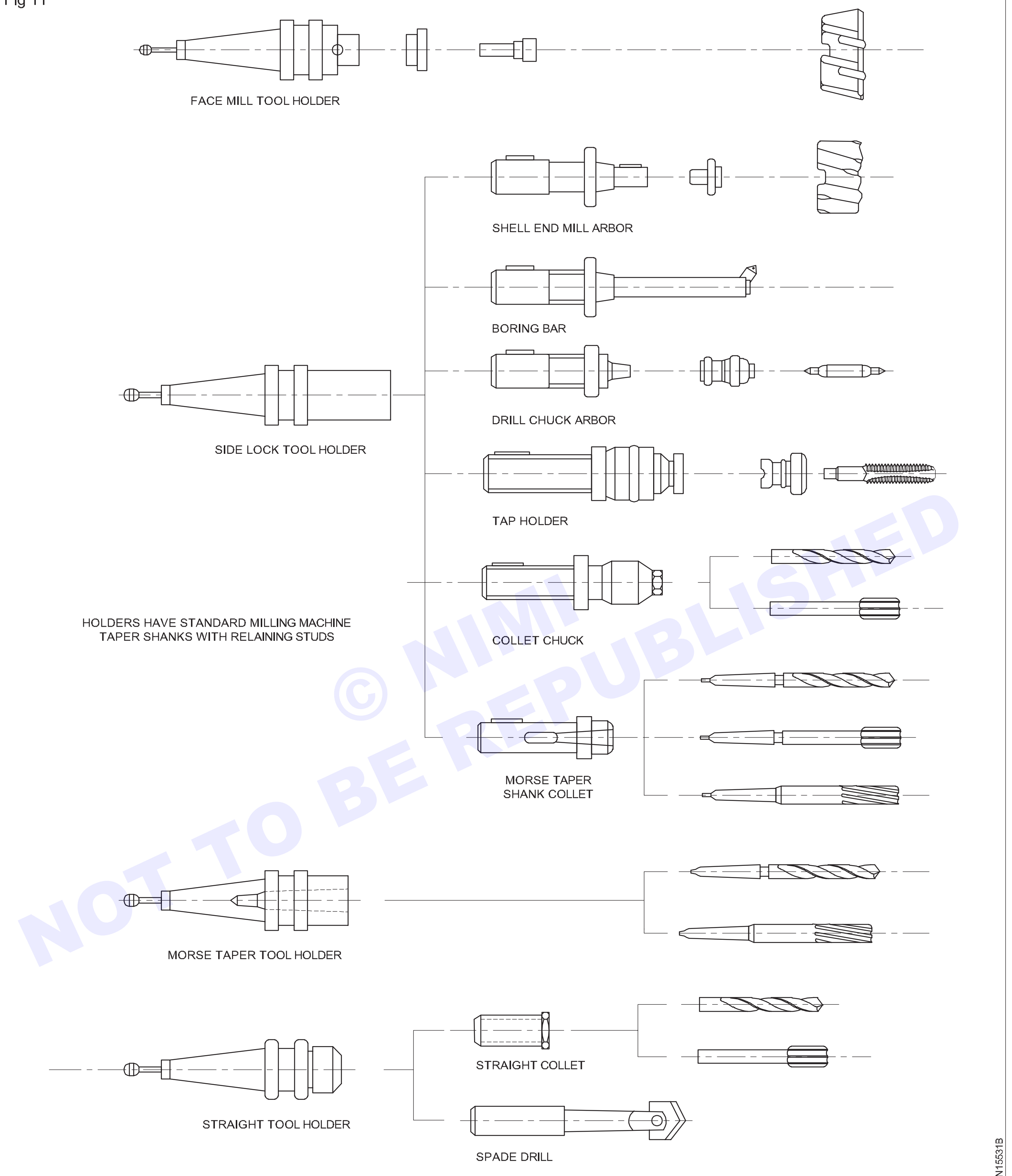
CHIP BREAKER

FOR APPLICATION

CN20N15531A

Types of holders are shown in (Fig 11)

Fig 11



TOOLING SYSTEM FOR A MACHINING CENTER

Machining process parameter - Cutting speeds, feeds and depth of cut

Objectives: At the end of this lesson you shall be able to
- **explain about the cutting speed**
- **explain about the feed**
- **explain the depth of cut.**

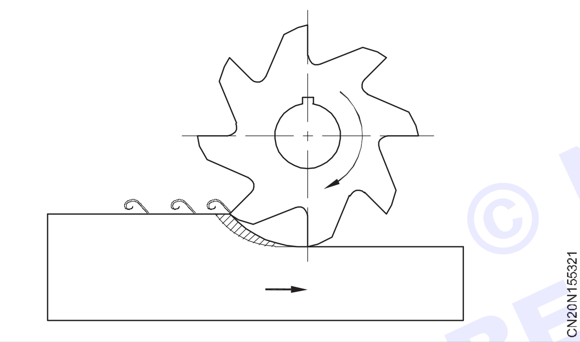
The efficiency of a milling operation depends upon the cutting speed, feed, and depth of cut.

If the cutter is run too slowly valuable time will be wasted, while excessive speed results in loss of time in replacing and regrinding cutters. Somewhere between these two extremes is the efficient cutting speed for the material being machined.

The rate at which the work is fed into the revolving cutters is important. If the work is fed too slowly time will be wasted and chatter may occur which eventually 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 work piece.

It is expressed in meters 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 machine ability, 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

$$\text{Cutting speed}(V) = \frac{\pi DN}{1000} \text{m/min}$$

$$N(\text{rpm}) = \frac{V \times 1000}{3.1416 \times D}$$

Since, only a few machines are equipped with a variable speed drive which allows them to be set to the exact calculated speed, a simplified formula can be used to calculate the revolution per minute. The p (3.1416) on the bottom line of the formula will divide the 1000 of the top line approximately 320 times. This results in a simplified formula which is close enough for most milling operations.

$$N(\text{rpm}) = \frac{V(\text{m}) \times 320}{D(\text{mm})} \qquad \text{rpm} = \frac{30 \times 320}{75} = \frac{9600}{75} = 128$$

where 'D' is diameter of the cutter.

Too fast a speed will shorten the cutter tool life. Too slow a speed will waste time.

Milling feeds and depth of cut

The two other factors which affect the efficiency of a milling operation are the milling Feed, 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 millimeters 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 (Figs 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 .

Feed per minute (Figs 4 & 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.

Fig 2

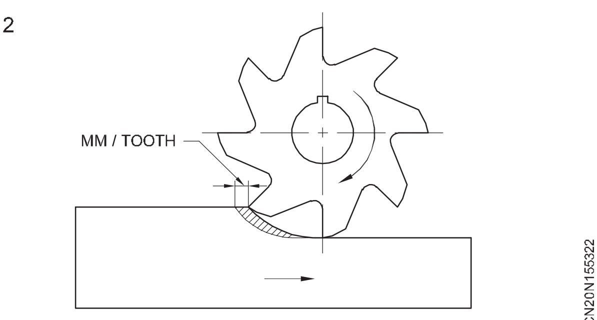


Fig 3

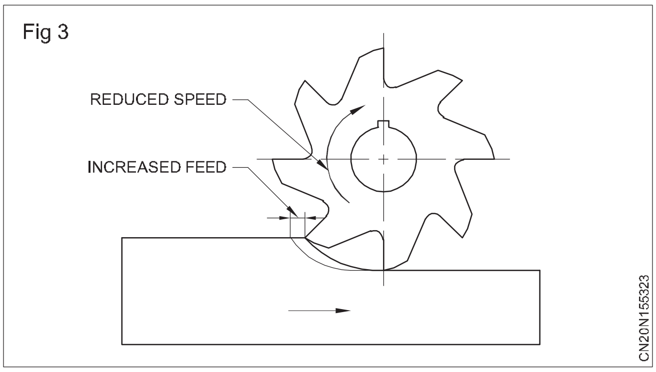


Fig 4

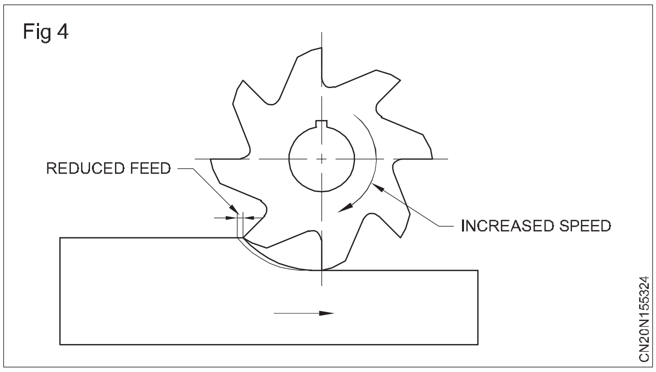
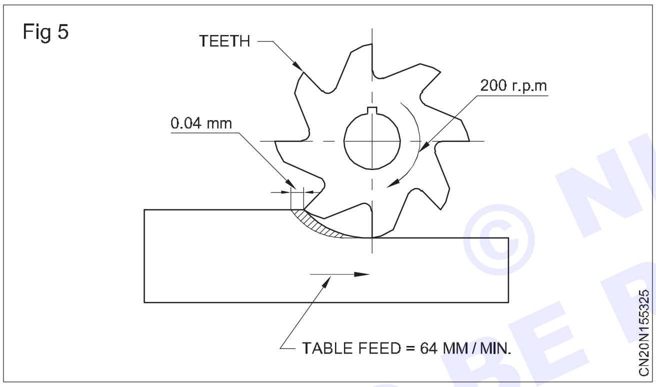


Fig 5



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. (Figs 6 & 7)

Feed per minute is calculated as follows

Feed in mm/min (or) Feed rate

F = Sz X Z X N

Where, Sz = Feed per tooth Z = Number of tooth N = RPM

F = Feedrate mm/min.

Fig 6

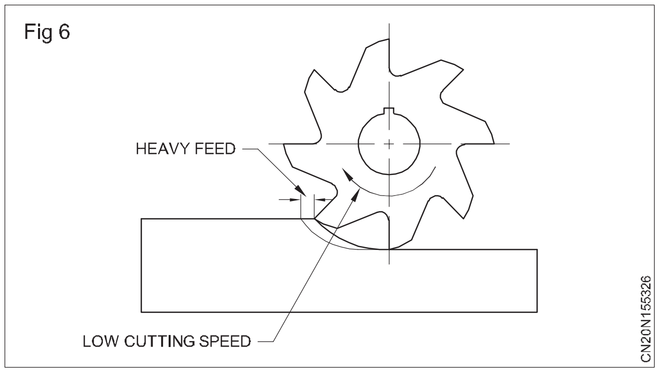
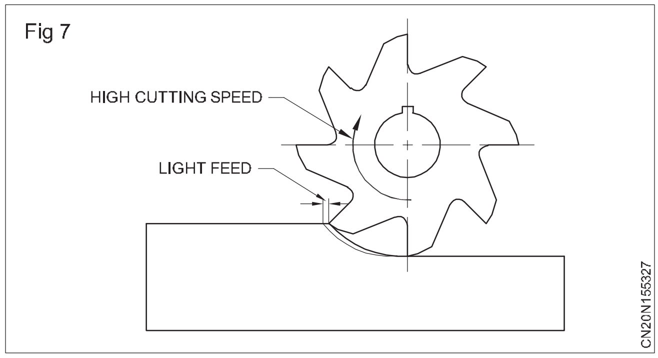
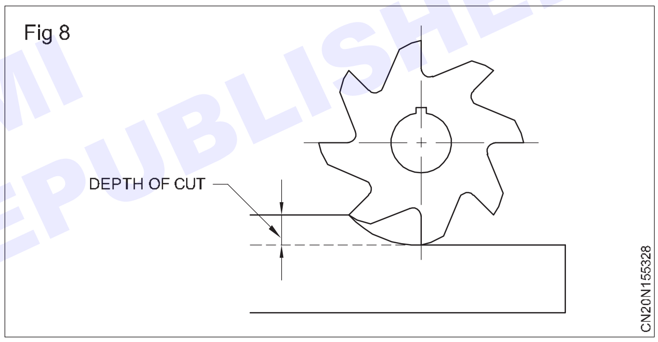


Fig 7



Depth of cut (Fig 8)

Fig 8



The depth of cut is the depth to which the cutter penetrates the work piece surface during a given cut. It is the perpendicular distance measured between the original and the final surface of the work piece.

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.

Recommended cutting speed, feed, Depth of cut for various materials are listed in Table 1 & 2

Table 1

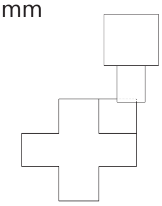
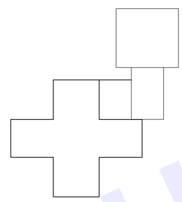
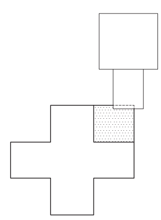
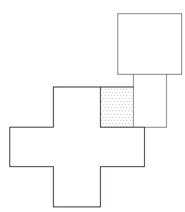
Speed/Feed/ Depth of cut for side milling with solid carbide end mill						
	Part material	Cutting speed Vcm/min	Feed rate Mm/Edge		Depth of cut mm	
			Rough	Finish	Rough	Finish (Radial)
Steel	Steel 130-250 BHN	60-90	0.04-0.08	0.02-0.04	Case 1: 1-4 mm 	0.1-0.5
	Steel 180-250 BHN	40-70	0.04-0.08	0.02-0.04		0.1-0.5
	Cast Steel 180-250 BHN	40-60	0.04-0.08	0.02-0.04		0.1-0.5
SS	Stainless Bar. Forged 200 BHN	40-60	0.03-0.06	0.02-0.04		0.1-0.5
	Stainless steel cast 200-330	30-50	0.03-0.05	0.02-0.04	Case 2: 5-10% of cutter dia 	0.1-0.5
CL	Grey CI 180-200 BHN	60-100	0.06-0.1	0.04-0.08		0.2-0.5
	Nodular CI 250 BHN	50-70	0.06-0.1	0.04-0.08		0.2-0.5
AL	Aluminium cast 60 - 100 BHN	150-300	0.1-0.2	0.06-0.12		0.3-0.6
	Aluminium cast 75 - 130 BHN	150-300	0.1-0.2	0.06-0.12		0.3-0.6

Table 2

(Speed / Feed / Depth of cut for side milling with solid carbide end mill)						
	Part material	Cutting speed Vcm/min	Feed rate Mm/Edge		Depth of cut mm	
			Rough	Finish	Rough	Finish (Radial)
Steel	Steel 130-250 BHN	150-220	0.12-0.2	0.05-0.1	Case 1: 1-4 mm 	0.1-0.5
	Steel 180-250 BHN	120-180	0.1-0.2	0.05-0.1		0.1-0.5
	Cast Steel 180-250 BHN	100-160	0.1-0.15	0.05-0.1		0.1-0.5
SS	Stainless Bar. Forged 200 BHN	80-120	0.12-0.2	0.05-0.1		0.1-0.5
	Stainless steel cast 200-330	60-100	0.08-0.15	0.05-0.1	Case 2: 10-20% of cutter dia 	0.1-0.5
CL	Grey CI 180-200 BHN	180-260	0.15-0.25	0.06-0.12		0.2-0.5
	Nodular CI 250 BHN	150-220	0.12-0.2	0.06-0.12		0.2-0.5
AL	Aluminium cast 60 - 100 BHN	350-500	0.15-0.3	0.06-0.12		0.3-0.6
	Aluminium cast 75 - 130 BHN	300-500	0.15-0.3	0.06-0.12		0.3-0.6

Cutting fluid & coolant used for machining

Objectives: At the end of this lesson you shall be able to
- **state what is cutting fluid**
- **state the function of cutting fluids & their advantages**
- **list the different types of cutting fluids.**

Cutting fluids and compounds are the substances used for efficient cutting while cutting operations take place.

Functions

The functions of cutting fluids are

- To cool the tool as well as the workpiece
- To reduce the friction between the chip and the tool face by lubricating
- To prevent the chip from getting welded to the tool cutting edge
- To flush away the chips
- To prevent corrosion of the work and the machine

Advantages

As the cutting fluid cools the tool, the tool will retain its hardness for a longer period; so the tool life is more.

Because of the lubricating function, the friction is reduced, and the heat generated is less. A higher cutting speed can be selected.

As the coolant avoids the welding action of the chip to the tool-cutting edge, the built-up edge is not formed. The tool is kept sharp, and a good surface finish is obtained.

As the chips are flushed away, the cutting zone will be neat.

The machine or job will not get rusted because the coolant prevents corrosion.

Properties of a good cutting fluid

- A good cutting fluid should be sufficiently viscous.
- At cutting temperature, the coolant should not catch fire.
- It should have a low evaporation rate.
- It should not corrode the workpiece or machine.
- It must be stable and should not foam or fume.
- It should not create any skin problems for the operator.
- Should not give off a bad smell or cause itching, which are likely to irritate the operator, thus reducing efficiency.
- Should be transparent.

Types of cutting fluids

The following are the common cutting fluids:

- Straight mineral oil
- Chemical solution (synthetic fluids)
- Compounded or blended oil
- Fatty oils
- Soluble oil (Emulsified oil)

Straight mineral oil

Straight mineral oils are the coolants that can be used undiluted. Use of straight mineral oil as a coolant has the following disadvantages:

- It gives off a cloud of smoke.
- It has little effect as a cutting fluid.

Hence, straight mineral oils are poor coolants. But kerosene, which is a straight mineral oil, is widely used as a coolant for machining aluminum and its alloys.

Chemical solution (synthetic oil)

These consist of carefully chosen chemicals in dilute solution with water. They possess a good flushing and cooling action and are non-corrosive and non-clogging. Hence, they are widely used for grinding and sawing. They do not cause infection or skin trouble and are artificially colored.

Compounded or blended oil

These oils are used in automatic lathes. They are much cheaper and have more fluidity than fatty oil.

Fatty oil

Lard oil and vegetable oil are fatty oils. They are used on heavy-duty machines with less cutting speed. They are also used on bench works for cutting threads by taps and dies.

Soluble oil (emulsified oil)

Water is the cheapest coolant, but it is not suitable because it causes rust to ferrous metals. An oil called soluble oil is added to water, which creates a non-corrosive effect with water in the ratio of about 1:20. It dissolves in water, giving a white milky solution. Soluble oil is an oil blend mixed with an emulsifier.

Other ingredients are mixed with the oil to provide better protection against corrosion and help prevent skin irritations.

Soluble oil is generally used as a cutting fluid for center lathes, drilling, milling, and sawing.

Soft soap and caustic soda serve as emulsifying agents.

A chart showing coolants for different metals is given below.

Recommended cutting fluids for various metals and different operations.

Material	Drilling	Reaming	Threading	Turning	Milling
Aluminium	Soluble oil	Soluble oil	Soluble oil	Soluble oil	Soluble oil
	Kerosene	Kerosene	Kerosene		Lard oil
	Kerosene	Mineral oil	Lard oil		Mineral oil
	Lard oil				Dry
Brass	Dry	Dry	Soluble oil	Soluble oil	Dry
	Soluble oil	Soluble oil	Lard oil		Soluble oil
	Mineral oil				Mineral oil
	Lard oil				Dry
Bronze	Dry	Dry	Soluble oil	Soluble oil	Dry
	Soluble oil	Soluble oil	Lard oil		Soluble oil
	Mineral oil				Mineral oil
	Lard oil				Lard oil
Cast iron	Dry	Dry	Dry	Dry	Dry
	Air jet	Soluble oil	Sulphurized oil	Soluble oil	Soluble oil
Copper	Dry	Soluble oil	Soluble oil	Soluble oil	Dry
	Soluble oil	Lard oil	Lard oil		Soluble oil
Steel alloys	Soluble oil	Soluble oil	Sulphurized oil	Soluble oil	Soluble oil
	Sulphurized oil	Sulphurized oil	Lard oil		Mineral
	Mineral lard oil	c	c		
General purpose steel	Soluble oil	Soluble oil	Sulphurized oil	Soluble oil	Soluble oil
	Sulphurized oil	Sulphurized oil	Lard oil		Lard oil
	Lard oil	Lard oil			
	Mineral lard oil				

Workpiece holding devices

Objectives: At the end of this lesson you shall be able to
- **define workpiece holding device**
- **distinguish between a three jaw chuck and a four jaw chuck**
- **identify different workpiece holding devices.**

Work-Holding on CNC lathe

On any machine tool, to carry out the machining process in an efficient manner, both the workpiece and tool must be held rigidly. This is accomplished by using jigs and fixtures. Fixtures hold the workpieces in the correct position with respect to cutter during the operation while jigs hold and guide the tool to the correct position on the workpiece. To machine a workpiece in the lathe it is necessary to secure it in some manner to the end of the spindle, the only true requirement being that the workpiece is held in such a way as to resist deflection by the cutting forces. Work holding devices in a lathe are used either for holding the workpiece or for supporting the workpiece during machining. Few of the widely used work holding devices are explained below.

Chucks

A chuck is a device, which is used for holding and rotating the job of shorter length during machining. A chuck is usually equipped with three or four jaws and accordingly they are classified as three-jaw chucks and four-jaw chucks.

Three jaw chuck

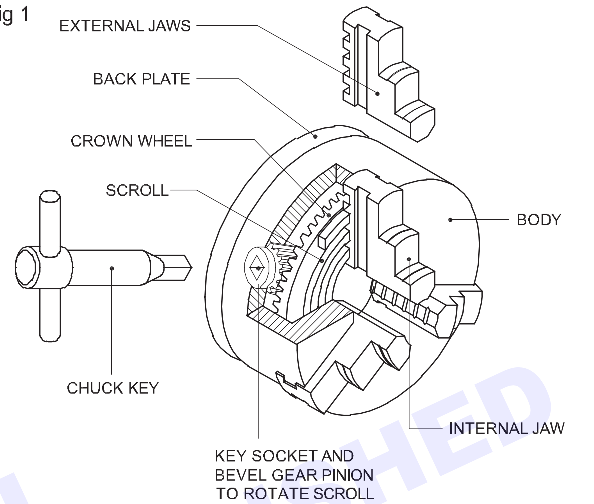
The three-jaw chuck is also known as self-centering chuck. The majority of the chucks have two sets of jaws for holding internal and external diameters. Only perfectly round work, or work with equally spaced flats, divisible by three, should be held in a three jaw chuck.

The construction of a three jaw chuck shows that the scroll not only clamps a component in place but also locates the component. This is fundamentally a bad practice, since any wear in the scroll and / or the jaws impairs the accuracy of location. Further, there is no means of adjustment possible to compensate for this wear. The jaws of this type of chuck are not reversible, and separate internal and external jaws have to be used.

Parts of a three-jaw chuck (Fig 1)

- Back plate
- Body
- Jaws
- Crown wheel
- Pinion

Fig 1



Back plate

The back plate is fastened at the back of the body by means of Allen screws. It is made out of cast iron. Its bore is tapered to suit the taper of the spindle nose. It has a key-way which will fit into the key provided on the spindle nose. There is a step in the front on which the thread is cut. The threaded collar, which is mounted on the spindle, locks the chuck by means of the thread, and locates by means of the taper and the key.

Body

The body is made out of cast steel, and the face is hardened. The body has three openings-120° apart to assemble the jaws and operate them. Three pinions are fixed on the periphery of the body to operate the jaws by means of a chuck key. The body is hollow in cross-section. The crown wheel is housed inside the body.

Jaws

The jaws are made out of high carbon steel, hardened and tempered, which slide on the openings of the body. Generally, there are two sets of jaws, viz. external jaws and internal jaws. External jaws are used for holding solid works. Internal jaws are used for holding hollow works. The steps on the jaws increase the clamping range. The back side of the jaws are cut out of scroll thread. Each jaw is numbered in a sequential manner, which will help in fixing the jaws in the corresponding numbered slots.

Crown Wheel

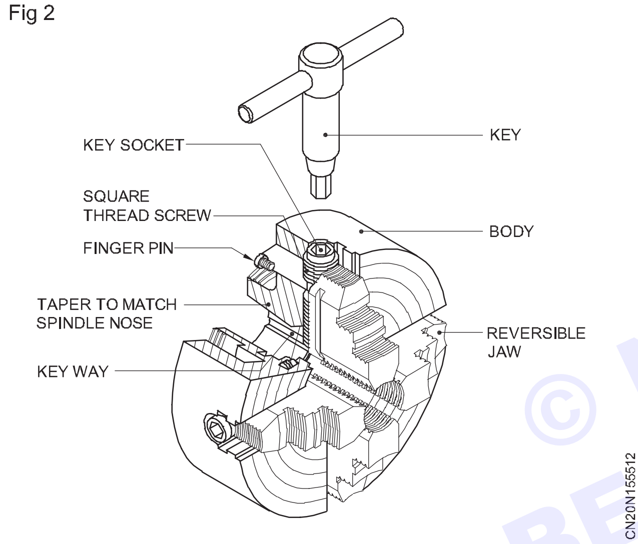
The crown wheel is made out of alloy steel, hardened and tempered. On one side of the crown wheel a scroll thread is cut to operate the jaws and the other side is tapered on which bevel gear teeth are cut to mesh the pinion. When the pinion is rotated by means of the chuck key, the crown wheel rotates, thus causing the jaws to move inward or outward depending upon the rotation.

Pinion

The pinion is made out of high carbon steel, hardened and tempered. It is fitted on the periphery of the body. On the top of the pinion, a square slot is provided to accommodate the chuck key. It has a tapered portion on which the bevel gear teeth are cut, which match with the crown wheel.

Four jaw chuck (Fig 2)

Fig 2



The four-jaw chuck is also known as an independent chuck, since each jaw can be adjusted independently; a work can be trued to within 0.001" or 0.02 mm accuracy, using this chuck.

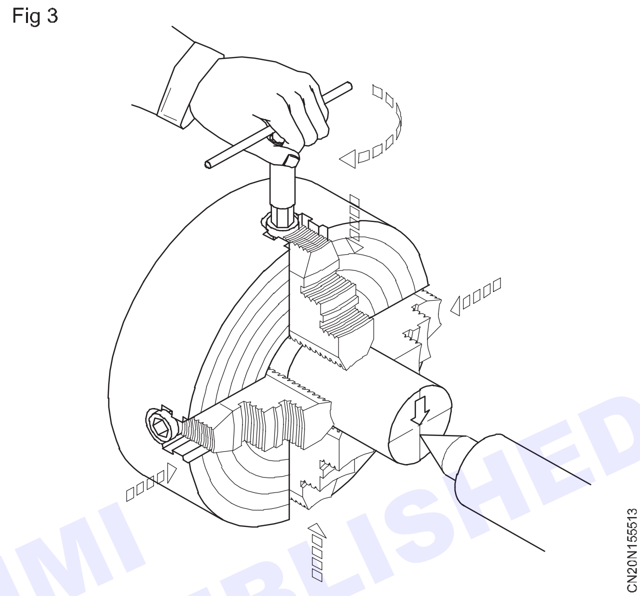
This type of chuck is much more heavily constructed than the self-centering chuck and has much greater holding power. Each jaw is moved independently by a square thread screw. The jaws are reversible for holding large diameter jobs. The independent four-jaw chuck has four jaws each working independently of the others in its own slot in the chuck body and actuated by its own separate square threaded screw. By suitable adjustment of the jaws, a workpiece can be set to run either true or eccentric with the machine centre.

Finished jobs when held in a four-jaw chuck can be trued with the help of a dial test indicator.

The checking of the workpiece should be carried out near the chuck and repeated as far from it as the workpiece permits, to ensure that the work is not held in the chuck at an angle to the axis of rotation.

The independent adjustment also provides the facility of deliberately setting the work off-centre to produce an eccentric workpiece. (Fig 3)

Fig 3



The parts of a four-jaw chuck are

- back plate
- body
- jaws
- square threaded screw shaft.

Specification of a chuck

To specify a chuck, it is essential to provide details of the:

- type of chuck
- capacity of the chuck
- diameter of the body
- width of the body
- the method of mounting to the spindle nose.

Lathe centre

Objectives: At the end of this lesson you shall be able to
- **explain face plates, mandrels, rests**
- **explain T-bolts, clamps, angle plates, V block and vices**

It is used to support lengthy works for carrying out turning operations. When the work is held in the chuck, the centre assembled in the tailstock supports the overhanging end of the work. The work is to be provided with a centre drilled hole on the face of the overhanging end. The centre, which is accommodated in the main spindle sleeve, is known as the 'live centre' and the centre fixed in the tailstock spindle is known as the 'dead centre'. In construction, both centres are identical. Lathe centres have a conical point of 60° included angle, the body provided with a Morse taper shank and a tang. (Figs 1, 2 & 3)

Fig 1

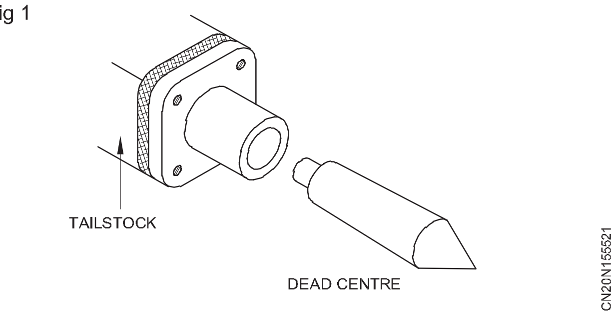


Fig 2

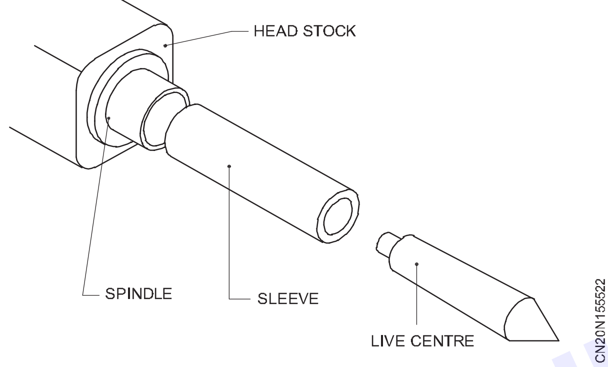
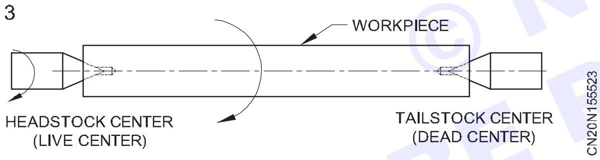


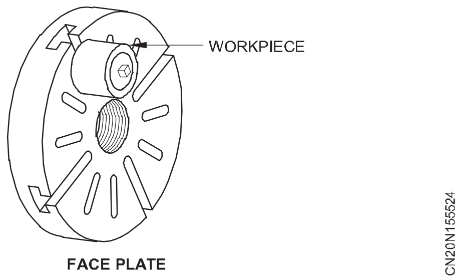
Fig 3



Face plates

Workpieces that cannot be accommodated in a chuck or between centres because they are asymmetrical or have a complex shape can be bolted to the faceplate either directly or indirectly via an angle plate. Bolts, clamps and dogs are used with the faceplate to hold the workpiece. Fig 4 shows a faceplate. The faceplate is directly mounted on the spindle.

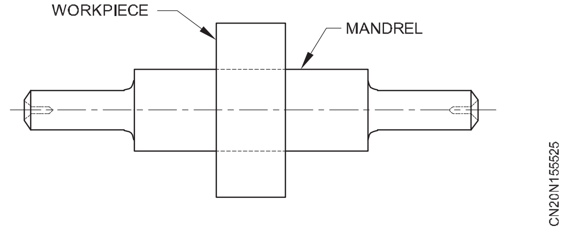
Fig 4



Mandrels

A workpiece having a large hole in it like a pipe cannot be held firmly in a chuck, because of its reduced strength. In such cases, mandrels are used to hold the workpieces from inside. A mandrel holding a workpiece is shown in Fig 5. During machining, mandrel is held between centres and the workpiece is then machined similar to machining between centres.

Fig 5



Rests

A rest is used for providing additional support to long workpiece when it is machined between centres or held in a chuck. If thin and long workpieces are not supported during machining, then there is a possibility that it may bend due to self-weight or due to the cutting forces exerted by the cutting tool on the workpiece. Two types of rests are in common use; these are the steady (or fixed) and follower (or traveling) rest.

Work holding devices used on milling machine

Various types of work holding devices are used for milling machine operations they are explained as follows:

- T-bolts and clamps
- Angle plates
- V-block
- Machine vices

T-Bolts and clamps

Bulky workpiece of irregular shapes are clamped directly on the milling machine table by using T-bolts and clamps. Different type of clamps are used for different patterns of work. All these clamps carry a long hole, through which clamping bolt passes. This hole permits the bolt for adjustment according to the size and shape of the job.

Angle plates

- When work surfaces are to be milled at right angles to another face, angle plates are used for supporting the work.
- The angle plate is made from high- quality material (generally spheroidal cast iron) that has been stabilized to prevent further movement or distortion.
- Slotted holes or "T" bolt slots are machined into the surfaces to enable the secure attachment or clamping of workpieces to the plate, and also of the plate to the worktable.
- Angle plates also may be used to hold the workpiece square to the table during marking-out operations.
- Adjustable angle plates are also available for workpieces that need to be inclined, usually towards a milling cutter.

V block

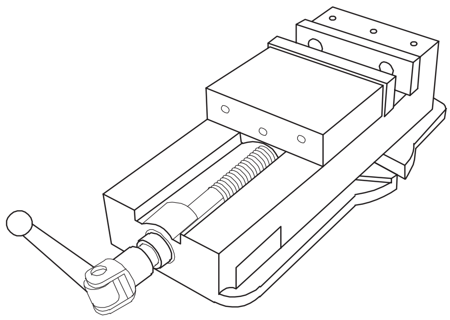
The V blocks are used for holding shafts on a milling machine table in which keyways and slots are to be milled.

Vice

Vices are the most common appliance for holding work on milling machine tables. According to its quick loading and unloading arrangement. Vices are of three types,

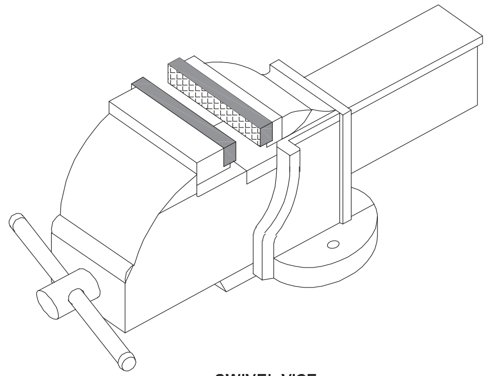
a **Plain vice:** The plain vice is directly bolted on the milling machine table is the most common type of vice used on plain milling operations, which involves heavy cuts, such as in slab milling. It's especially low construction enables the work to remain quite close to the table. This reduces the chance of vibration to a mini- mum. The base carries accommodate 'T' bolts to fix the vice on the table. Work is clamped between the fixed and movable jaw and for holding workpieces of irregular shape special jaws are sometimes used. (Fig 6)

Fig 6



b **Swivel vices:** The swivel vice is used to mill an angular surface in relation to a straight surface without removing the work from the vice. It has got a circular base graduated in degrees. The base is clamped on the table by means of T-bolts (Fig 7)

Fig 7



SWIVEL VICE

c **Universal vices:** It can be swiveled in a horizontal plane similar to a swivel vice and can also be titled in any vertical position for an angular cut. The vice is not rigid in construction and is used mainly in tool room work. It enables the milling of various surfaces, at an inclination to one another, without removing the workpiece.

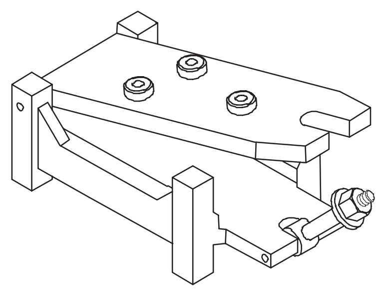
Jigs and Fixtures

Objectives: At the end of this lesson you shall be able to
- **state the advantages and disadvantages of using jigs and fixtures**
- **distinguish between the functions of jigs and fixtures**
- **understand drill jigs.**

A great deal of importance is placed today on improving productivity in manufacturing processes. Application of jigs and fixtures has contributed a lot towards this direction.

Jigs and fixtures (Figs 1 & 2) are devices used in manufacturing or assembling. They also facilitate in carrying out special operations accurately.

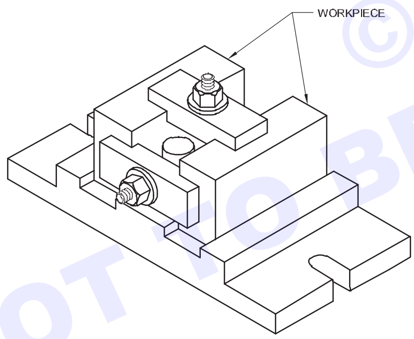
Fig 1



DRILL JIG

CN20N155611

Fig 2



MILLING FIXTURE

CN20N155612

Advantages of using jigs and fixtures

- It eliminates the setting time required before machining.
- It increases the machining accuracy and also increases production capacity.
- Requires less skilled operation
- They reduce the production cost.
- Increases the machine and labor utilization.
- They simplify the work handling.
- Increase the quality of production in the industry.
- They enable the quick setting of a tool and proper. positioning of the work.

Disadvantages of jigs and fixtures

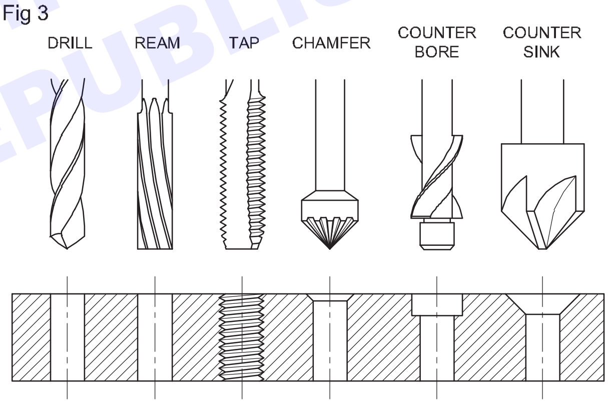
- Can wear away over time.
- Can have complicated designs.
- High initial set up costs and time.
- Can use a lot of material and be bulky.

Jigs

A jig is a special device which holds, supports, located and also guides the cutting tool during operation. Jigs are designed to accommodate one or more components at a time.

Jigs are available for drilling or boring. Drilling jigs are used to drill, ream, tap and to perform other allied operations. (Fig 3)

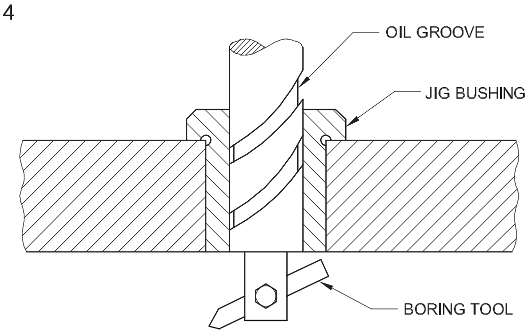
Fig 3



CN20N155613

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

Fig 4

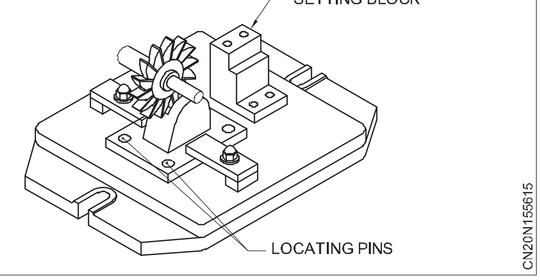


CN20N155614

Fixtures

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

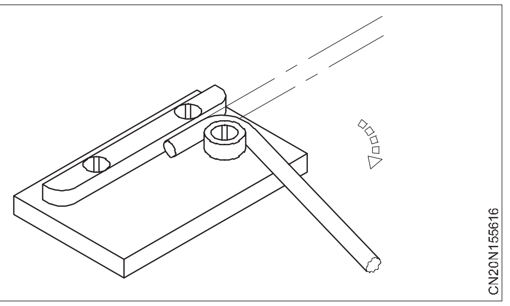
Fig 5



Fixtures of different types are made for

- Milling
- Turning
- Grinding
- Welding
- Assembly
- Bending etc. (Fig 6)

Fig 6



Programming sequence, ISO G codes and M codes

Objectives: At the end of this lesson you shall be able to
- **state about the CNC program**
- **list the three types of CNC program**
- **explain the ISO G and M codes**
- **describe the structure of the CNC program**
- **brief the absolute and incremental methods of programming.**

Program

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

Types of CNC programming

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

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

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

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

All of these CNC machine programming methods involve G-code and M-code. G-code is the programming language that instructs the CNC machine what to do, facilitating accurate and repeatable parts, M -code controls all of the 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 represented 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 moral 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 milli sec

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 linear	Coordinate data for three 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 rotation start or stop.

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

M01 - Optional stop

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

M02 - Program end

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

M03 - Spindle on clockwise

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

M04 - Spindle on counter clockwise

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

M05 - Spindle stop

By programming ‘M05’ the spindle rotation is stopped.

M08 - Coolant on

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

M09 - Coolant off

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

M30 - Program end & rewind

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

G - Codes (preparatory functions)

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

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

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

G00 - Rapid traverse

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

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

Format

1 G00 X ---------;

2 G00 Z-----------;

3 G00 X------ Z----;

G00 - code used for the following operations

1 Machining start

Making the tool approach the work piece.

2 During machining

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

3 Machining end

Separating the tool from the work piece.

G01 - Linear interpolation (straight cutting)

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

Format

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

Application

a Facing

b Grooving etc.

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

Application

a Straight turning

b Drilling etc.,

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

Application

a Taper turning

b Chamfering

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

Function F

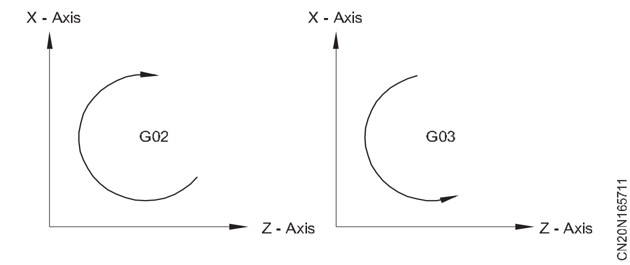
The feed rate is used to move the tool from one point to another point with constant feed rate. Feed is normally is given mm/rev. or mm/min. The rapid traverse rate and feed rate both are controlled by feed override switches in the machine panel.

Example

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

Circular interpolation (Fig 1)

Fig 1



G02-Circular interpolation clockwise direction

G03-Circular interpolation Anti clockwise direction

Format

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

OR

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

Where

X Z	-	End point of Arc
I K	-	Distance between start point of arc to center point of arc in X & Z axis
R	-	Radius of the arc
F	-	Feed

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

Example

G02 X 40.0 Z-5.0 R 5.0 F 0.1

G03 X 40.0 Z-5.0 R5.0 F 0.1

Where, R=Radius

G04-Dwell

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

Format

G04 (U, X, P) time

Example

G04 U 1.0 (Dwell of 1.0 second)

Note

Decimal point is not available in 'P'

Ex. Dwell of 2.5 seconds.

G04 U 2.5

G04 X 2.5

G04 P 2500

G28 - Zero return (Home position, first reference value)

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

1 G28 U0

2 G28 W0

3 G28 U0 W0

G30 - Second reference return

It is same as G28.But is to be 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

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

Where

V = cutting speed

D = Diameter of the work piece in 'millimeter'

N = spindle speed in rpm

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

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

Ex. G97 S300 M03.

With this spindle rotates at 300 rpm.

For the following should use G97 always

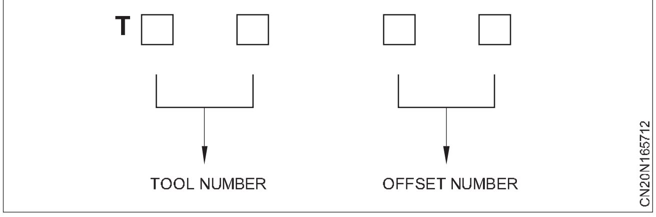
a Threading

b Tapping

c Drilling etc

Tool function (Fig 2)

Fig 2



Address: T

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

Format

Example: T01 01

Tool number

The left most two digits specify the number of tool.

Offset number

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

Types of offsets

There are two types offsets:

1 Wear offset

2 Geometrical offset

1 Wear offset

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

2 Tool geometry offset

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

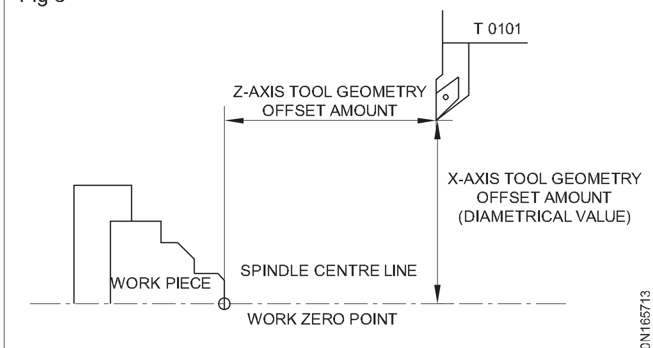
Tool geometry offset (Fig 3)

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

Tool wear offset

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

Fig 3



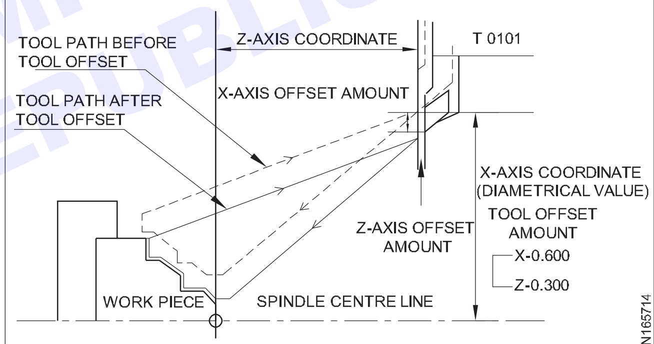
TOOL GEOMETRY OFFSET

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

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

Procedure for setting work coordinate system (Fig 4)

Fig 4



WORK ZERO POINT

Step 1 Mark sure that the component is securely clamped.

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

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

Step 4 Enter S500 M03;

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

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

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

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

Step 9 Now switch off the spindle.

Step 10 Press MENU offset. The wear geometrical and work shift are displayed on CRT.

Step 11 Now, press GEOM soft key and position the cursor using cursor movement buttons to be required offset number corresponding to the tool used.

Step 12 Press measure(m) key and press Z. Enter Zero(MZ0) press measure (m) key.

Step 13 Now rotate the spindle in appropriate direction and machine on OD

Step 14 Do not move X axis.

Step 15 Take Z away from the job.

Step 16 Stop the spindle.

Step 17 Press MENU OFFSET

Step 18 Press ‘GEOM’ soft key.

Step 19 Position the cursor to the required tool offset number.

Step 20 Press M….X….

Step 21 Input “The OD dimension measured. The X-offset for the said tool is set.

Step 22 Repeat the procedure for all tools.

Step 23 After taking offset, select MDI and enter S0.

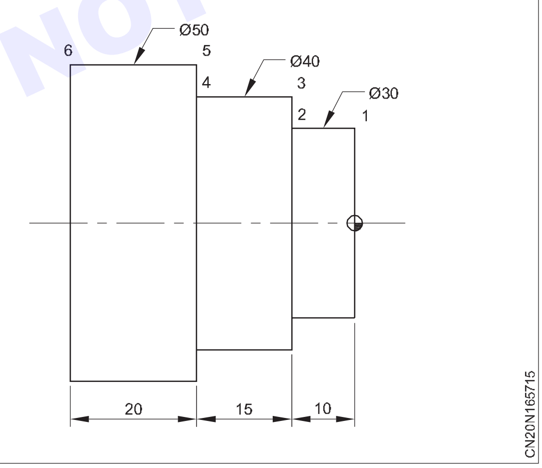
Programming method

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

Absolute method (Fig 5)

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

Fig 5



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

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

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

Incremental method (Fig 6)

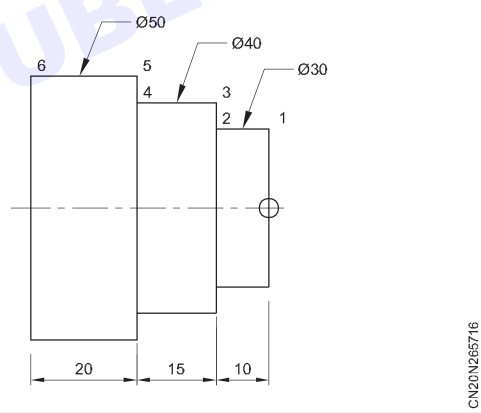
In this system, tool move from the previous point. In the incremental programming the address “U” (diametrical) for “X” axis and the address “W” for “Z” axis is used to distinguish incremental program from the absolute program.

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

Example

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

Fig 6



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

CNC lathe types

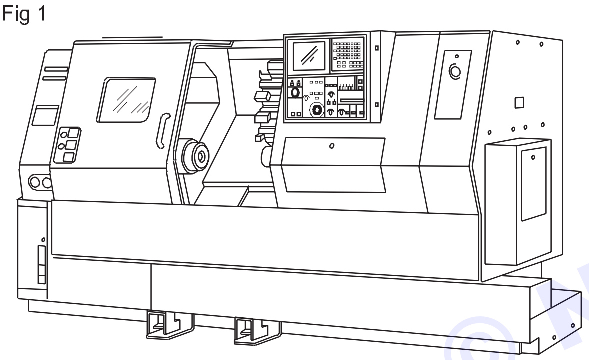
Objectives: At the end of this lesson you shall be able to
- **list the two types of turning centers**
- **explain the process of axes identification for horizontal and vertical turning centers.**

CNC lathe types and machine axis

Turning is a machining process used to make cylindrical parts in which the cutting tool moves in a linear fashion while the work piece rotates. A turning center is a lathe with a computer numerical control (CNC).

There are two types of CNC turning centers: Horizontal (Fig 1) and Vertical (Fig 2). Horizontal turning centers are the most common. Vertical turning centers are typically called a vertical turret lathe or VTL.

Fig 1



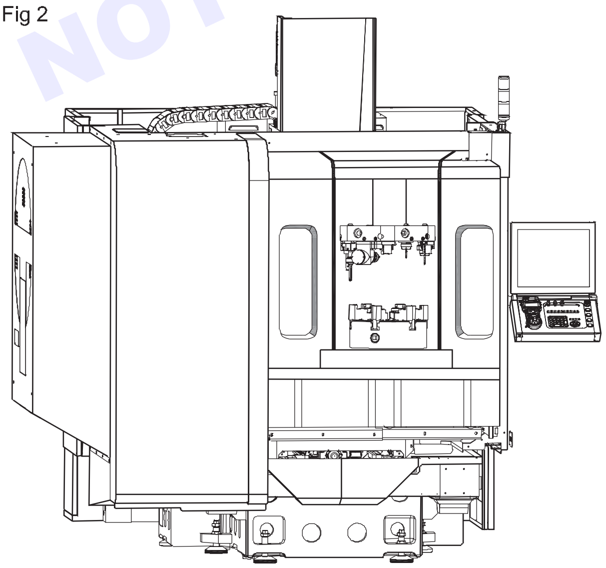
HORIZONTAL TURNING CENTER

CN20N166011

Vertical turning centre

Vertical turning centre are used where heavy parts has to be machined. The turret (tool holding device) moving up and down. The chuck mounted in vertical position. (Fig 2)

Fig 2



VERTICAL TURNING CENTER

CN20N166012

Machine axis identification

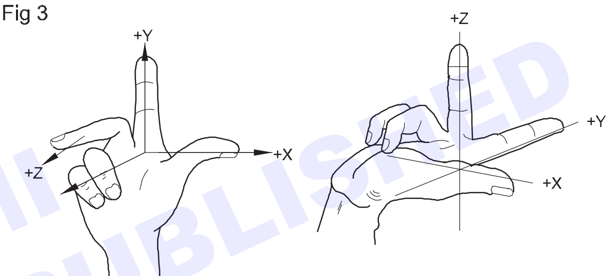
NC coordinates system

All the NC machine toolmaker's use of Cartesian coordinate system for the sake of simplicity. The guiding coordinate system followed for designating the axes is the well known as right hand coordinate system. (Fig 3)

Designation of axes (Fig 3)

First axes to be identified is the z axis.This is then followed by x and y axes respectively.

Fig 3



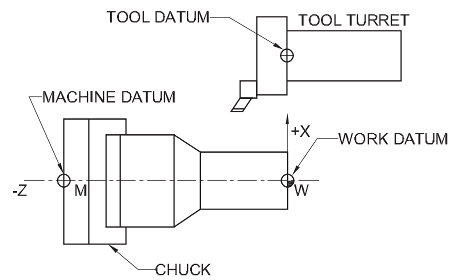
RIGHT HAND COORDINATE SYSTEM

CN20N166013

Axis system for horizontal turning center (Fig 4)

In turning centers, the spindle axis is designated as Z. The radial axis perpendicular to the z-axis and away towards the principle tool post is termed as x-axis. The machine datum or home position may be the intersection of spindle axis and clamping plane. At the start, the controller display will show the axis position with respect to home The work piece datum is fixed by the programmer on the work piece for the convenience of part programming. The difference between the tool tip position and the turret datum is termed as offset.

Fig 4



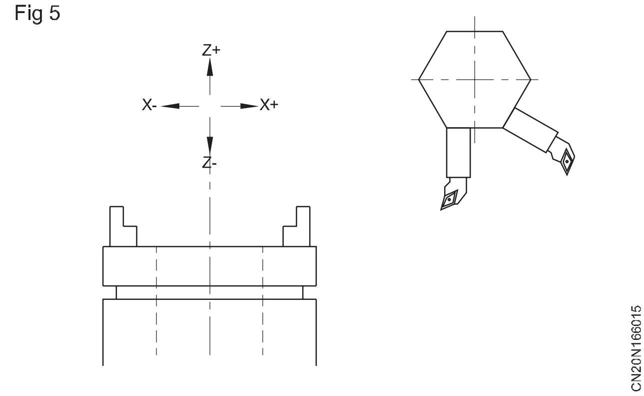
AXES SYSTEM USED IN TURNING CENTRES

CN20N166014

Vertical turning center (Fig 5)

In the vertical turning center the X axis movement is left and right, Z axis movement is up and down.

Fig 5



X-axis

The principle motion direction of cutting tool on the work piece is designated as x-axis. It is perpendicular to the z-axis and should be horizontal and parallel to the work holding surface whenever possible.

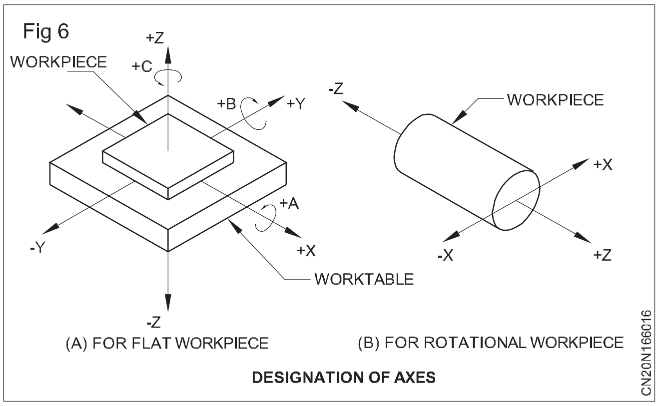
Z-axis

The z-axis motion is along the spindle axis or parallel to the spindle axis. In the case of machine without a spindle such as shapers and planers, the z-axis is perpendicular to the work holding surface.

For machines such as milling, drilling and lathe, the cutting tools move in the negative z direction to move a tool into the work piece. The positive z motion increases the clearance between the tool holder and work piece surface.

When there are several spindles and slide ways, the spindle perpendicular to the work holding surface may be chosen as the principle spindle. The primary Z motion is then related to the primary spindle. The tool motions of other spindles designated as A, B and C respectively. (Fig 6)

Fig 6



Part programming for turning centers

Diameter programming

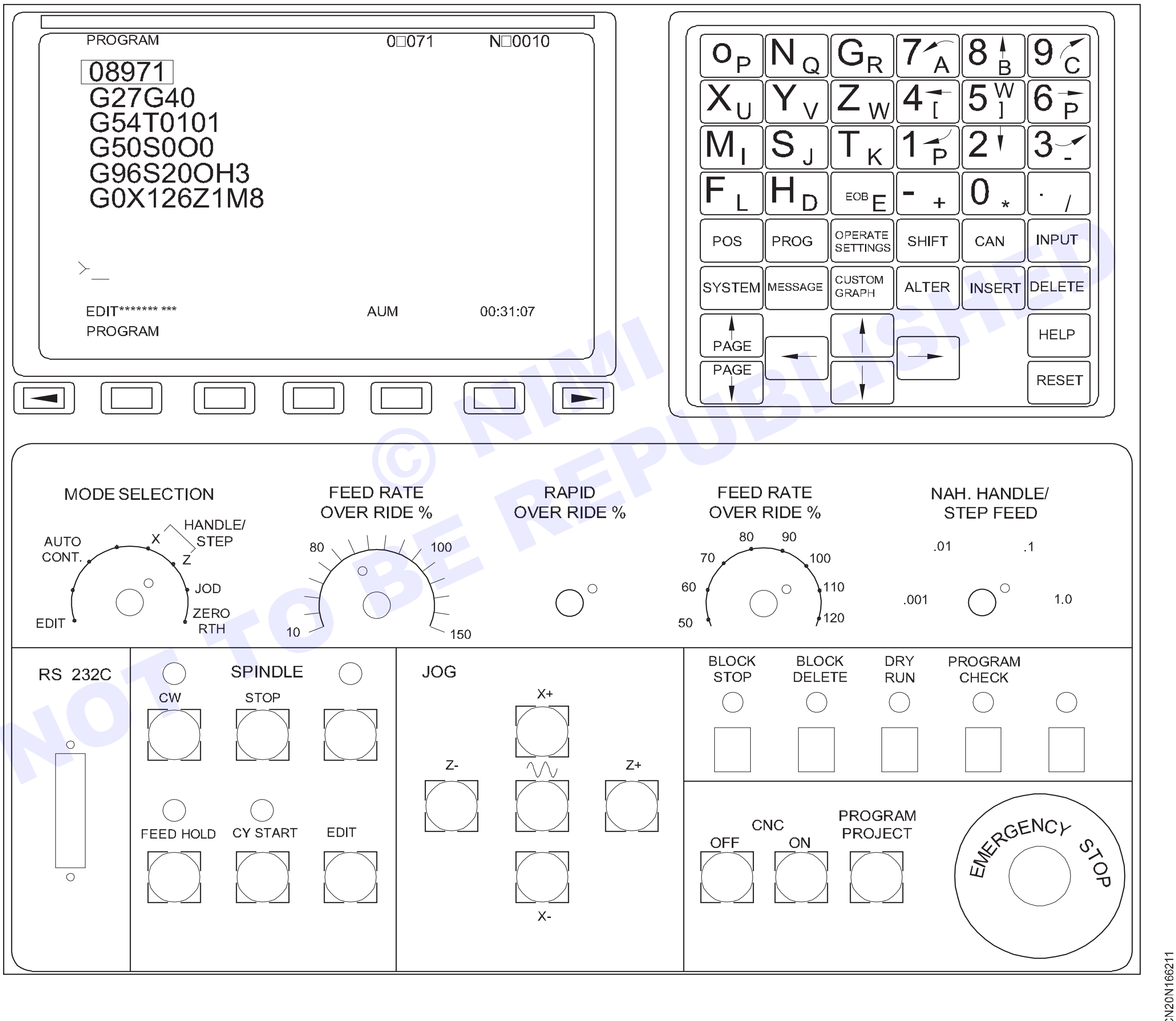
The dimensioning of a turned component is generally specified by its diameters. However, in turning operation, the tool should approach the work piece in radial direction for machining. Hence, for the sake of simplicity, most of the turning centers are provided with diameter programming facility.

This means that all the movements of the tool along X-axis should be doubled to represent the diametral rather than radial movement. The selection of radius or diameter programming depends upon the system variable set during the integration of controller with the machine tool.

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

Objectives: At the end of this lesson you shall be able to
- **state the various operational modes**
- **state the work piece zero point**
- **state the tool zero point**
- **explain the machine reference point**
- **list the various types of offset**
- **prepare the part program for facing, plain turning, step turning, chamfering, radius turning and drilling.**

Fig 1



Operational modes

Jog mode

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

Incremental jog mode

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

Edit mode

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

Manual mode: MDI mode

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

Single block mode

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

Auto mode

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

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

Zero points and reference points

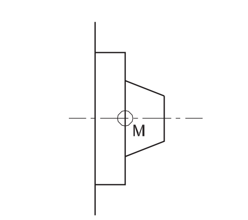
Zero point

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

Machine zero point or machine datum (M)

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

Fig 2

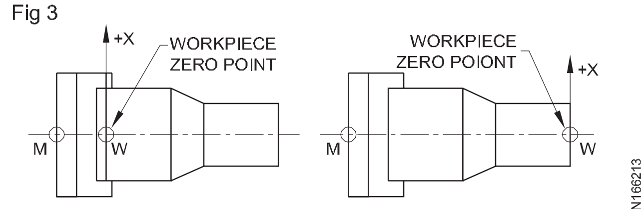


MACHINE ZERO PONT

Work piece zero point (W)

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

Fig 3



WORK PIECE ZERO POINT

Tool zero point (T)

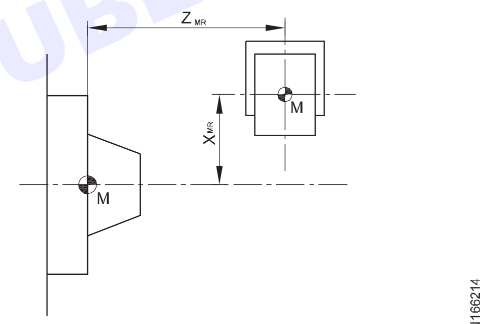
When machining a work piece, the tool must be controlled in precise relationship with the work piece along the machining path. This requires a point in the tool turret be taken as reference point, which is known as tool zero point.

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

Machine reference point (R)

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

Fig 4



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

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

Programming details (Fig 5)

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

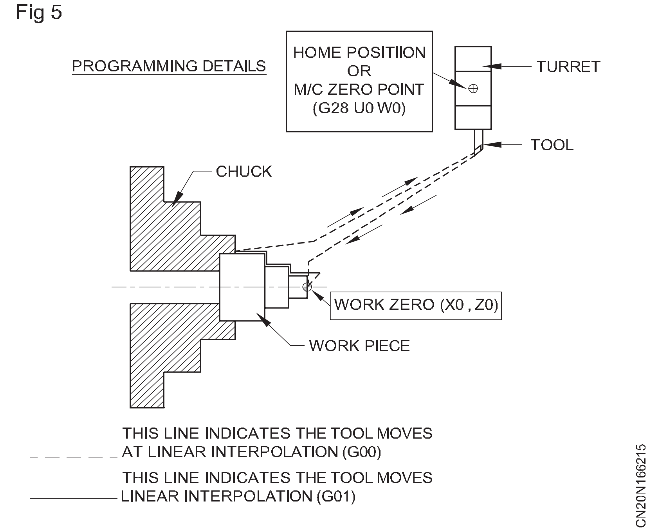
———— This line indicates the tool moves at linear interpolation (G01)

Operation devices

Edit alphanumeric keys

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

Fig 5



Feed rate override switch

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

Rapid rate override switch

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

Spindle speed override switch

Enables manual overriding of the spindle speed during part machining.

Feed hold

Stops the motion of all axes temporarily during machining.

Cycle start

Machine rest

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

Emergency stop

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

Single block ON/OFF

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

Coolant ON/OFF Controls the coolant.

Data input/output

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

Chip conveyor forward backward Moves the chip conveyor.

Dry run

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

Machine lock and auxiliary function lock

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

Operational modes

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

Selection of tools, speed feed & depth of cut

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

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

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

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

Procedure for JOG feed

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

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

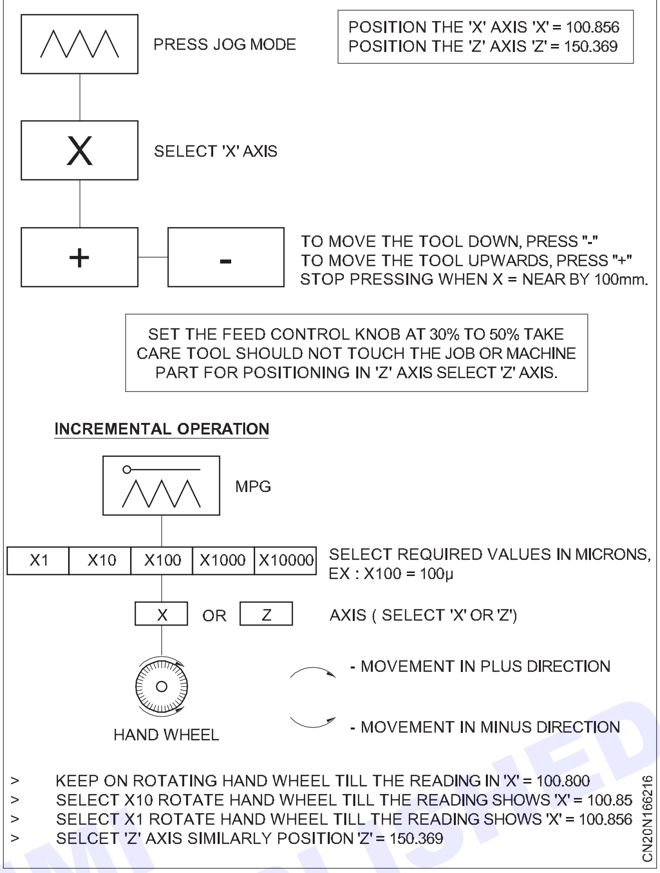
Procedure for INCREMENTAL feed (Fig 6)

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

Procedure for MANUAL HANDLE feed

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

Fig 6



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

The instructor will demonstrate the various axis and models.

Axes convention of CNC machines

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

Types of CNC machines

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

Types of CNC lathes

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

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

FRONT lathe ... an engine lathe type

REAR lathe ... a unique slant bed type

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

Axes designation

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

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

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

Typical configuration of a CNC lathe with two turrets

Schematic representation of a vertical CNC lathe

Fig 1

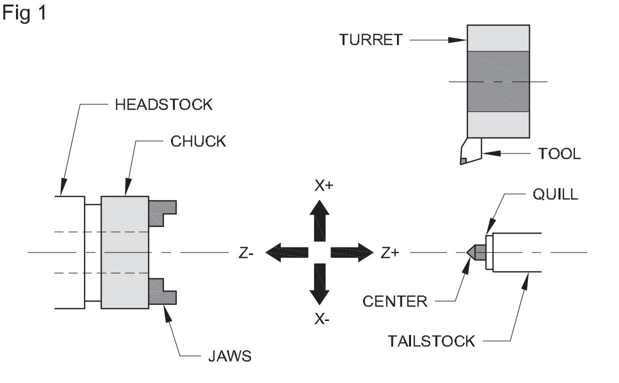


Fig 2

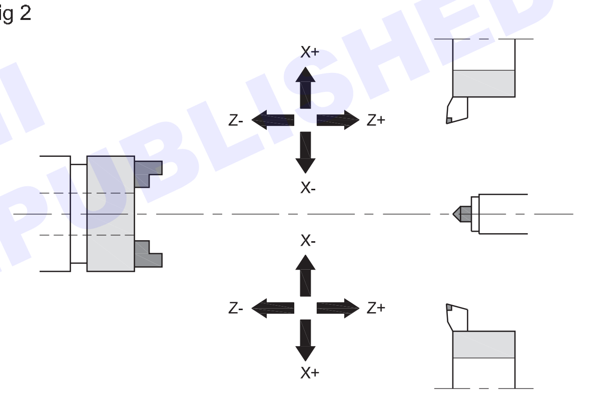
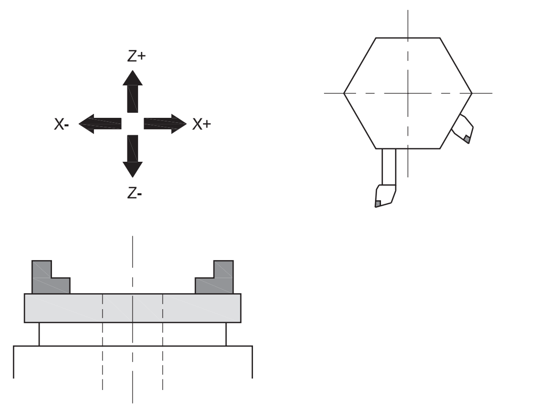


Fig 3



Two-axis lathe

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

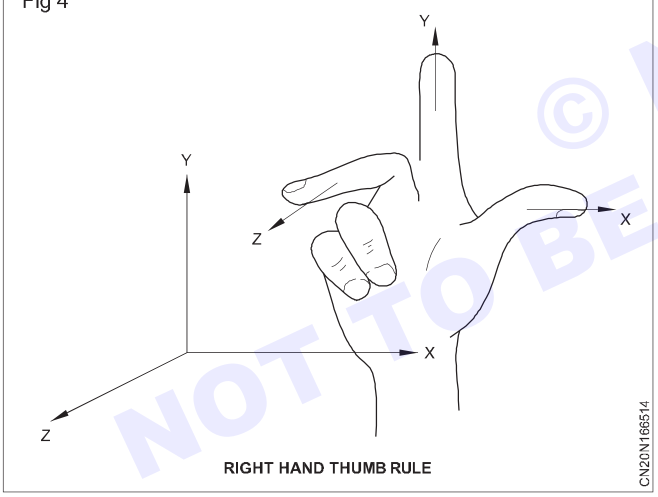
Three-axis lathe

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

Axis - nomenclature

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

Fig 4



RIGHT HAND THUMB RULE

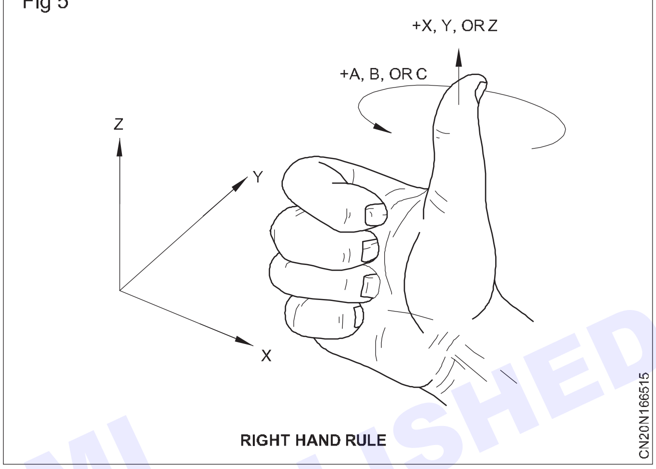
Auxiliary axes on NC machine

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

Right hand rule

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

Fig 5



RIGHT HAND RULE

Concepts of coordinate geometry

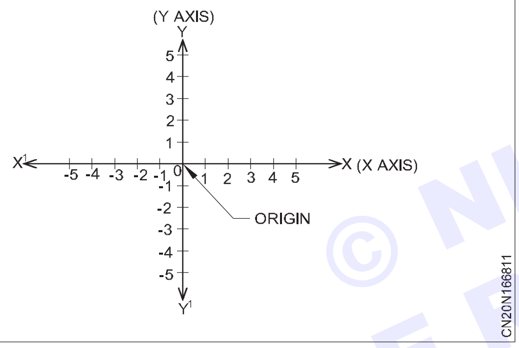
Objectives: At the end of this lesson you shall be able to
- **brief the concept of coordinate geometry**
- **describe the machine geometry**
- **state about machine zero, work zero and reference point in CNC machine.**

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

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

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

Fig 1



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

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

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

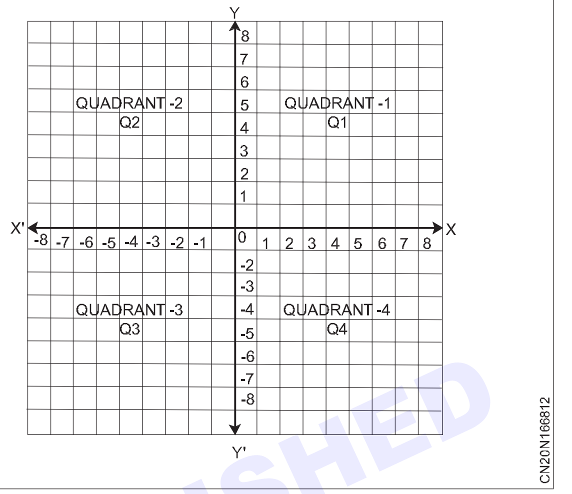
The plane here is known as the Cartesian plane or co-ordinate plane or XY-plane. The X & Y axes are known as coordinate axes.

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

Coordinates of a Point (or) Locating of coordinate points

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

Fig 2



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

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

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

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

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

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

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

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

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

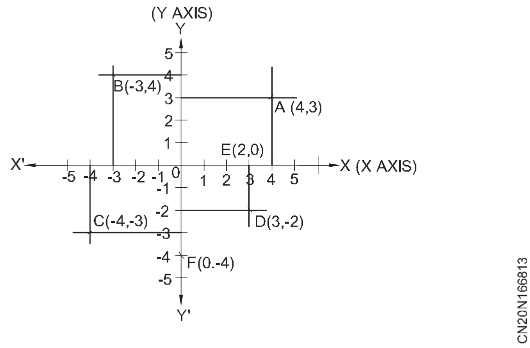
The point "A" is at a distance of 4 units measured along positive point of X-axis from origin. The same point is at a distance of 3 units measured along positive point of Y-axis from origin.

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

Hence the coordinates of A are (4,3)

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

Fig 3



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

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

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

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

Hence the coordinates of D is (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” is (2,0).

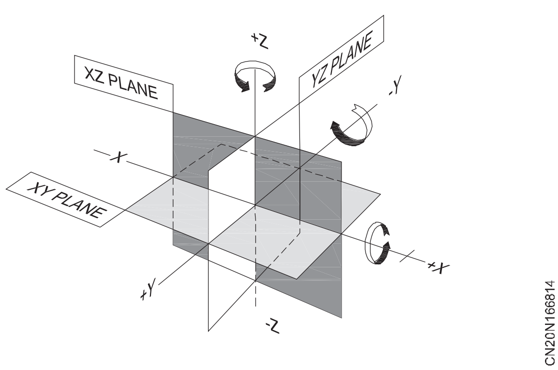
The point “F” is at a distance of -4 units from the X-axis and at a distance zero from the Y-axis. Therefore, the x-coordinate of “F” is 0 and y-coordinate is -4.

Hence the coordinates of “F” is (0, -4)

Machine geometry

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

Fig 4



Axis orientation - turning

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

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

Fig 5

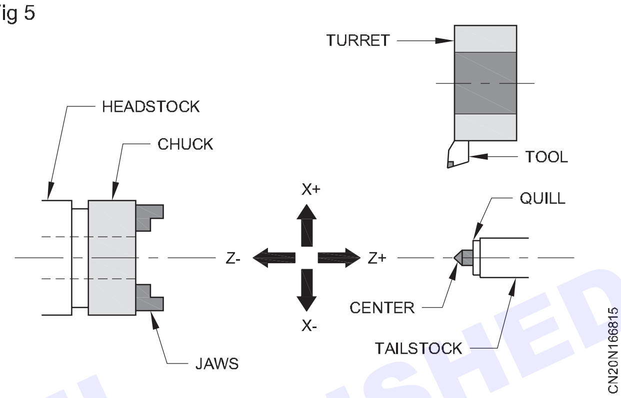


Fig 6

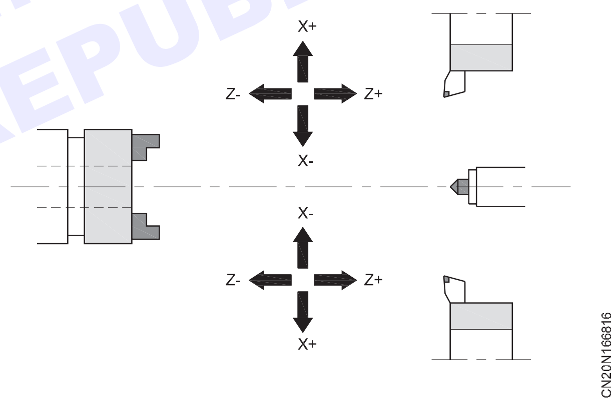
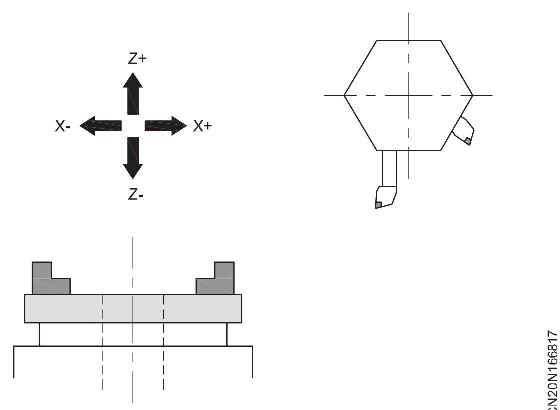
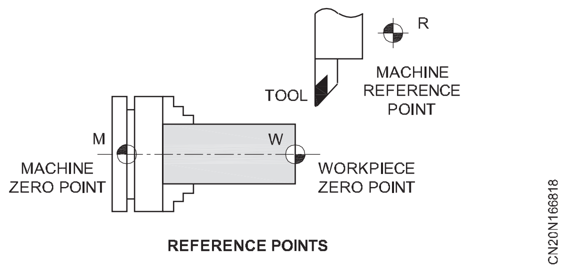


Fig 7



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

Fig 8



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

Machine zero point

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

Work zero point

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

Reference point

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

Coordinate system on CNC machine

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

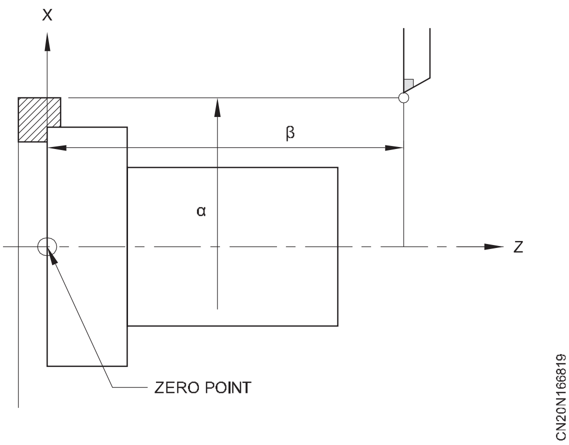
X_Z_

This command is referred to as a dimension word

Coordinates are specified in one of following three coordinate systems:

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

Fig 9



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

Machine coordinate system

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

Workpiece coordinate system

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

Setting a work piece coordinate system

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

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

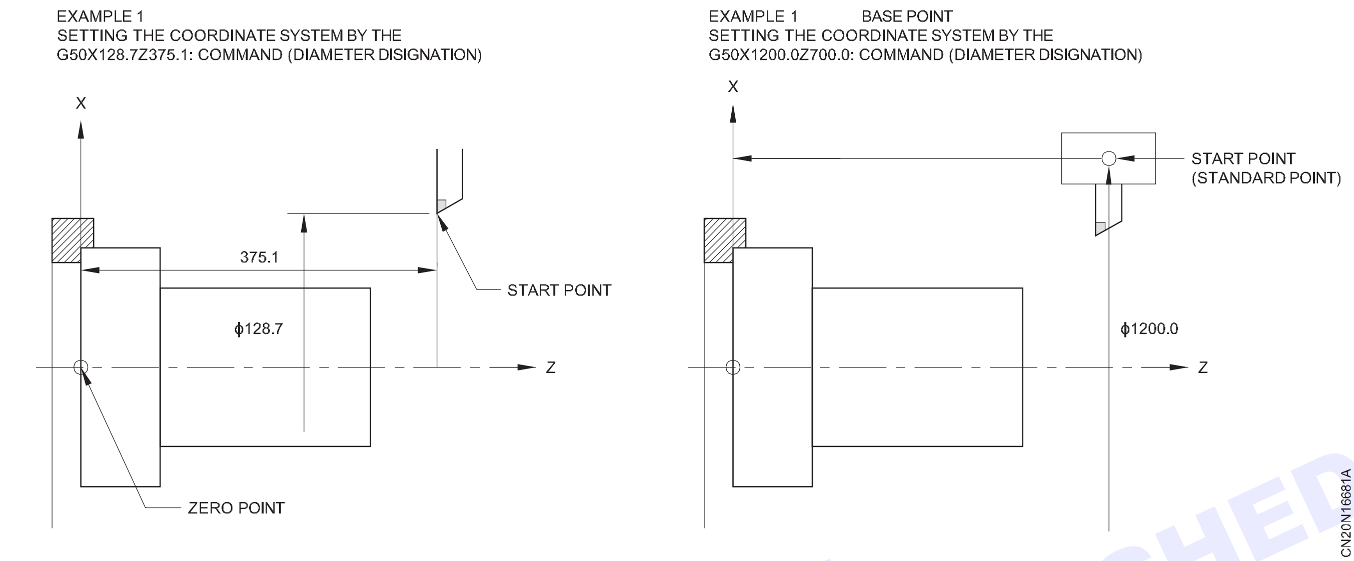
Then, use program commands G54 to G59 to select which work piece coordinate system to use. When an absolute command is used, a work piece coordinate system must be established in any of the ways described above.

Setting a work piece coordinate system by G50

G50 IP_

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

Fig 10



Selecting a work piece coordinate system

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

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

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

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

G54 Work piece coordinate system 1

G55 Work piece coordinate system 2

G56 Work piece coordinate system 3

G57 Work piece coordinate system 4

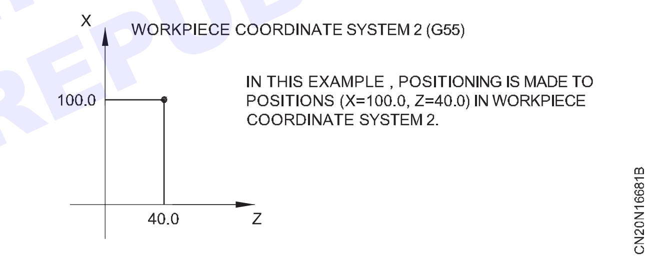
G58 Work piece coordinate system 5

G59 Work piece coordinate system 6

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

Examples

Fig 11



Programming, Zero Points

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

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

Stock removal in CNC turning external and internal

Objectives: At the end of this lesson you shall be able to
- **state which G code is used for stock removal in CNC program**
- **describe the format for the G71 code**
- **explain the external stock removal cycle program**
- **brief the internal stock removal program.**

Stock removal in CNC turning

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

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

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

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

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

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

Parameters

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

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

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

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

W: Amount left for finishing in the Z-axis

D: Depth of roughing cut

F: Cutting feed rate (in/rev or mm/rev) overrides feed rates between P block and Q block

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

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

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

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

G71 U ... R .. .

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

Parameters

First block

U: Depth of roughing cut

R: Amount of retract from each cut

Second block

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

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

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

W: Amount left for finishing in the Z-axis

F: Cutting feed rate (in/rev or mm/rev) overrides feed rates between P block and Q block

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

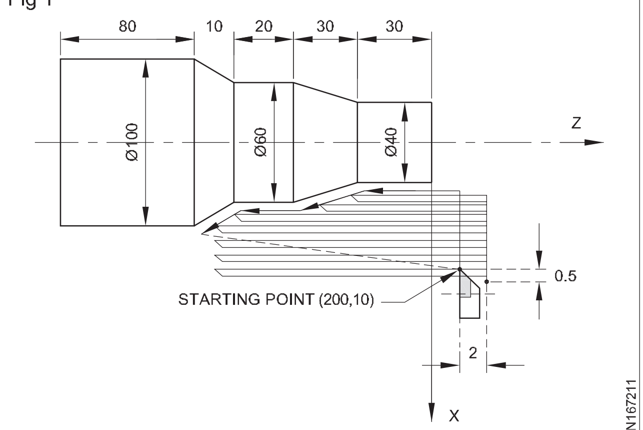
Note: Do not confuse address U in the first block, depth of cut per side, and address U in the second block, stock left on diameter. The I and K parameters may be used only on some controls and the retract amount R is set by a system parameter

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

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

Example program for external stock removal (Fig 1)

Fig 1



ROUGH TURNING CYCLE

00004

G00 X200 Z10 M3 S800

G71 U2 R1 F 200

G71 P80 Q120 U0.5 W0.2

N80 G00 X40 S1200

G01 Z-30 F100

X60 W-30

W-20

N120 X100 W-10

G70 P80 Q120

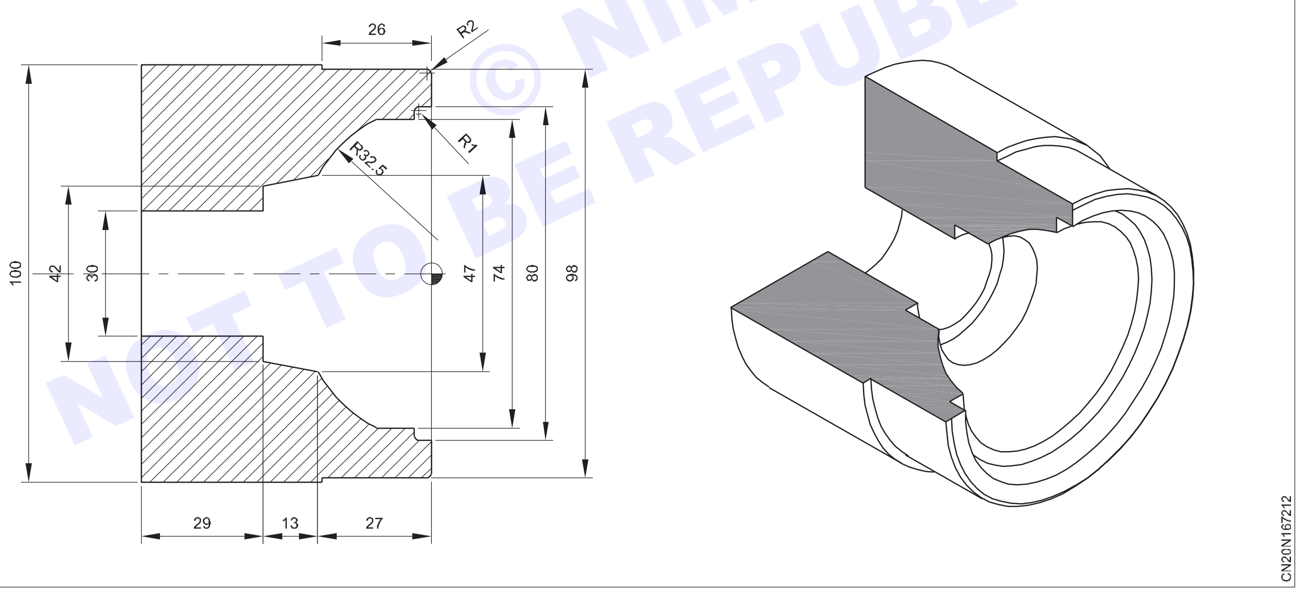
M30

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

G71 cycle example

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

Fig 2



03000;

N05 T0101;

N10 M4 S1800;

N15 G0 X102 Z0 M8;

N20 G71 U2 R1;

N25 G71 P30 Q45 U0.6 W0.2 F0.25;

N30 G1 X94;

N35 G3 X98 Z-2 R2;

N40 G1 Z-26;

N45 G1 X100;

N50 G70 P30 Q45;

N55 G0 X200 Z200 M9;

N60 T0808;

N65 M4 S1500;

N70 G0 X28 Z5 M8;

N75 G1 Z0 F0.15;

N80 G71 U1 .5 R1;

N85 G71 P90 Q130 U-0.4 W0.2 F0.15;

N90 G1 X80 F0.05;

N95 G1 Z-3;

N100 G3 X78 Z-4 R1;

N105 G1 X70;

N110 G1 Z-9;

N115 G3 X47 Z-27 R32.5;

N120 G1 X42 Z-40;

N125 G1 X30;

N130 G1 Z-69;

N135 G70 P95 Q130;

N140 G0 Z200 M9;

N145 G28 U0 W0;

N150 M30;

Things to remember

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

Canned cycles

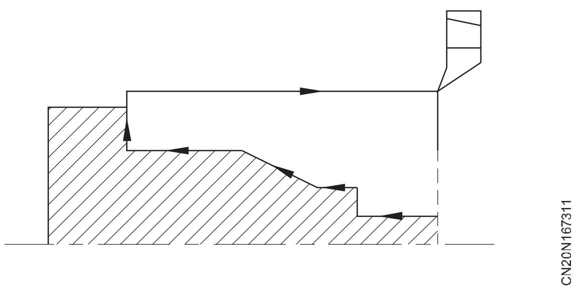
Objectives: At the end of this lesson you shall be able to
- **state the cycle used in CNC program**
- **learn to program for all canned cycles**
- **learn to program for threading OD/ID.**

Canned cycles

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

G70 - finishing cycle (Fig 1)

Fig 1



Format

G70 P_Q_F_;

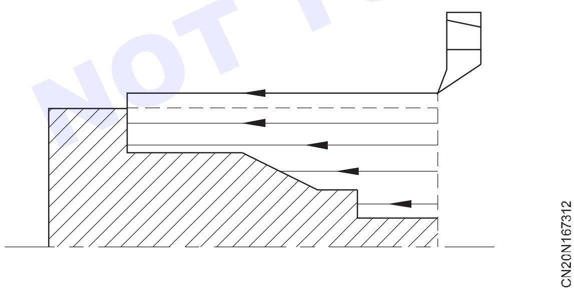
P : Starting Block Number

Q : Ending Block Number

F : Finishing Feed

G71 - turning cycle (Fig 2)

Fig 2



Format

G71 U_R_;

G71 P_Q_U_W_F_;

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

R : Relief Amount

P : Starting Block Number

Q : Ending Block Number

U : Finishing Allowance in X Axis

W : Finishing Allowance in Z Axis

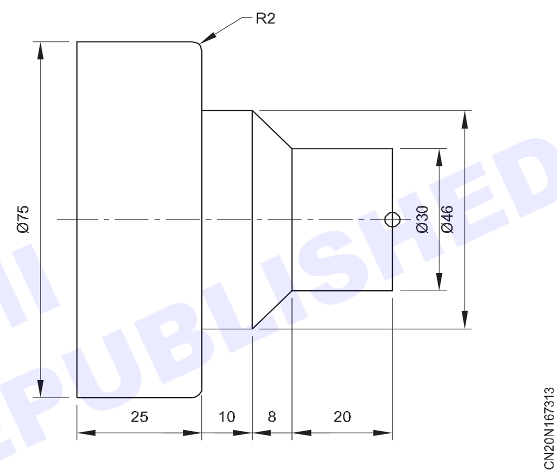
F : Feed

Example for G71 turning cycle

%

O0026 (Fig 3)

Fig 3



G28 U0 W0 ;

G50 S1200 T0300 ;

G96 S250 M03 ;

G00 X65.0 Z0.0 T0303 M08 ;

G01 X0.0 F0.1 ;

G00 X61.0 Z2.0 ;

G71 U1.0 R1.0 ;

G71 P10 Q20 U1.0 W0.5 F0.3 ;

N10 G00 X30.0 ;

G01 Z-20.0 F0.1 ;

X46.0 Z-28.0 ;

Z-38.0 ;

G1 X71 ;

G03 X75.0 Z-40 R2 ;

G01 Z-63 ;

N20 X80.0 ;

G00 X 80.0 Z2.0 M09 ;

G28 U0 W0 M05 ;

M01 ;

N2 ; (OD FINISHINING)

G28 U0 W0 ;

G50 S2600 T0700 ;

G96 S150 M03 ;

G00 X26.0 Z0.0 T0705 M08 ;

G01 X0.0 F0.1 ;

G00 X65.0 Z3.0 ;

G70 P10 Q20 F0.12 ;

G00 X80.0 Z2.0 M09 ;

G28 U0 W0 ;

M30 ;

G72 - facing cycle

Format

G72 W_R_;

G72 P_Q_U_W_F_;

W : Depth of cut per pass in Z Axis

R : Relief Amount

P : Starting Block Number

Q : Ending Block Number

U : Finishing Allowance In X Axis

W : Finishing Allowance In Z Axis

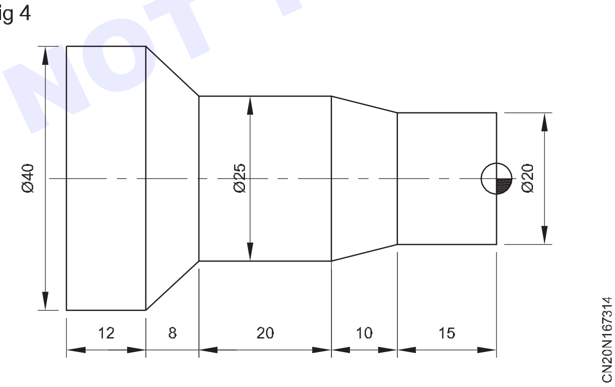
F : Feed

Example for G72 facing cycle

%

O0027 (Fig 4)

Fig 4



N1 ;

G28 U0 W0 ;

G50 S1300 T0200 ;

G96 S150 M03 ;

G00 X55.0 Z2.0 T0202 M08 ;

G72 W1.0 R2 ;

G72 P20 Q30 U1.0 W0.5 F0.3 ;

N20 G00 Z-65.0;

G01 X40 ;

Z-53.0 ;

X25.0 Z-45.0 ;

Z-25.0

X20.0 Z-15.0 ;

Z0.0 ;

N30 ;

G00 X55.0 Z2.0 M09 ;

G28 U0 W0 M05 ;

M01 ;

N2 ; (OD Finishing)

G28 U0 W0 ;

G50 S2600 T0700 ;

G96 S150 M03 ;

G00 X55.0 Z2.0 T0707 M08 ;

G70 P20 Q30 F0.12 ;

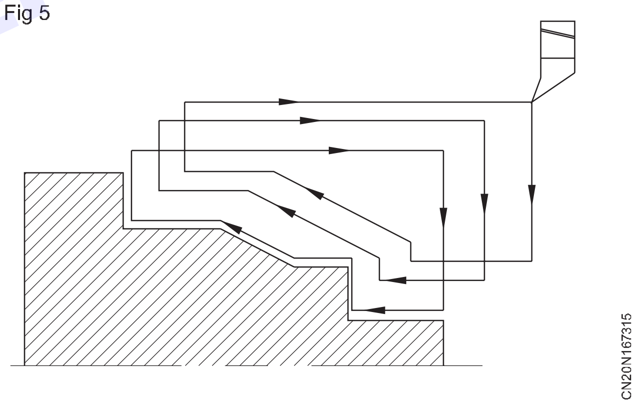
G00 X65.0 Z2.0 M09 ;

G28 U0 W0 M05 ;

M30 ;

G73 - Pattern repeating cycle (Fig 5)

Fig 5



Format

G73 U_W_R_;

G73 P_Q_U_W_F_;

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

W : Total amount of stocks in Z axis

R : Number of passes

P : Starting block number

Q : Ending block number

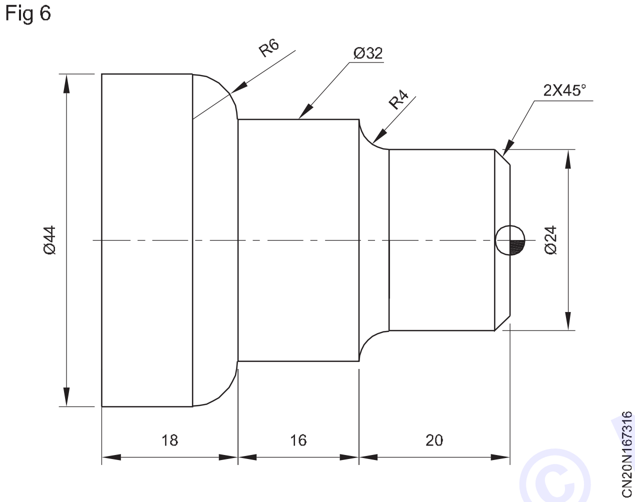
U : Finishing allowance In X axis

W : Finishing allowance in Z axis

F : Feed

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

Fig 6




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

G74 - Peck drilling cycle

Format

G74 R_;

G74 Z___Q___F___;

R : Retract value

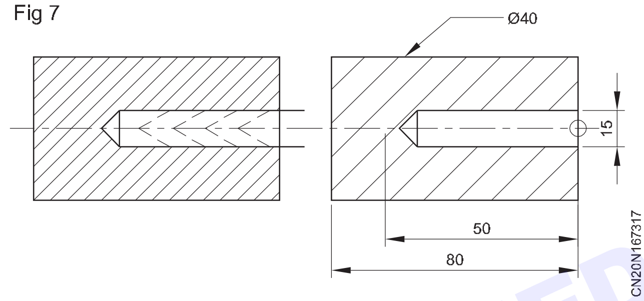
Z : Depth of the hole

Q : Depth of cut per pass in Microns

F : Feed rate

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

Fig 7




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

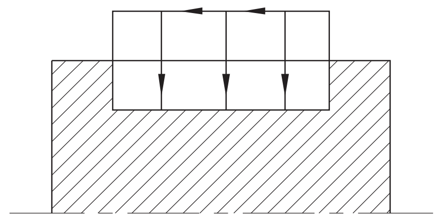
G75 - Grooving cycle (Fig 8)

Format

G75 R___;

G75 X____Z___P____Q___R___F___;

Fig 8



CN20N167318

R : Relieving the tool (mm)

X : Groove diameter (mm)

Z : Groove length (mm)

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

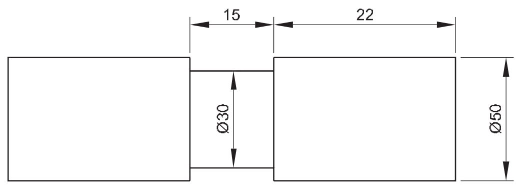
Q : Shift value in Z axis (microns)

F : Feed

%

O0030 (Fig 9)

Fig 9



CN20N167319

G28 U0 W0 ;

T0500 ;

G97 S1200 M03 ;

G00 X52.0 Z5.0 T0505 M08 ;

G00 Z-26.0 ;

G75 R2.0 ;

G75 X30.0 Z-37.0 P500 Q3000 F0.05 ;

G00 X46.0 ;

Z5.0 M09 ;

G28 U0 W0 M05 ;

M30 ;

G76 - Multiple thread cutting cycle(Fig 10)

Format:

G76 P___Q____R___;

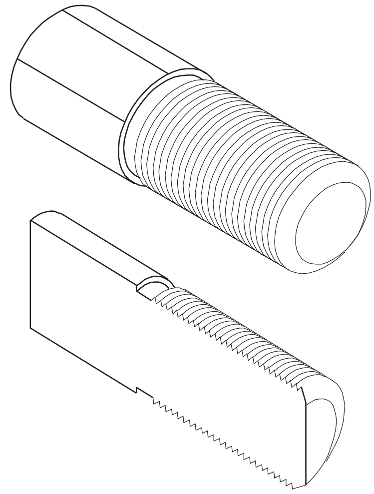
G76 X___Z___P___Q____F____;

Explanation for the cycle:

P : NCA

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

Fig 10



CN20N16731A

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

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

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

Z : Thread length

P : Height of thread (microns)

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

F : Feed (pitch of the thread)

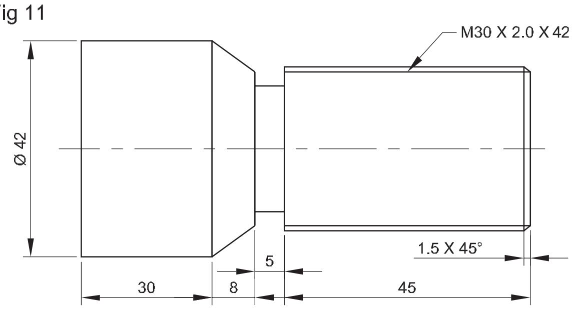
Example

OD Threading

%

O0031 (Fig 11)

Fig 11



OD THREADING

CN20N16731B

G28 U0 W0 ;

T0400 ;

G97 S600 M04 ;

G00 X32.0 Z5.0 T0404 M08 ;

G76 P030060 Q150 R20 ;

G76 X 27.54 Z-42.0 P1226 Q300 F2.0 ;

G00 X32.0 ;

Z5.0 M09 ;

G28 U0 W0 M05 ;

M30 ;

Threading calculation

Minor dia, d = D - (2h)

(h = 0.6134 p, for metric thread)

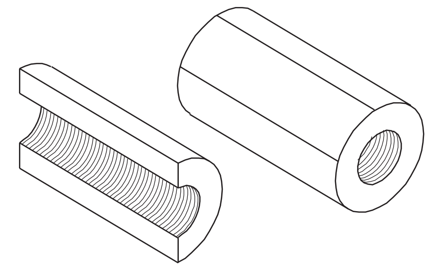
h = 0.6134 2.0=1.226 mm.

d = 30 - (2 x1.226)

= 27.54 mm.

ID Threading (Fig 12)

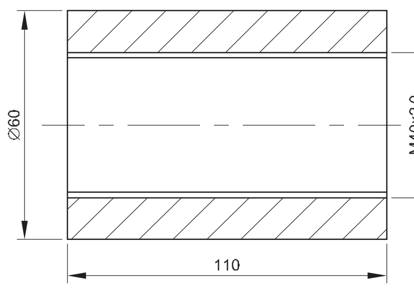
Fig 12



%

O0032 (Fig 13)

Fig 13




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

Threading calculation

Minor dia, d = D - (2h)

(h = 0.6134 P, for metric thread)

h = 0.6134 x 2 = 1.226 mm

d = 40 - (2 X 1.226)

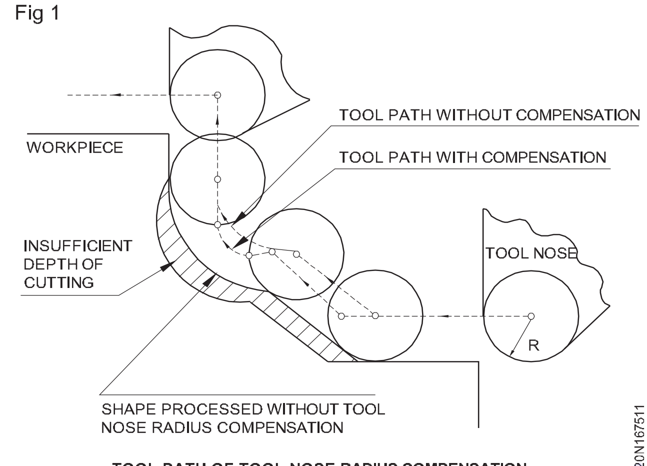
=37.548 mm.

Tool Nose Radius compensation (TNRS)

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

Overview of tool nose radius compensation (Fig 1)

Fig 1

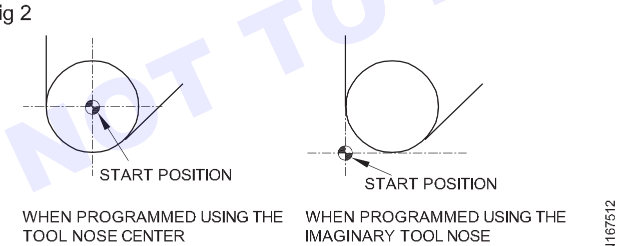


TOOL PATH OF TOOL NOSE RADIUS COMPENSATION

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)

Fig 2



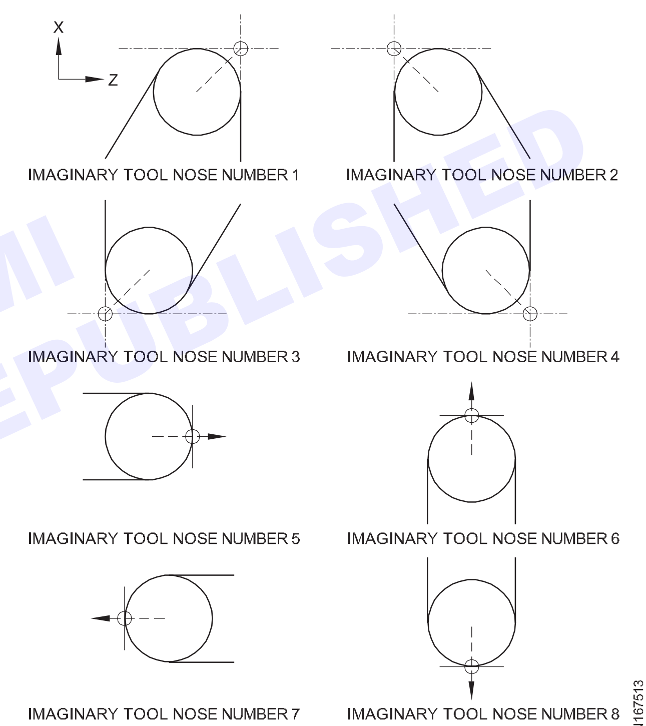
TOOL NOSE RADIUS CENTER AND IMAGINARY TOOL NOSE

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 Fig 2.

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.

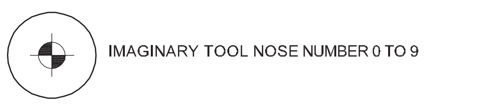
Fig 3



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. (Figs 5 & 6)

Fig 5

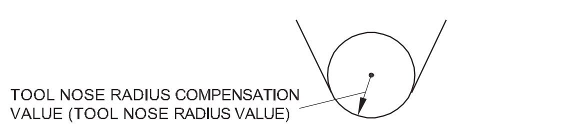
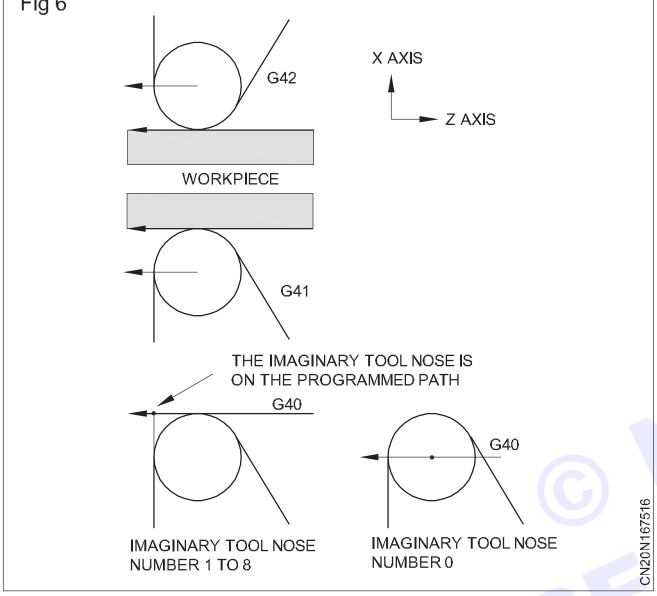


Table 1 (Tool geometry offset)

Geometry offset number	OFGX (X-axis geometry offset)	OFGZ (Z-axis geometry offset)	OFGR (Tool nose radius geometry offset)	OFT (imaginary tool nose direction)	OFGY (Y-axis geometry offset amount)
G01	10.040	50.020	0	1	70.020
G02	20.060	30.030	0	2	90.030
G03	0	0	0.20	6	0
G04	:	:	:	:	:
G05	:	:	:	:	:
:	:	:	:	:	:

Fig 6



Position the cursor at appropriate place and input the values

The tool nose radius compensation value during execution is the sum of the geometry offset and the wear offset.

Work position and Move command

In tool nose radius compensation, the position of the work piece with respect to the tool must be specified (Fig 7& 8)

Fig 7

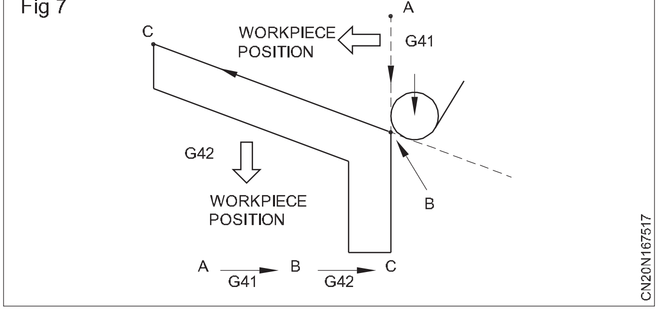
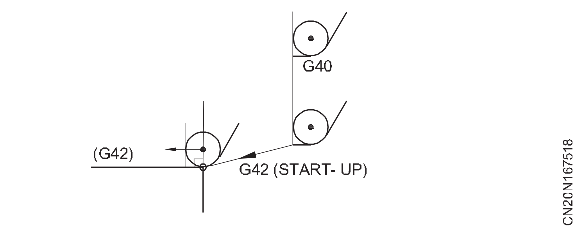


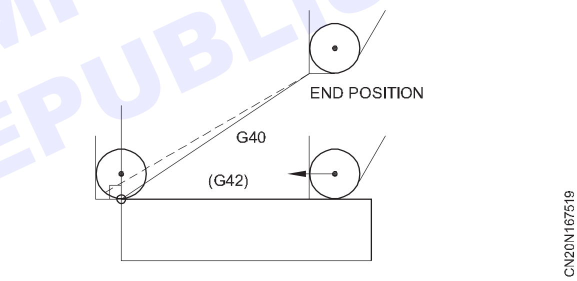
Fig 8



Tool movement when the work piece position changes

The work piece position against the tool changes at the corner of the programmed path as shown in Fig 9.

Fig 9



Although the work piece does not exist on the right side of the programmed path in the above case. the existence of the work piece is assumed in the movement from A to B. The work piece position must not be changed in the block next to the start-up block. In the above example, if the block specifying motion from A to B were the start-up block, the tool path would not be the same as the one shown.

Start-up

The block in which the mode changes to

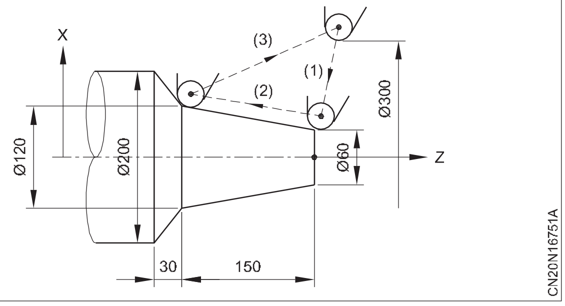
G41 or G42 from G40 is called the start-up block.

G40 _ ;

G41 _ ; (Start-up block)

Transient tool movements for offset are performed in the start-up block. In the block after the start-up block, the tool nose centre is positioned Vertically to the programmed path of that block at the start position (Fig 10)

Fig 10



Offset cancel

The block in which the mode changes to G40 from G41 or G42 is called the offset cancel block. G41 _ ; G40 _; (Offset cancel block) The tool nose centre moves to a position vertical to the programmed path in the block before the cancel block. The tool is positioned at the end position in the offset cancel block (G40)

Examples

(G40 mode)

1 G42 G00 X60.0 ;

2 G01 X120.0 W-150.0 F10 ;

3 G40 G00 X300.0 W150.0 I40.0 K-30.0 ;

Introduction to the computer aided manufacturing software and concept of toolbar & ribbon setting attribute & user interface orientation

Objectives: At the end of this lesson you shall be able to
- **understand the concept and purpose of Computer-Aided Manufacturing (CAM) software**
- **describe the basic commands used in CAM software**
- **identify and use various components of the toolbar and ribbon interface**
- **state about attributes and navigate the user interface effectively.**

Introduction

Computer-aided manufacturing (CAM) is the use of software to control machines and the related devices and equipment in the manufacture of components. The purpose of CAM is to increase productivity and improve tooling, to enhance greater consistency of quality, minimizing raw material wastage. CAM software is also useful for training and academic educational purposes.

CAM processes have come out of CAD and sometimes CAE computer-aided engineering, as a model generated by CAD, verified by CAE and put into CAM software, which controls the CNC machine tool.

CAM is a numerical control (NC) programming tool, for 2D/3D models generated in CAD. CAM does not eliminate the need for skilled professionals, but it assists manufacturing personnel in building skills through visualization, simulation, and optimization tools. Integration of CAD with other components of CAD/CAM/CAE, Product Life Cycle Management (PLM) environment requires an effective CAD data exchange, such as IGES/STL/Parasolid formats. The output from CAM is usually a text file of G-code/M-code, add thousands of commands, which can be transferred to a CNC program.

CAM is specifically effective in typical areas like high-speed machining, streamlining tool paths, multifunction machining, 5-axis machining, and automatic machining.

A finished component has to undergo different machining processes in its conversion from raw material i.e.,

a **Rough machining:** From raw stock, raw casting passes through zig-zag clearing, plunge roughing, rest roughing, trochoidal milling aiming at maximum material removal in the least amount of time, maintaining dimensional accuracy.

b **Semi-finishing:** In this process, a small amount of material is only removed, to enable the tool to cut accurately such as Raster passes, constant step-over process, pencil milling, etc.

c **Finishing machining:** It involves very light material removal to produce a polished finished product. The feed is increased to target SFM with high speed referred as high-speed machining (HSM).

Usefulness of CAM: Good CAM software is equally as important to manufacturers as the powerful machines and the tools they use to cut desired parts. Machine shops of all sizes and budgets are reaping benefits of good CAM software beyond efficiency programming their machining jobs. Users can structure their job, set their Toolpath, then use the simulation function to make sure their plan goes accordingly. Part gouges or collisions can potentially ruin a very crucial and expensive CNC machine, threatening to affect that shop's profitability or ability to take on additional projects. It is much easier to readjust a toolpath in CAM software than it is to fix or replace a CNC machine.

Structuring your job with CAM software

Let's take a sprocket, for example. You created your sprocket using CAD software, now how will you make it? CAM software looks at what designed & determines how to machine it out of materials. For starters, most CAM software products provide a standard "Job Tree" for machining strategy organization. Then you will need to set up and save the features of the machine from your shop within the CAM software. This is important for developing programs that are specific to your machine, allowing you to easily machine future projects, editing machine settings as needed. Next, you will identify your stock, allowing you to set initial work coordinates, material type, and the tools to be used during machining. Lastly, you set your cutting conditions, tool patterns, tool crib, and tool holder for error-free CNC programming. One of the greatest benefits of CAM software is the ability to save the information you put in the system, making future projects much more easily programmed.

Industrial applications of Computer-Aided Manufacturing (CAM)

1 Automotive industry

In the car-making industry, CAM is used to produce important parts like engines and gearboxes with great accuracy. It helps make sure that every piece is made correctly and quickly. CAM also helps in building new car designs faster by creating models and prototypes, which speeds up the whole process of bringing new cars to the market.

2 Aerospace industry

For airplanes, CAM is vital because the parts, like turbine blades and wing sections, need to be incredibly precise. CAM helps make these parts lighter and stronger, which is essential for safety and efficiency. Using CAM also saves time and money by automating the production of these complex parts.

3 Electronics industry

CAM is crucial in making electronic gadgets. It's used to design and cut tiny parts for Printed Circuit Boards (PCBs), which are found in almost every electronic device. CAM also helps create the outer cases of electronics, making sure everything fits perfectly.

4 Medical industry

In healthcare, CAM is used to make custom items like implants, artificial limbs, and dental braces that fit patients perfectly. It also helps in creating precise surgical tools, ensuring doctors have reliable instruments during operations.

5 Tool and die making

CAM is important for making molds and dies, which are used to shape materials in various manufacturing processes. These molds need to be very precise, and CAM makes sure they are made exactly to the needed specifications without much manual work.

6 Consumer goods

When it comes to everyday products like appliances, toys, and electronics, CAM helps in designing and mass-producing these items quickly and accurately. It also plays a role in packaging by creating the tools needed to make product packages.

7 Furniture industry

In furniture-making, CAM helps create custom designs and shapes. It allows machines like CNC routers to cut and shape wood parts precisely, reducing waste and making the process more efficient. This means manufacturers can easily meet specific customer requests.

List of some popular CAM software

1 Mastercam

One of the most widely used CAM software, known for its user-friendly interface and extensive toolpath options for machining.

2 SolidCAM

Integrated directly within CAD software like SolidWorks, SolidCAM offers powerful machining solutions and seamless workflow.

3 Fusion 360

A cloud-based platform that combines CAD, CAM, and CAE, making it ideal for both design and manufacturing processes.

4 Autodesk FeatureCAM

Known for automating CAM programming, it helps reduce programming time by recognizing features of the part automatically.

5 NX CAM

Developed by Siemens, NX CAM provides comprehensive solutions for complex machining, used extensively in high-precision industries like aerospace.

Mastercam interface: Mastercam's ribbon interface is based on standard Windows design guidelines. The ribbon comprises familiar controls, including a Quick Access Toolbar (QAT), tabs, contextual tab groups, galleries, buttons, and the Backstage. It also features special on-screen controls, and movable, dockable Managers and function panels.

Start Mastercam using your preferred method

- Double-click Mastercam's desktop icon. (Fig 1)

Fig 1

Or,

- Launch Mastercam from the Windows Start menu.

The following image displays the elements that make up the Mastercam interface. The corresponding sections provide brief descriptions of each element and its use. (Fig 2)

1 **Quick Access Toolbar (QAT):** A customizable set of commonly used functions that are always available in the interface. The QAT can be located above or below the ribbon.

2 **Tab:** Groupings of related controls. Tabs are organized in a simple-to-complex workflow from left to right.

3 **Tab group:** A region of the tab that contains a set of related controls.

4 **Contextual tab:** A tab that displays when you make a specific selection in Mastercam. A contextual tab presents controls and commands relevant to your current activity.

5 **Tooltip:** A small window with descriptive text that displays when you hover over a command or control.

6 **Selection bar:** A toolbar that combines AutoCursor controls and general selection tools used to select entities in the graphics window. There are two selection modes, Standard Selection and Solid Selection, which are activated based on the function you are using. AutoCursor controls allow you to detect and snap to locations as you move the cursor over geometry in the graphics window. AutoCursor becomes active whenever Mastercam prompts you to select a position in the graphics window.

7 **Quick masks:** A group of controls that helps you to select all entities of a certain type, or to select only entities of certain type. Most Quick Mask controls are divided in half. Clicking the left or right side of the

control toggles between selection modes. When a Quick Mask control is selected it highlights to indicate the selection mask is on. You can toggle multiple Quick Masks at a time.

8 **Status bar:** A bar across the bottom of the workspace that shows the coordinate position of the cursor and provides quick access to modify planes, construction mode, and Z depth. The right side of the Status Bar has wireframe, shading, and translucency controls that change the appearance of your part. The left side of the Status bar indicates the number of selected entities and the status of Section View. (Fig 3)

Fig 2

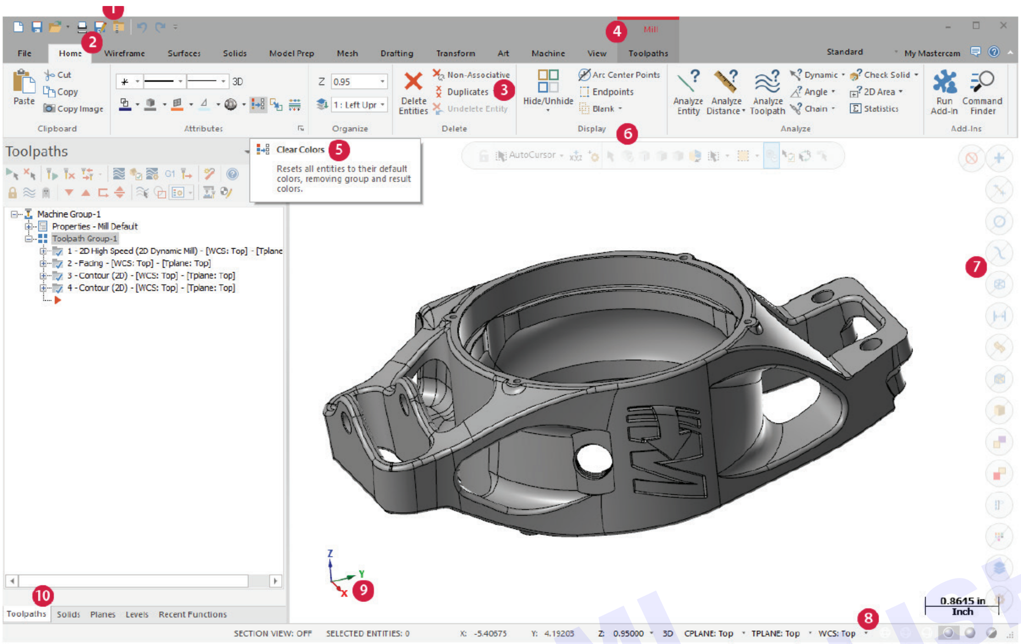
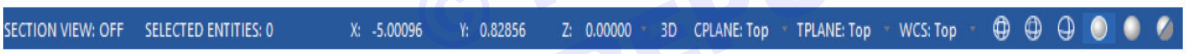
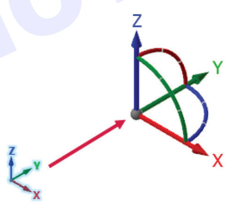


Fig 3



9 **Dynamic Gnomon:** The on-screen gnomon allows you to manipulate views and planes interactively. The gnomon comprises three axes connected at the origin, with selection points that let you choose different types of transformations. (Fig 4)

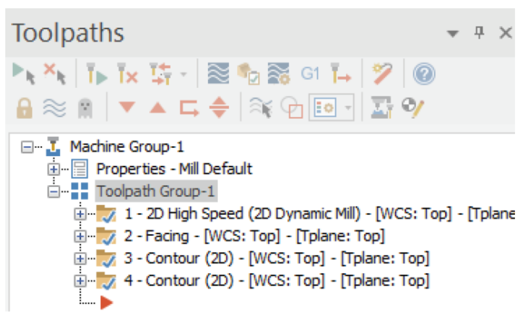
Fig 4



10 **Managers:** Managers include controls for toolpaths, solids, planes, levels, recent functions, and Art. The Managers can be rearranged to suit your workflow.

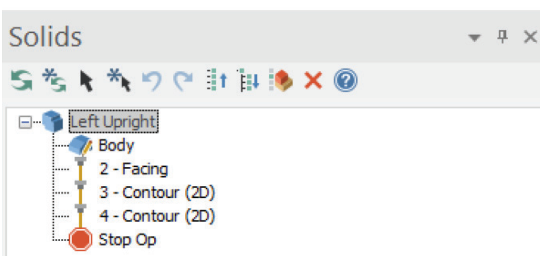
- **Toolpaths manager:** Lists the toolpath groups and machine types for the current file. Use the Toolpaths Manager to control your operations. (Fig 5)

Fig 5



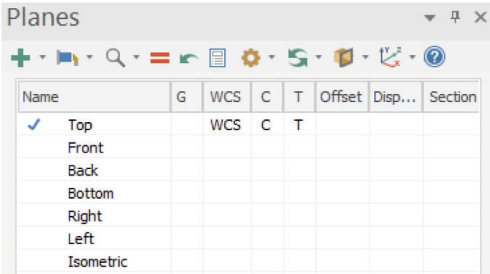
- **Solids manager:** Lists each solid in the current file along with its operation history and associated toolpaths. Use the Solids Manager to edit solids and their operations. (Fig 6)

Fig 6



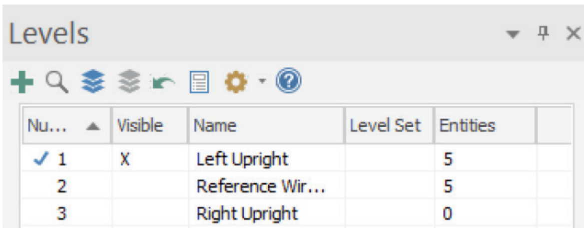
- **Planes manager:** Shows the planes in the current file. Use the Planes Manager to adjust planes and work offsets. (Fig 7)

Fig 7



- **Levels manager:** Shows the levels in the current file. Use the Levels Manager to control your levels. (Fig 8)

Fig 8



11 **Graphics window:** The space in which you view, create, and modify your parts. The graphics window also displays information about the current units (inches or millimeters), and the coordinate axes for the current view or plane.

Common Interface controls

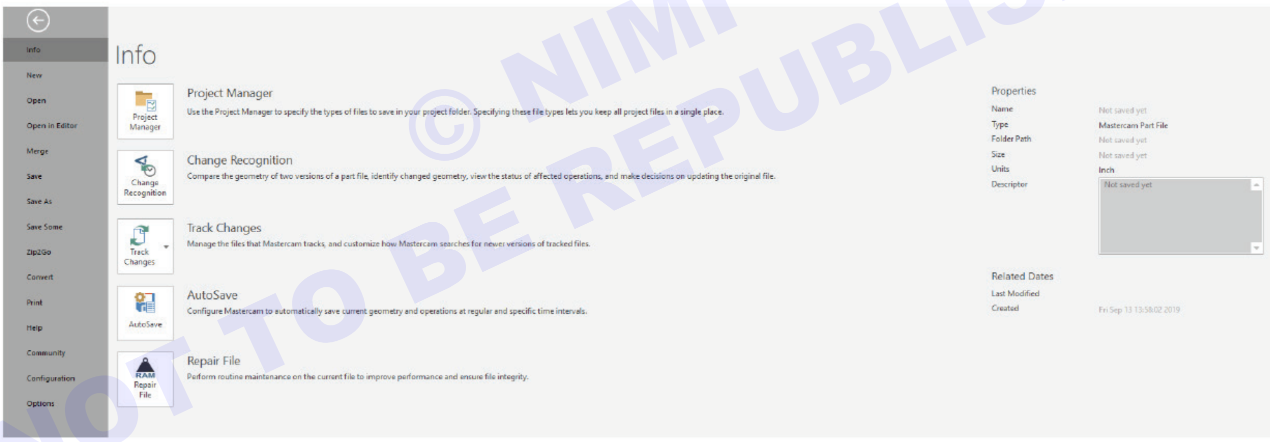
The following table shows common controls used in the Mastercam interface. The icons may vary slightly depending on their location in Mastercam. (Fig 9)

Fig 9

Icon	Option	Icon	Option
	OK and Create New		Cancel
	OK/Save		Help
	Apply		About Mastercam

The File drop-down in Mastercam is also known as the Backstage. Click File to open the Backstage view. The image below shows the Info page of the Backstage with links on the left side to other pages and functions. (Fig 10)

Fig 10



The following list provides brief descriptions of some of the functions and pages located in the Backstage. Click each page to explore the Backstage.

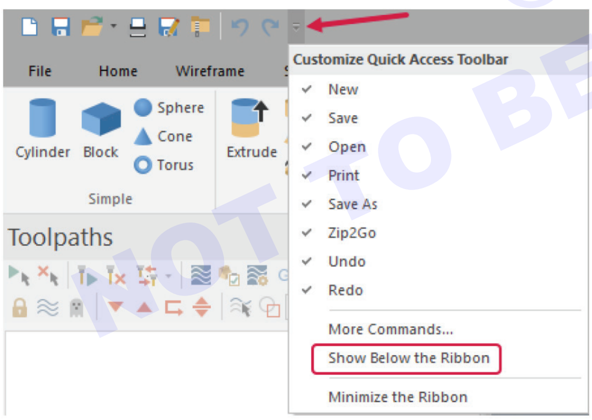
- **Info:** Presents information and properties of the current Mastercam file. The Info page gives you access to functions that take action on the currently open file, including: Project Manager, Change Recognition, Track Changes, AutoSave, and Repair File.
- **New:** Opens a new file.
- **Open:** Accesses the Open page where you can select a recent document or browse for a file to open. You can pin frequently used files or folders to the Recent Documents and Recent Folders sections.
- **Open in Editor:** Opens a file using a file editor of your choice.
- **Merge:** Imports and merges entities from an existing part file into the current file.
- **Save/Save As:** Saves the current part file or saves the file with a new name. You can pin frequently used folders to the Recent Folders section of the Save As page.
- **Save some:** Saves only entities that you select in the graphics window.
- **Zip2Go:** Opens the Zip2Go Wizard. The Zip2Go Wizard compresses the currently open Mastercam part and its associated files (configuration, machine definition, post files, tool libraries, etc.) into a .Z2G, .ZIP, or .MCAM- CONTENT file. A Zip2Go file is helpful when sharing file information with other users or Technical Support.

- **Convert:** Provides access to Import and Export functions, as well as the Migration Wizard so that you can update older Mastercam files to the latest version.
- **Print:** Previews, configures, and prints the current Mastercam file.
- **Help:** Presents information about Mastercam and licensing. Includes links to the latest What's New and Resolved Issues, tutorials, and the Help system. Visit this page to check for software updates. Access to some of these resources requires an active Internet connection.
- **Community:** Links your account to Mastercam.com, Mastercam University, the Mastercam Community App, Mastercam Forums, Mastercam Knowledge Base, and the Customer Feedback Program. These resources require an active internet connection.
- **Configuration:** Opens the System Configuration dialog box where you set system defaults for Mastercam.
- **Options:** Opens the Options dialog box where you customize the Mastercam interface.

Customizing the Quick Access Toolbar: The Quick Access Toolbar (QAT) is a collection of frequently used functions. The QAT is always available and can be displayed above or below the ribbon. You can add to or remove functions from the QAT.

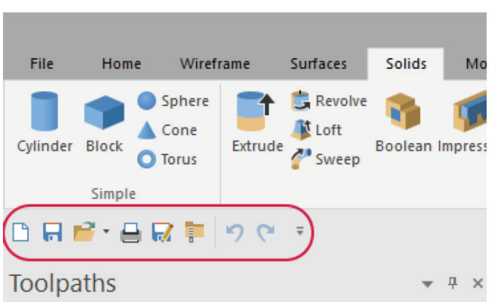
1 Click the Customize Quick Access Toolbar drop-down and select Show Below the Ribbon. (Fig 11)

Fig 11



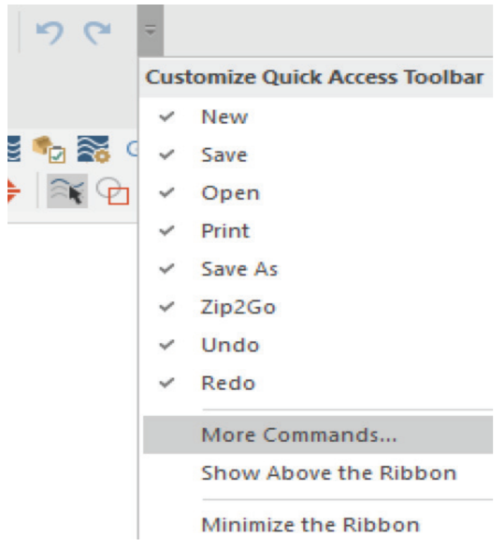
The QAT displays between the ribbon and the managers. (Fig 12)

Fig 12



2 Click the drop-down again and select More Commands. (Fig 13)

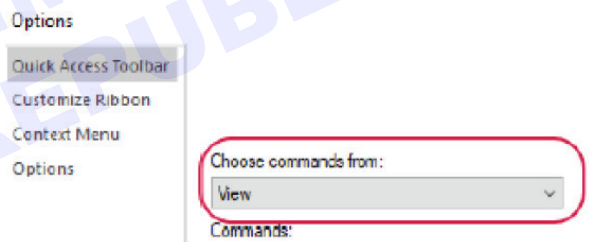
Fig 13



The Options dialog box opens.

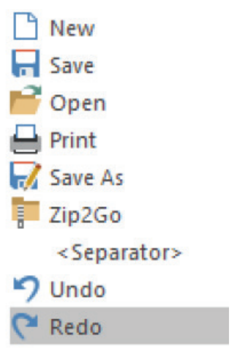
3 On the Quick Access Toolbar page, select View from the Choose commands from drop-down to see a list of commands that appear in the View tab. (Fig 14)

Fig 14



4 Select Redo in the right-hand pane. The command you add to the QAT will appear below this selection. (Fig 15)

Fig 15



5 Select Copy from the Commands list, and then click Add. (Fig 16)

The command appears below the Redo command in the right-hand pane. (Fig 17)

Fig 16

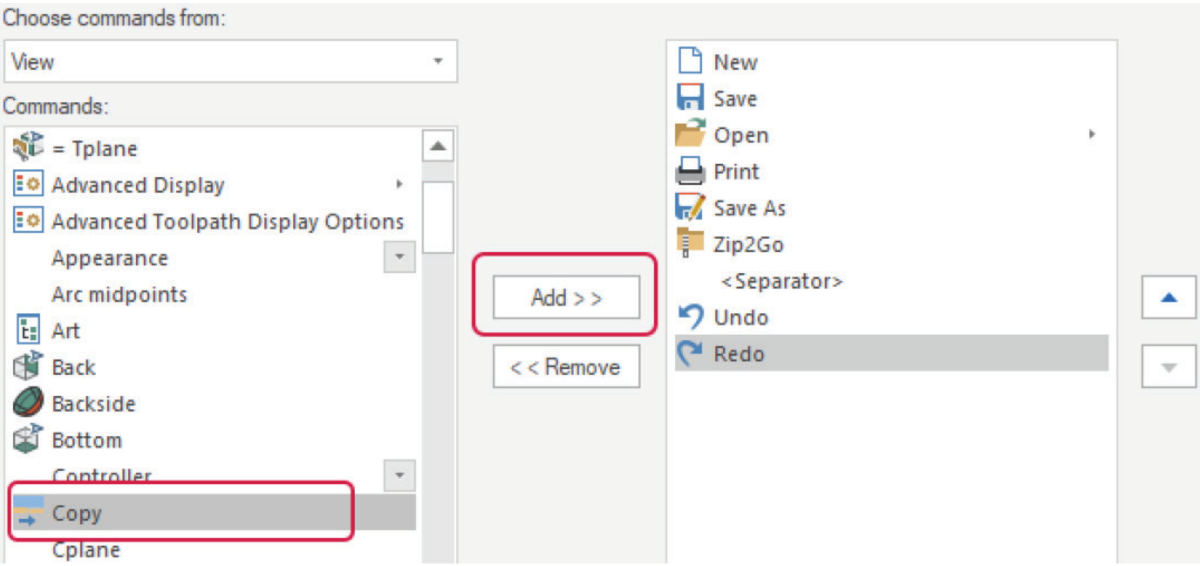
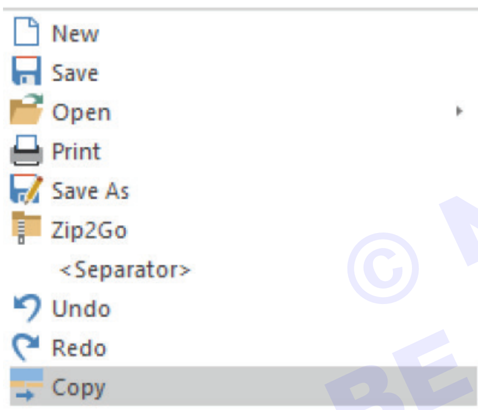
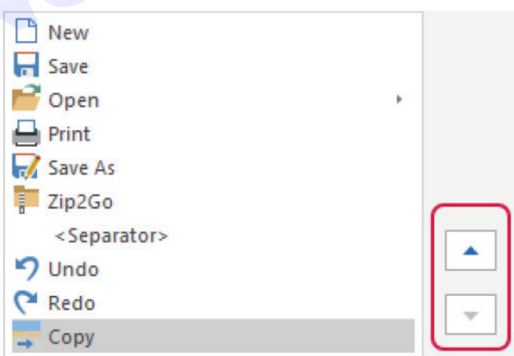


Fig 17



You can use the up and down arrows to rearrange the list. (Fig 18)

Fig 18

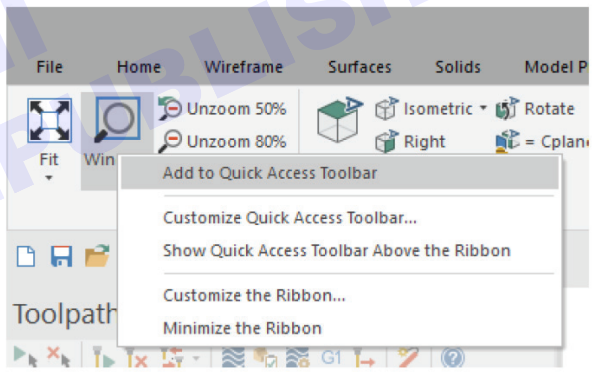


6 Click OK, and the Copy command shows in the QAT. (Fig 19)

Fig 19

Alternatively, you can add any command in the ribbon to the QAT by right clicking the command in the ribbon and selecting Add to Quick Access Toolbar. (Fig 20)

Fig 20



7 Return the QAT to its placement above the ribbon.

Exporting a file

When exporting, you might need to set a few options to guarantee a successful conversion of the file data for your purposes. In most cases, you need only select the software package to which you are exporting the file.

1 Open any Mastercam file. If prompted to save changes to the current Mastercam file, select Don't Save.

2 Select File, Save As, and then click Browse. (Fig 21)

The Save As dialog box opens. (Fig 22)

3 Select the file type you want to export your file as, from the Save as type drop-down list. (Fig 23)

4 Click Save in the Save As dialog box.

Mastercam will save the file the selected format.

Fig 21

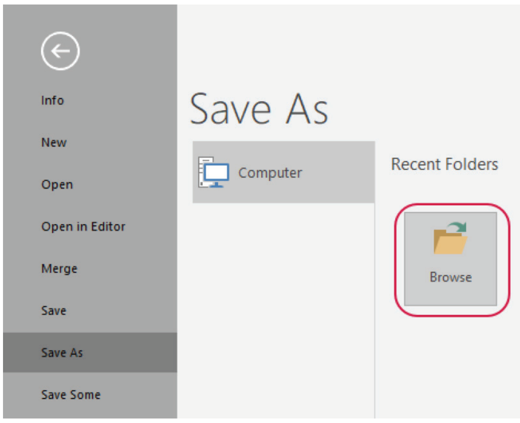


Fig 22

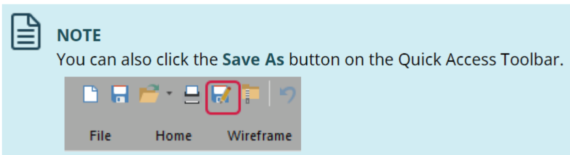
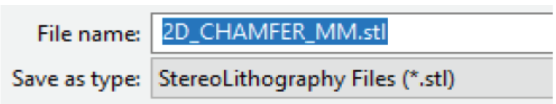


Fig 23



Ribbon interface

The ribbon interface comprises multiple tabs that group together related functions and controls. Tabs are organized in a simple-to-complex workflow from left to right.

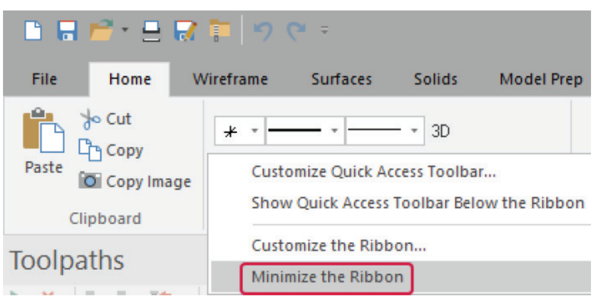
Some tabs are contextual and only display when needed in the workflow. For instance, you must select a piece of wireframe geometry for the Wireframe Selection contextual tab to display on the ribbon.

You can choose to show all available ribbon functions or simplify your view by showing only the most commonly used functions.

Personalize the ribbon tabs

1 Right-click the ribbon and select Minimize the Ribbon. (Fig 24)

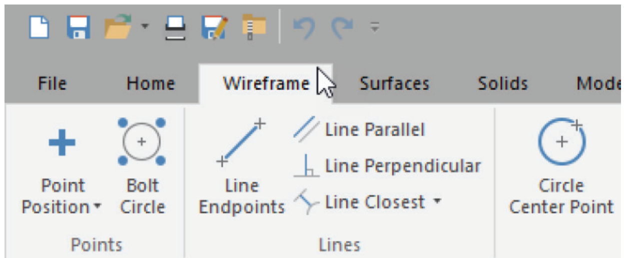
Fig 24



Only the tab names display in the interface.

2 Click any tab to display it. Click in the graphics window to hide the tab again. (Fig 25)

Fig 25

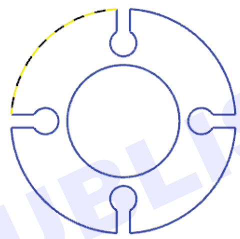


3 To redisplay the full-sized tabs, simply double-click any tab or right-click the ribbon and deselect Minimize the Ribbon.

4 Open any MasterCam file. If you see a message for switching units from inch to metric, click OK to allow the change.

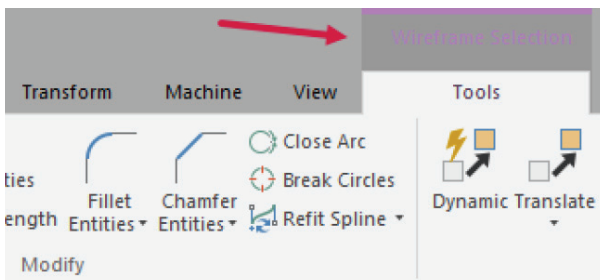
5 Select the geometry as shown below. (Fig 26)

Fig 26



6 The Wireframe Selection contextual tab displays. This tab contains Mastercam functions relevant to your selected wireframe geometry. (Fig 27)

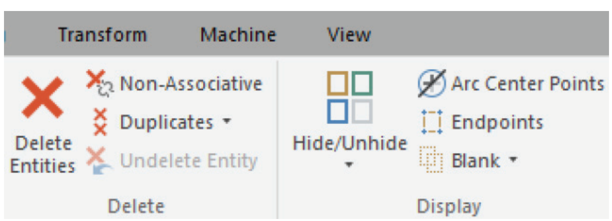
Fig 27



7 Press [Esc] to deselect the geometry in the graphics window.

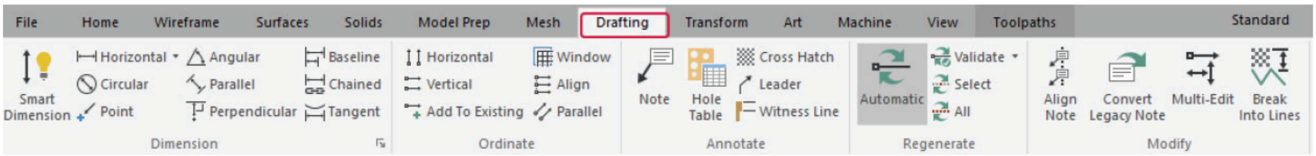
The Wireframe Selection contextual tab disappears from the ribbon. (Fig 28)

Fig 28



8 Select the drafting tab. (Fig 29)

Fig 29

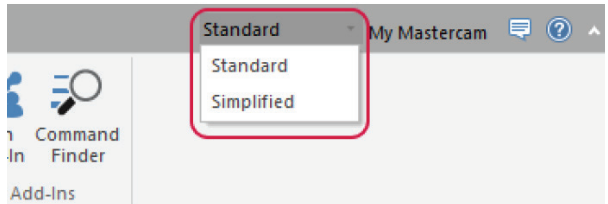


9 Click the ribbon mode drop-down. Ribbon mode alters the number of functions displayed on the ribbon.

- **Standard:** All Ribbon functions display.
- **Simplified:** The most commonly used Ribbon functions display. Some of the more complex Analyze functions, drafting functions, Curve and Line functions, Delete functions, and View options are no longer available. (Fig 30)

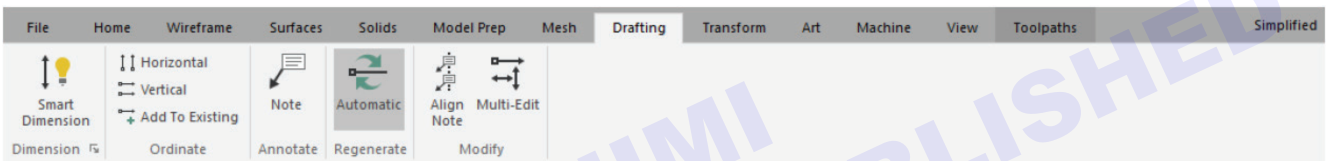
10 Select simplified. The number of functions on the Drafting tab changes. Mastercam remembers your Ribbon mode selection between sessions. (Fig 31)

Fig 30



11 Return ribbon mode to standard.

Fig 31

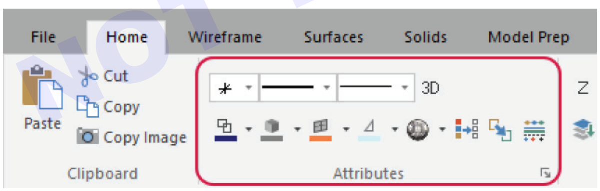


Setting attributes

Entity attributes control point and line styles, as well as the colors associated with entity types you create in Mastercam. You can change the attributes of selected entities in the graphics window two ways:

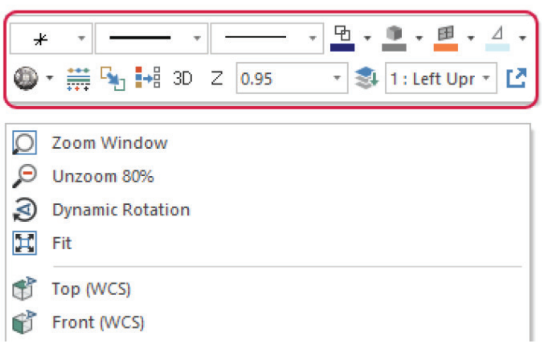
1 Use the controls on the Home tab. (Fig 32)

Fig 32



2 Use the right-click mini-toolbar. (Fig 33)

Fig 33

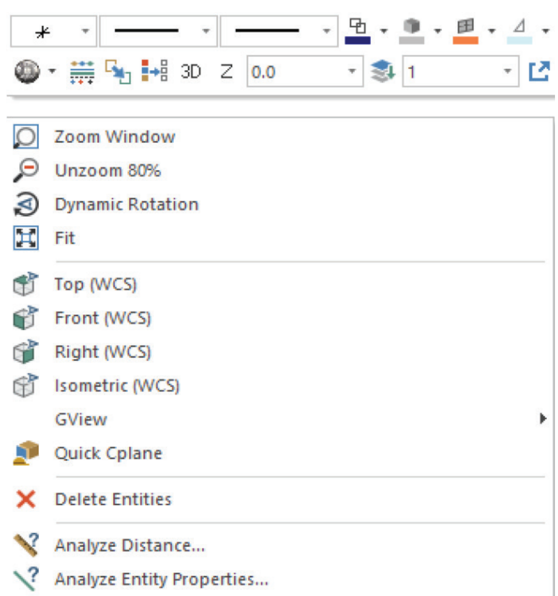


When nothing is selected in the graphics window, you can also set entity attributes for the current file using these controls. To set entity attributes that persist between sessions, use the Colors and CAD pages of the System Configuration dialog box.

1 Select File, New to start with a new part file.

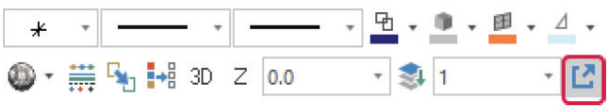
2 Right-click the graphics window to open the mini-toolbar and context menu. (Fig 34)

Fig 34



3 Select Toggle Attributes Panel on the mini-toolbar. (Fig 35)

Fig 35

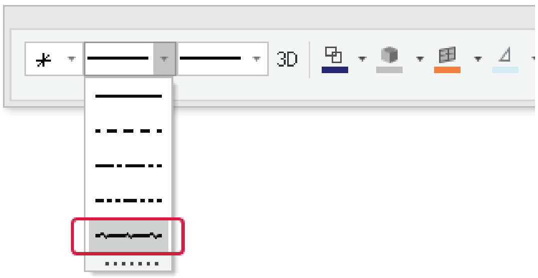


The mini-toolbar now displays independent of the right-click menu. You can resize and position the mini- toolbar anywhere in the graphics window or anywhere on your desktop. (Fig 36)

4 With nothing selected in the graphics window, click the Line Style drop-down, and choose the style shown below. (Fig 37)

Fig 36

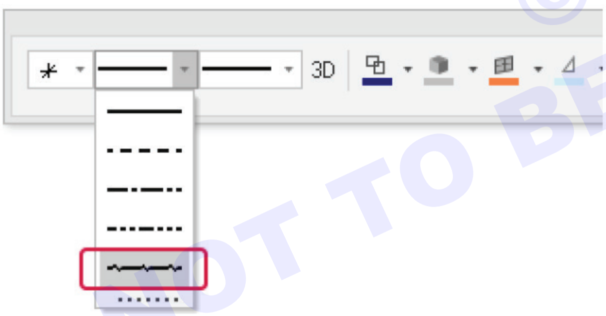
Fig 37



Any new geometry you create uses the line style you select.

5 Select the Line Width shown below. (Fig 38)

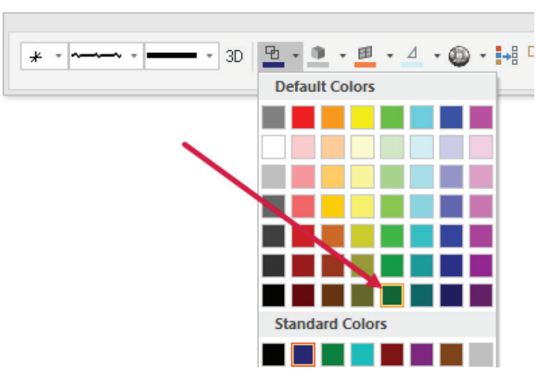
Fig 38



New geometry you create uses the line width you select.

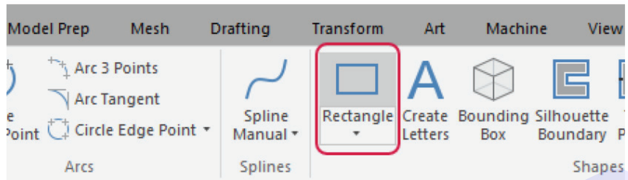
6 Click the Wireframe Color drop-down, and select green as shown below. (Fig 39)

Fig 39



7 On the Wireframe tab, choose Rectangle. (Fig 40)

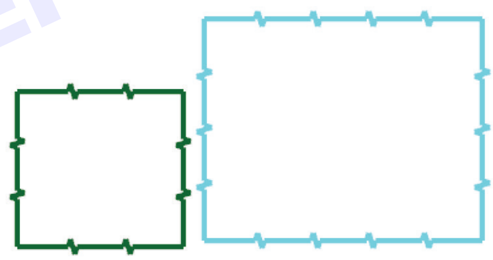
Fig 40



The Rectangle function panel opens.

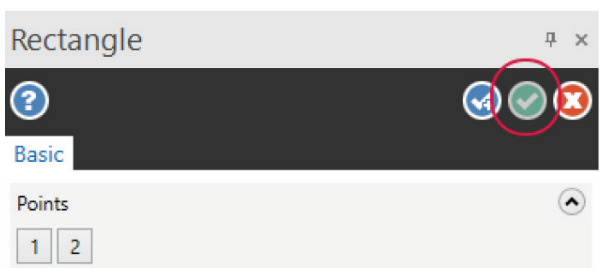
8 Click in the graphics window and draw two rectangles of any size. Follow the prompts in the graphics window. (Fig 41)

Fig 41



9 Click OK.

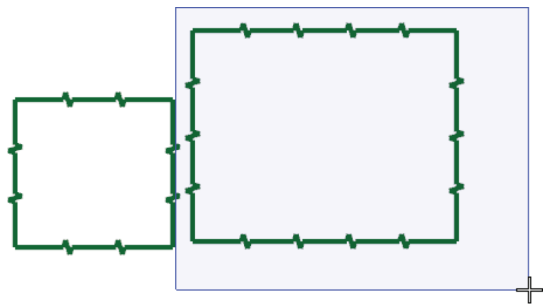
Fig 42



Your rectangle wireframe geometry is created.

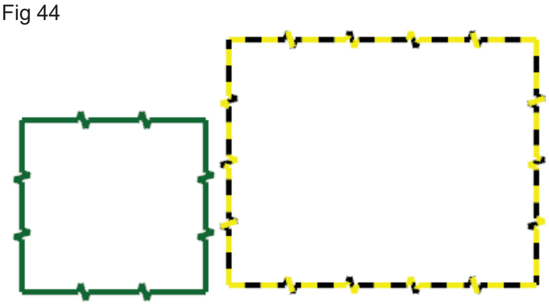
10 Hold down your left mouse button and drag to draw a window around one rectangle in the graphics window. (Fig 43)

Fig 43



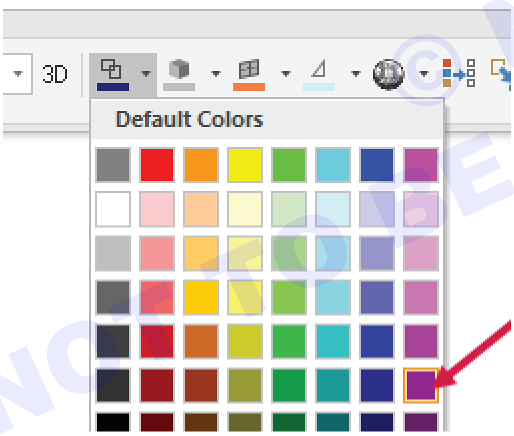
Mastercam selects the rectangle. (Fig 44)

Fig 44



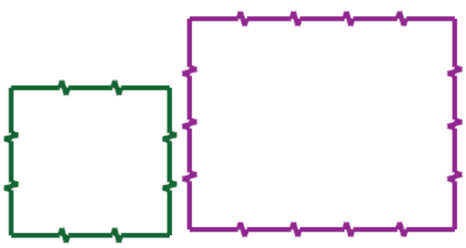
11 Click the Wireframe Color drop-down and select another colour (e.g. purple) (Fig 45)

Fig 45



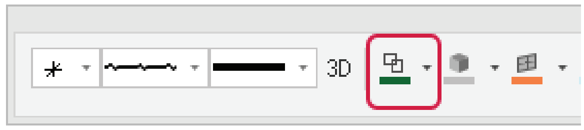
Only the rectangle you selected changes to purple. (Fig 46)

Fig 46



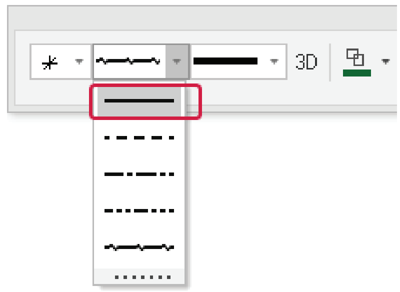
The default wireframe color does not change on the mini toolbar. Any new wireframe geometry you create still uses the default color, which is green. (Fig 47)

Fig 47



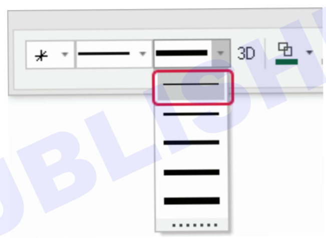
12 With nothing selected in the graphics window, set the Line Style to the option shown below. This resets it to default. (Fig 48)

Fig 48



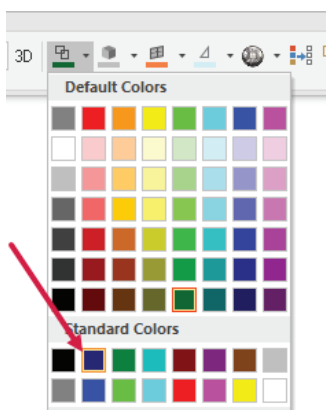
13 Set the Line Width to the default option shown below. (Fig 49)

Fig 49



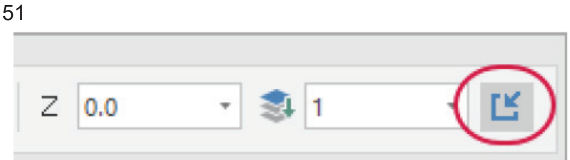
14 Set the Wireframe Color to the default blue shown below. (Fig 50)

Fig 50



15 Select Toggle Attributes Panel to return the mini-toolbar to the right-click menu. (Fig 51)

Fig 51



16 Click File, Save. Name your file and save it.

3D model - Import and simulation in CAM software

Objectives: At the end of this lesson you shall be able to
- **explain about 3D model import in CAM**
- **define the workflow in CAM software**
- **describe about selection of tool and cutting parameters for facing toolpath**
- **explain about simulation in CAM software.**

3D Modelling

A 3D model is a digital representation of the physical object to be manufactured, and it provides the necessary geometric information to generate toolpaths for machining. The precision of the 3D model directly influences the quality of the final product. With advances in CAD, complex geometries can now be modeled and imported into CAM software, allowing for intricate and precise machining operations.

Advantages of Using 3D Models in CAM

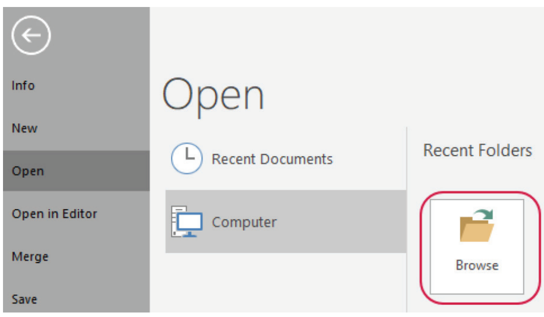
- **Visualization**: Engineers and machinists can visualize the end product before actual manufacturing.
- **Precision:** 3D models ensure that the exact dimensions and tolerances are maintained.
- **Complex Designs:** Enables the machining of complex parts that would be difficult to manufacture using traditional methods.

Importing the 3D Model

Importing files from other applications is similar to opening native Mastercam files. Depending on the file, you might need to specify how Mastercam imports elements of the file. For example, you have to convert a SOLIDWORKS (.sldprt) file to a Mastercam part. You do not need SOLIDWORKS to open or convert a SOLIDWORKS file.

1 **Select File, Open. Select Computer, then Browse. (Fig 1)**

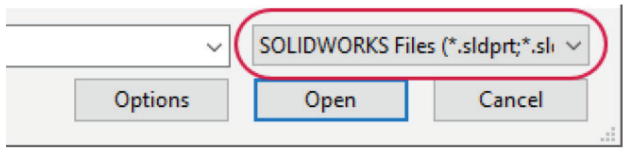
Fig 1



The Open dialog box displays.

2 In the Open dialog box, select SOLIDWORKS Files ("*.sldprt"; "*sldasm"; "*slddrw") from the drop-down. (Fig 2)

Fig 2



The dialog now displays only SOLIDWORKS parts.

3 Select the part you want to open.

4 Select Open in the Open dialog box.

Select Don't Save if you are asked to save changes to the current Mastercam file. Mastercam converts and opens the part.

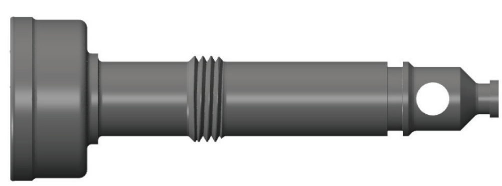
5 Click Outline Shaded and toggle Translucency off to view the part as a solid.(Fig 3)

Fig 3

The part will be displayed in Mastercam.

Example of part imported in Mastercam (Fig 4)

Fig 4



Introduction to Machine Simulation in CAM

Machine simulation in CAM is a virtual environment where the machining process is tested before actual production. It is a powerful tool that allows engineers to simulate the behavior of CNC machines, including the movement of the tool and the workpiece. This simulation helps in verifying the correctness of the toolpaths, ensuring that the machine operations will proceed as expected without causing damage to the machine, tool, or workpiece.

Benefits of Simulation

- **Prevention of Collisions:** Detect and avoid tool and machine collisions before they happen.

- **Validation of Toolpaths:** Ensure that the toolpaths are correctly defined and will produce the desired geometry.
- Material and Cost Savings: Reduce material waste by optimizing the cutting process and minimizing errors.

Workflow in CAM Software

When using CAM software, the process follows a step-by-step workflow that helps transform a digital design into a physical product. Let's break it down into simple steps.

Model Importation: The first step is to bring in the 3D design of the part you want to make. This design is usually created in CAD (Computer-Aided Design) software and saved in formats like STEP or IGES, which are common file types that CAM software can read. The CAM software needs the design file to start working on it.

Setup: Once the design is imported, the next step is to set everything up for machining.

- **Machine selection:** You choose the machine you'll use for the job, like a CNC lathe or mill. This helps the software understand what kind of tools and movements are available.
- **Stock setup:** You tell the software about stock shape and size.
- **Tool library configuration:** You set up the tools you'll use, like different types of inserts, their sizes, and shapes.

Toolpath generation: Now, the software takes over to figure out the best way to cut the material. This is called toolpath generation. The software creates a path for the cutting tool to follow, based on the shape of the 3D model and the machining strategy you choose. It's like planning a route for a road trip – you want the most efficient path to reach your destination.

Simulation: Before cutting anything, you run a simulation. This is a virtual test to see how the cutting tool will move and behave. The software shows a preview of the machining process, helping you check for any mistakes, like the tool hitting the material in the wrong place. It ensures everything is safe and correct.

Post-Processing: After everything looks good in the simulation, the final step is post-processing. The toolpaths are converted into G-code. This G-code contains all the instructions the machine needs to cut the part precisely as planned.

Procedure to run a simulation in Computer Aided Manufacturing software

Select configuration file

1 Start Mastercam using your preferred method:

a Double-click Mastercam's desktop icon. (Fig 5)

Fig 5

OR

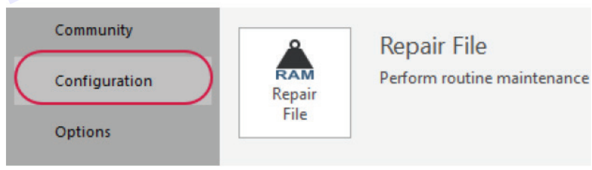
b Launch Mastercam from the Windows Start menu.

2 Select the default metric configuration file:

a Click the File tab.

b Choose Configuration from Mastercam's Backstage View to open the System Configuration dialog box. (Fig 6)

Fig 6



c Choose ...\mcamxm.config <Metric> from the Current drop-down list. (Fig 7)

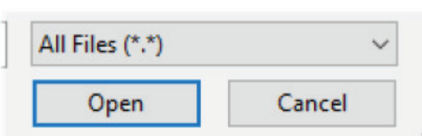
Fig 7

d Click OK.

Loading a machine definition: Your part file needs at least one machine group before you can create toolpaths. Mastercam creates a machine group whenever you select a machine.

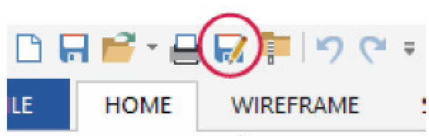
1 Open the part file.(Fig 8)

Fig 8



2 Save the part as partname.mcam. (Fig 9)

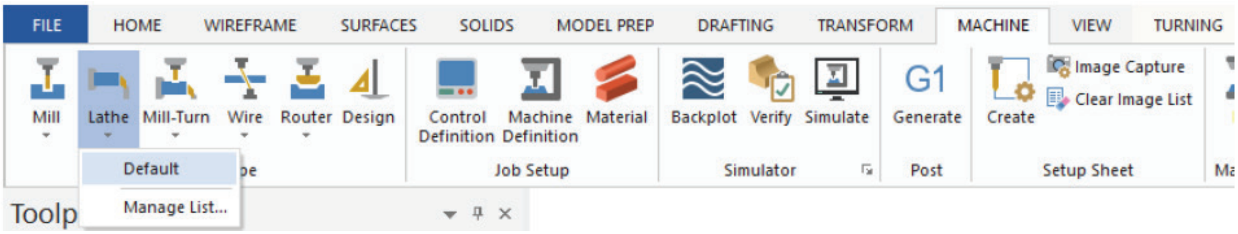
Fig 9



Saving the part under a new name helps prevent you from accidentally modifying the original.

3 From the Machine tab, choose the Lathe machine type, and select Default. (Fig 10)

Fig 10



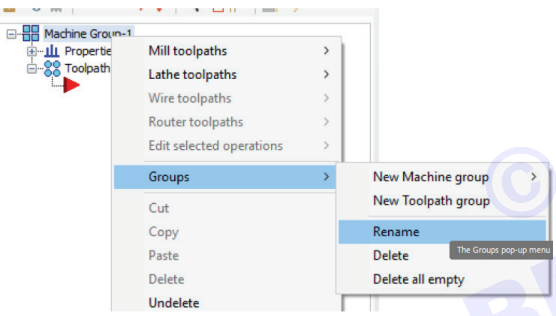
Rename the Machine Group

Machine groups store complete jobs for a specific machine. For example, if some toolpaths will be cut on a lathe, and other toolpaths on a mill, you can simply create a second machine group. Each machine group can store its own job setup information and tools and use a different set of toolpath defaults. The toolpaths from each group will post to separate NC files.

Mastercam lets you create as many machine groups as you need to organize your work.

1 Right-click the machine group, and select Groups, Rename from the pop-up menu. (Fig 11)

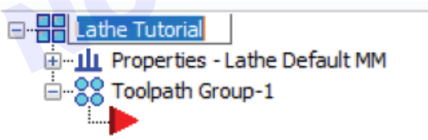
Fig 11



Mastercam highlights the current group name.

2 Type a new machine group name. (Fig 12)

Fig 12



The machine group name can be anything you want, but it is best to choose a name that describes the machine and its operations.

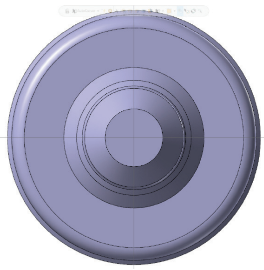
3 Select File, Save to save the file.

Orienting the Part: Before you can machine the part, you need to move the geometry to its machine orientation. In this exercise, you use Mastercam's dynamic gnomon to re-align the part geometry to a typical horizontal lathe.

1 If necessary, press [F9] to display axes. (Fig 13)

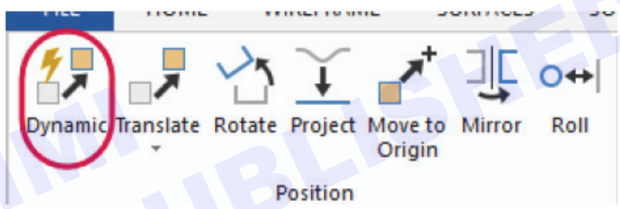
2 Right-click and change the GView to Isometric to view the entire part from another angle. You may wish to unzoom to get a clearer view of the origin.

Fig 13



3 Select Dynamic from the Transform tab. (Fig 14)

Fig 14

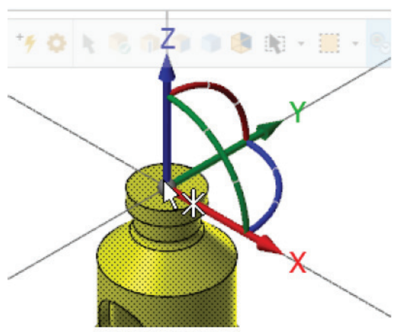


The Dynamic function panel displays.

4 Select the part and press [Enter] or click the End Selection button.

5 Use the Auto Cursor to place the dynamic gnomon on the origin and click. (Fig 15)

Fig 15

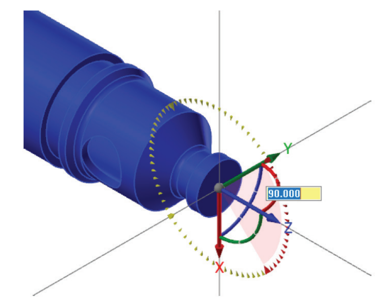


6 Confirm that Move is selected in the function panel. (Fig 16)

Fig 16

7 Select the center segment of the green arc to rotate the part 90 degrees so that its length lies on the X axis. Use your mouse to rotate the gnomon or enter 90 into the on-screen input box. (Fig 17)

Fig 17

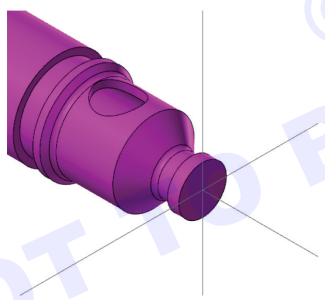


8 On the function panel, click the blue button (OK and Create New Operation). (Fig 18)

Fig 18

9 Select the part again, and rotate it 90 degrees so that it aligns with the Z axis. Use the same method you used previously to rotate the part from its original orientation. (Fig 19)

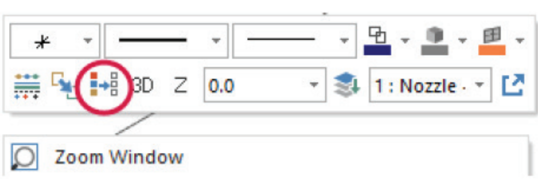
Fig 19



10 Click OK to accept the transformation.

11 Right-click and choose the Clear Colors command from the mini-toolbar to remove the result colors from the translated geometry. (Fig 20)

Fig 20



12 Save the file.

Creating 2D Geometry

Previously we imported solid with no wireframe geometry. Most Lathe operations use a two-dimensional profile of the part in order to chain the geometry. There are two ways to create such a profile from a solid part in Mastercam: the Turn Profile function and Solid chaining.

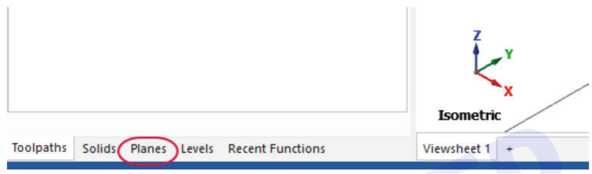
- Turn Profile generates a two-dimensional profile by creating wireframe geometry.
- Solid chaining eliminates the need to create wireframe geometry. It spins a profile around the selected axis "on the fly" when you select geometry for a toolpath.

Create a level for the new profile geometry

Use levels to organize part geometry. By organizing your files into levels, you can more easily control which areas of the drawing are visible at any time and what is selectable.

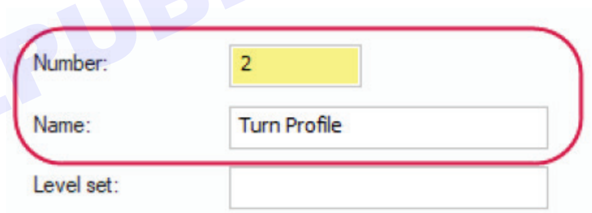
1 Click the Levels tab to open the Levels Manager. (Fig 21)

Fig 21



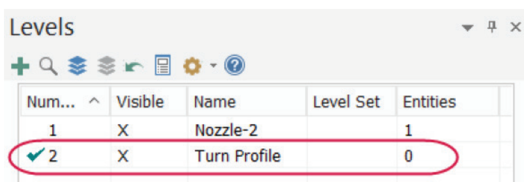
2 Create a new level by typing 2 and Turn Profile in the Number and Name fields. (Fig 22)

Fig 22



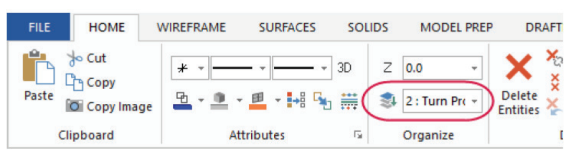
The new level displays in the table with zero entities. A blue check indicates that it is the current level. (Fig 23)

Fig 23



The current level displays on the Home tab. (Fig 24)

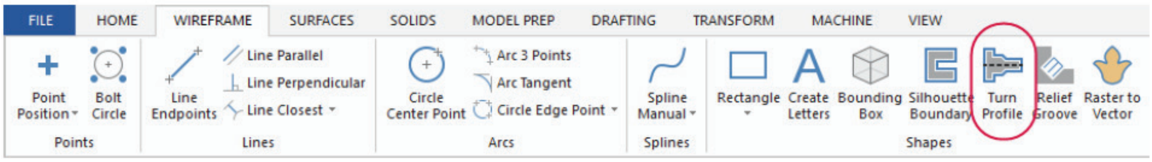
Fig 24



Create the profile

1 From the Wireframe tab, select Turn Profile. (Fig 25)

Fig 25



2 Select the part as prompted. Press [Enter], or click the End Selection button.

3 Use the Spin method.

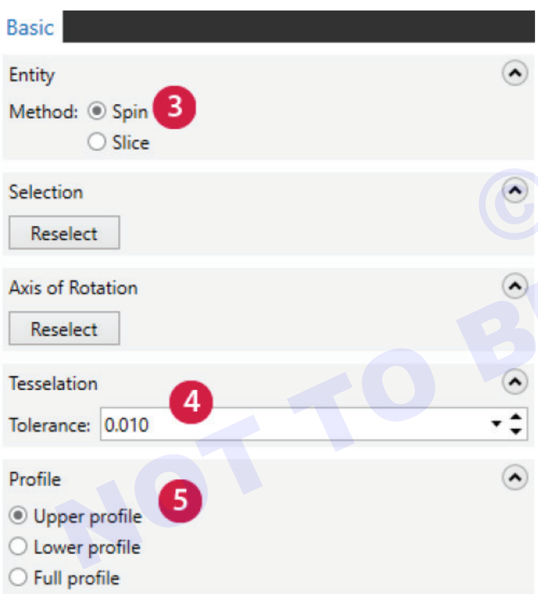
- The Spin method creates a profile by spinning the geometry about an axis.

- The Slice method produces a profile by creating a cross-section through the geometry on the XY plane.

4 Reduce the Tessellation tolerance to 0.01.

5 If necessary, choose the Upper profile option. (Fig 26)

Fig 26

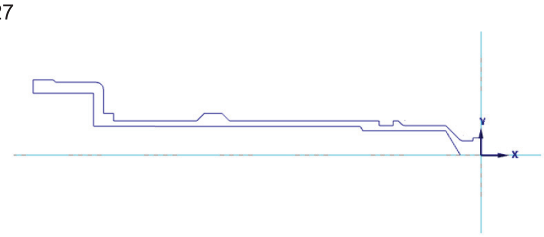


6 Click OK.

7 Return to the Levels Manager, and hide Level 1 (click the X in the Visible column) to view the profile.

The illustration shows the part from the TOP view. (Fig 27)

Fig 27



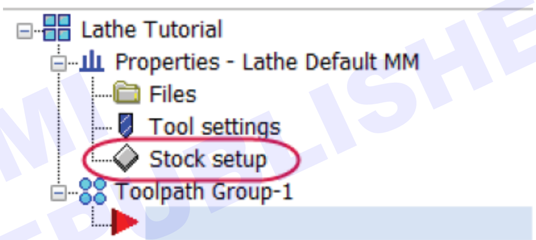
8 Save the file.

Setting Up the Stock in the Left Spindle

Creating the stock before creating the chuck jaws makes it easier to locate the stock relative to your part. Then, when you create the jaws, you can choose to automatically position them relative to the stock.

1 Open the Toolpaths Manager. Expand the Properties group and click Stock setup to open the Stock Setup tab of the Machine Group Properties dialog box. (Fig 28)

Fig 28

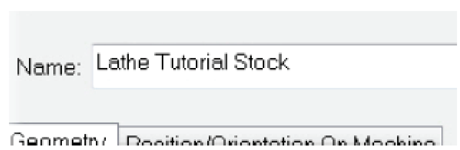


2 In the Stock section, select Left Spindle and click Properties. The Machine Component Manager - Stock dialog box opens. (Fig 29)

Fig 29

3 In the Name field, type Lathe Tutorial Stock to name the stock setup for the left spindle. (Fig 30)

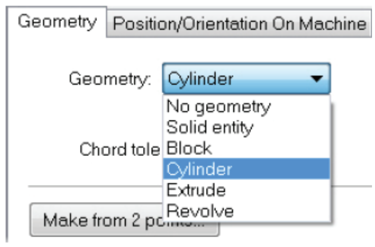
Fig 30



4 Choose Cylinder from the Geometry drop-down selections. (Fig 31)

Cylinder lets you create 3D bar stock. This is suitable for most turning operations.

Fig 31

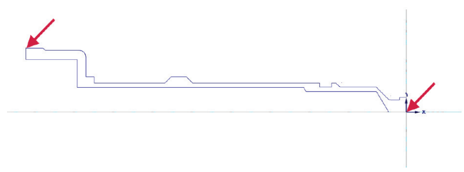


5 Click the Make from 2 points button. (Fig 32)

Fig 32

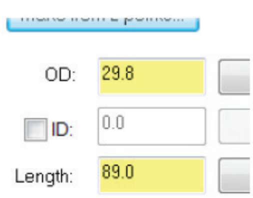
6 In the graphics window, select the origin and the top of the part's back face. If you have not already done so, change the GView to TOP. (Fig 33)

Fig 33



Mastercam calculates the distance between the points and enters the values of the part's outer diameter (OD) and length directly into the fields. (Fig 34)

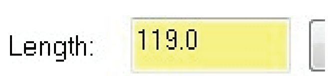
Fig 34



7 Add an additional 30 mm to the Length field.

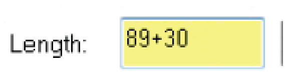
The additional stock (about 30 percent) provides enough material for the left spindle chuck to hold the stock in place during machining.

Fig 35



Fields that take number values have a built-in calculator that let you enter simple formulas directly into the field. (Fig 36)

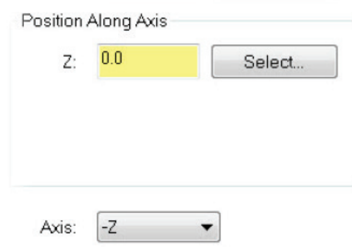
Fig 36



8 Confirm that the Position Along Axis is set to 0.0, and the Axis field has -Z for the stock direction.

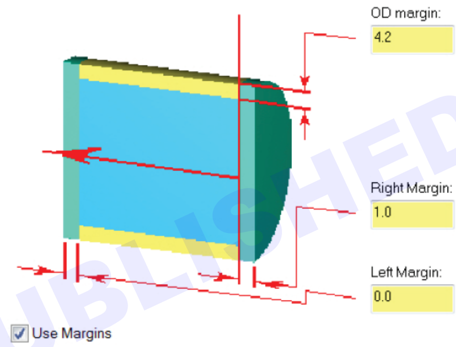
These two values determine the location and orientation of the cylinder. The center of the part's face lies on the origin, and the part is positioned along -Z. (Fig 37)

Fig 37



9 Select the Use Margins option to activate the stock margin fields. (Fig 38)

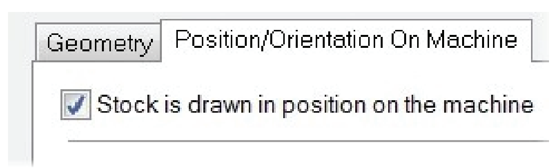
Fig 38



Enter 4.2 mm in the OD margin field and 1.0 mm in the Right Margin field. Mastercam adds the margin to the stock boundary.

10 Click the Position/Orientation On Machine tab, and make sure the option, Stock is drawn in position on the machine is selected. (Fig 39)

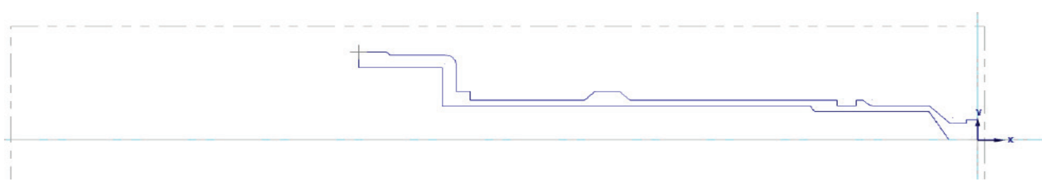
Fig 39



11 Return to the Geometry tab and click the Preview Lathe Boundaries button to view your results. (Fig 40)

12 Press [Enter] to return to the Machine Component Manager - Stock dialog box.

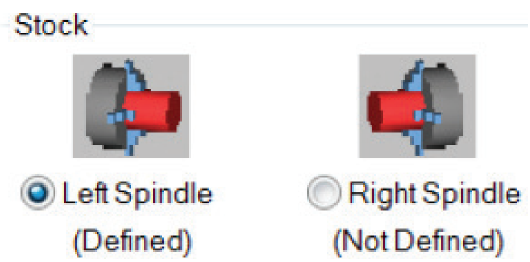
Fig 40



13 Click OK to accept your lathe stock setup settings.

The left spindle stock is defined. (Fig 41)

Fig 41



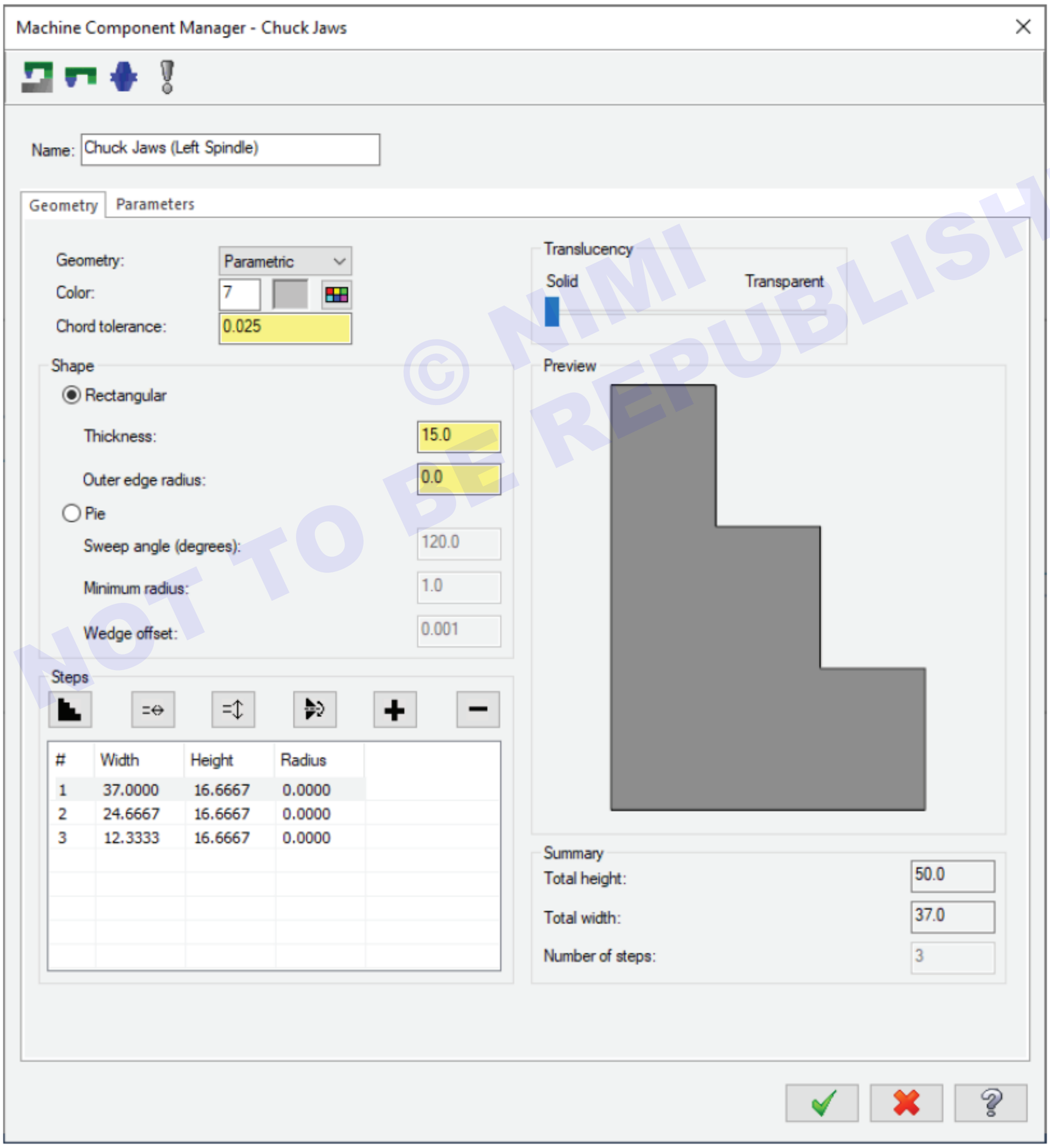
Defining the Chuck Jaws: Now define the position and grip length for the chuck jaws. You can only use the method described below after the stock has been set up.

1 In the Chuck Jaws section, select Left Spindle and click Properties. (Fig 42)

Fig 42

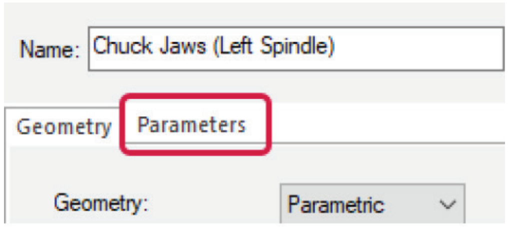
The Machine Component Manager - Chuck Jaws dialog box opens to the Geometry tab. (Fig 43)

Fig 43



2 Click the Parameters tab. (Fig 44)

Fig 44

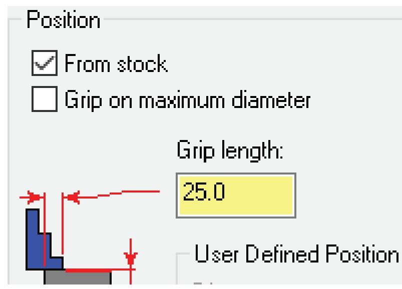


3 In the position section

- Select From stock.
- Enter 25 mm in the Grip length field.

Mastercam uses these settings to calculate the position of the chuck jaws relative to the stock, and to determine how much stock is being held by the chuck jaws. Since you added 30 mm of extra stock while setting up stock, a grip length of 25 mm gives enough clearance to cut off the part. (Fig 45)

Fig 45



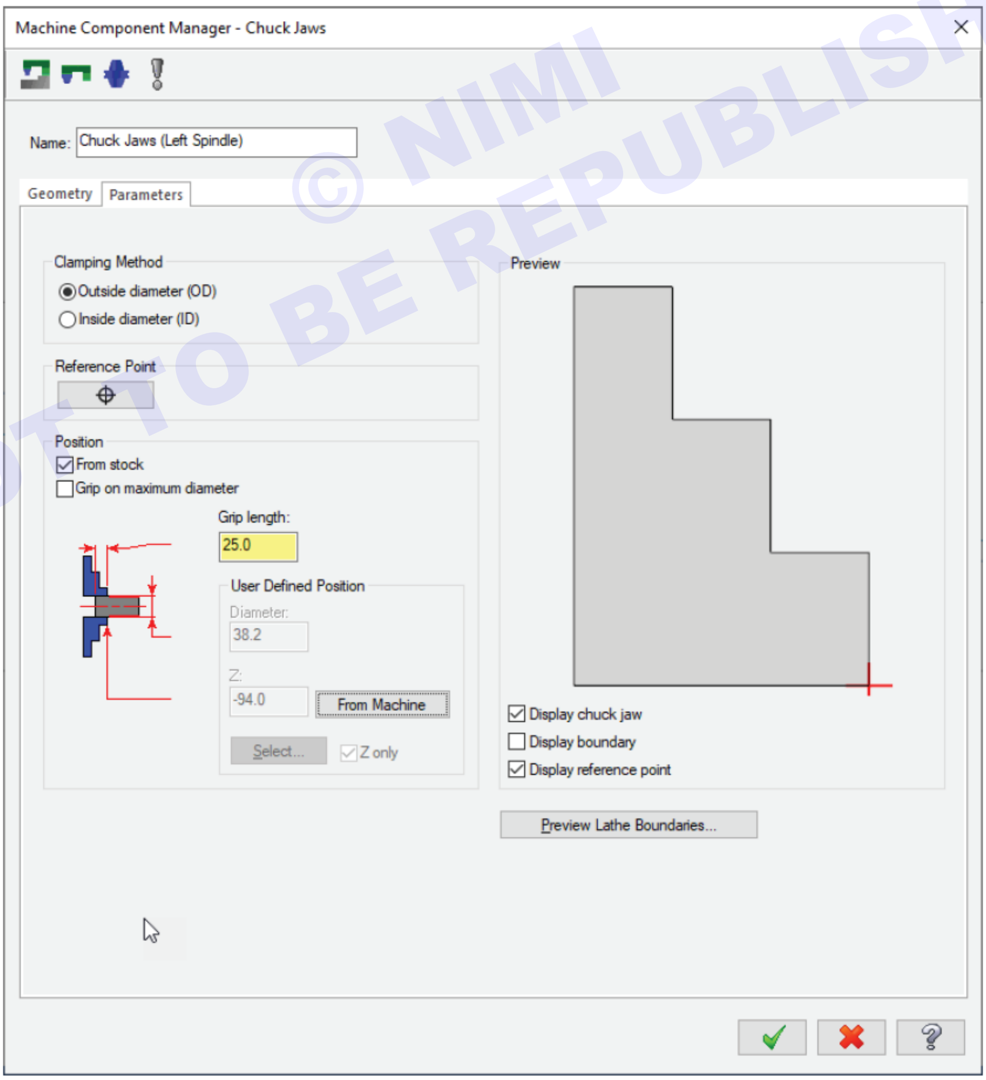
4 Confirm that the other values on the Parameters tab match those in the graphic below. (Fig 46)

5 Click OK to accept these settings.

The Stock Setup tab displays with the left spindle's chuck jaws defined. (Fig 47)

6 Select the Shade boundaries option to see the stock boundaries and chuck jaws more easily you have created. (Fig 48)

Fig 46



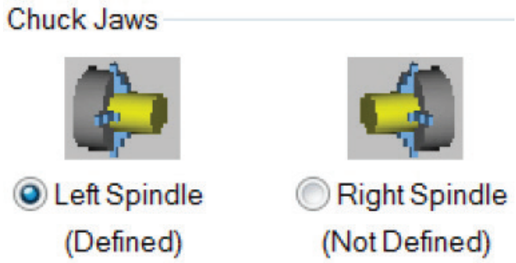


Fig 47

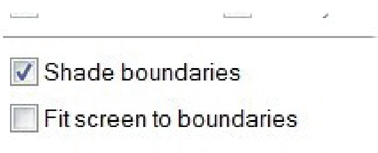


Fig 48

7 Click OK to close the Machine Group Properties dialog box.

8 Save the file (Fig 49)

You have prepared your part. Now you can create toolpaths.

Toolpath creation: Once you have set up your job, we will take the example of facing a part on a CNC lathe. Facing is a common operation used to create a flat surface on the end of a cylindrical workpiece. In this lesson, you will create the toolpaths necessary for the facing operation. After defining the toolpaths, you will simulate the operation to ensure accuracy and verify the process.

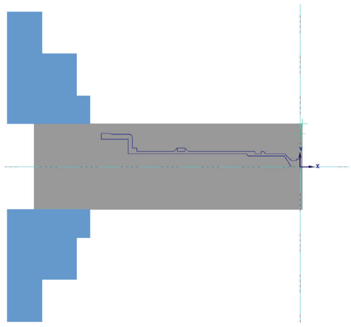


Fig 49

Face toolpaths prepare the face of the part for further machining. Once the face of the part is clean, you can use it to set tools or determine tool offsets.

You do not need to chain geometry to create a face toolpath. Mastercam can create the toolpath entirely from parameters you enter in the Lathe Face dialog box.

1 Choose Face from the Lathe Turning tab. (Fig 50)

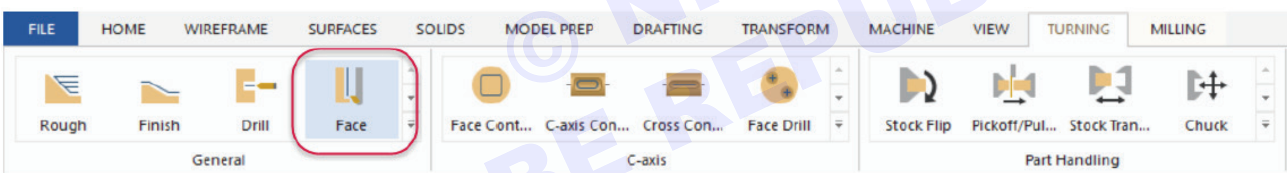


Fig 50

2 From the Toolpath parameters tab, select the default OD roughing tool: T0101 R0.8 OD ROUGH RIGHT - 80 DEG.

Your tools may have different tool numbers. (Fig 51)

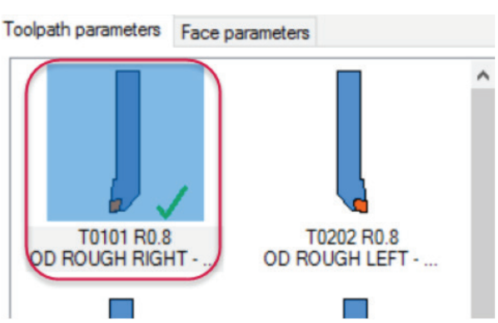


Fig 51

3 Keep all other parameters on this page at their default values.

Enter the cutting values

1 Click the Face parameters tab. (Fig 52)

2 Confirm that Use stock is selected.

Use stock is available only if you have defined the stock boundary in Stock Setup. (Fig 53)

3 If necessary, enter 0 in the text box, or click Finish Z and select the origin from the graphics window to place the finished face at the origin.

4 Keep all other parameters on this page at their default values.

5 Click OK to create the toolpath.

If you use stock for the start and end positions of each pass and the stock changes, the start and end positions of each pass are automatically updated when you regenerate the toolpath. (Fig 54)

Simulating and verifying the Toolpath: Verifying your toolpaths allows you to use solid models to simulate part machining against a selected stock definition. The result Verify creates represents the surface finish, and shows collisions, if any exist. You can identify and correct program errors before they reach the shop floor.

Fig 52

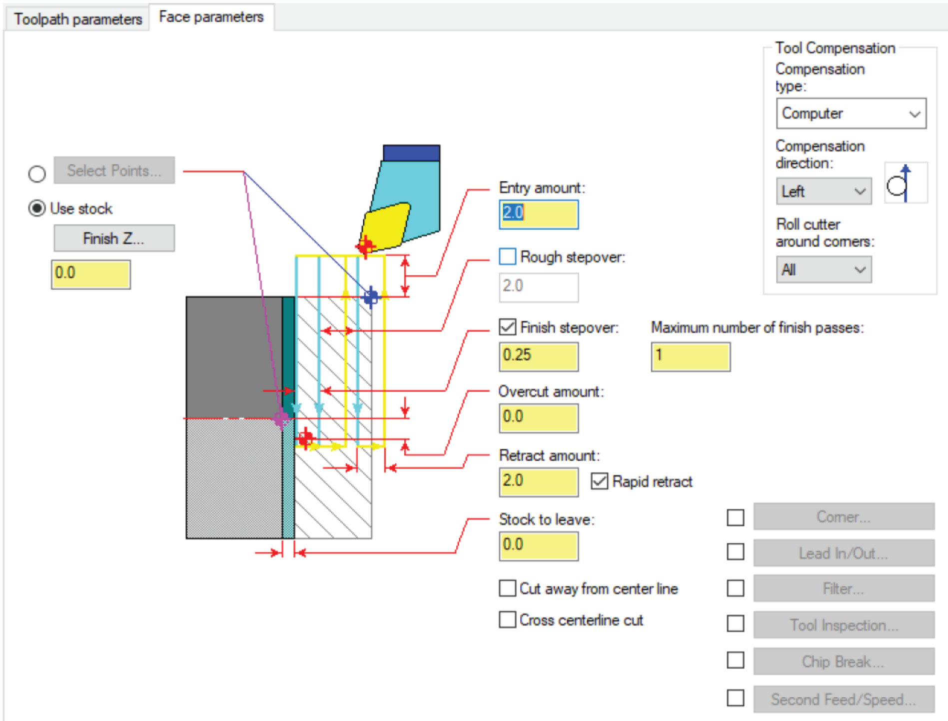


Fig 53

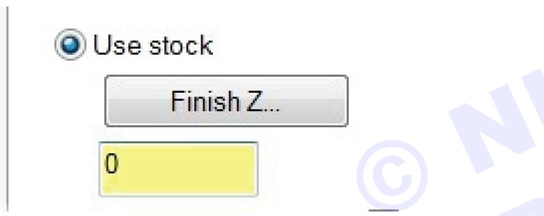
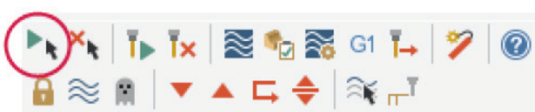


Fig 54

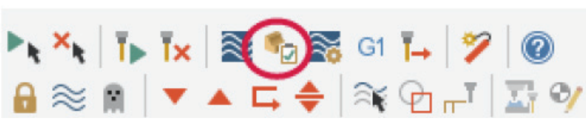
1 Click the Select all operations button in the Toolpaths Manager. (Fig 55)

Fig 55



2 Click the Verify selected operations button. (Fig 56)

Fig 56



Mastercam Simulator opens a view of the part in Verify mode in a separate window. (Fig 57)

3 Select the Initial Stock option in the ribbon bar's Visibility group to see the stock before machining. (Fig 58)

4 Select the Initial Stock option again to view the part profile against a translucent display of the initial stock. (Fig 59)

5 For a better view of the operations in the Mastercam Simulator, use the Page Up key to zoom into the part and [Ctrl + Arrows] to pan the view.

The views in Mastercam and Mastercam Simulator windows can be synchronized. Turn this option on or off from the Viewtab of Mastercam Simulator.

6 Click the Play button at the bottom of the Simulator screen. (Fig 60)

7 Use the Play [R], Step Forward [S], and Step Backward [B] keys to view the operations again at your own pace.

8 Minimize the Mastercam Simulator or move it to another monitor.

- You can dock the Mastercam Simulator on a second monitor and refresh your toolpaths as you make changes.

- When you close the Mastercam Simulator window, the current layout is saved and used at the application's startup.

Fig 57

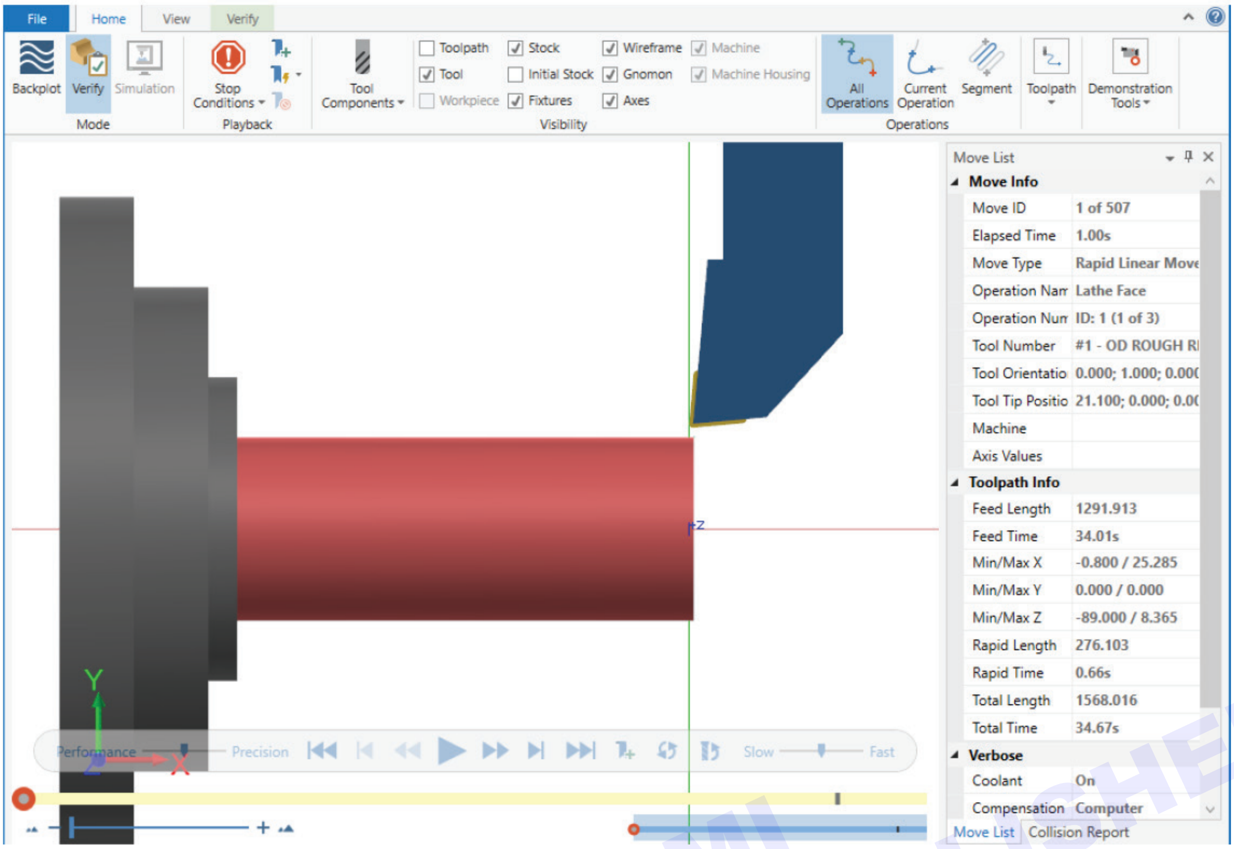


Fig 58

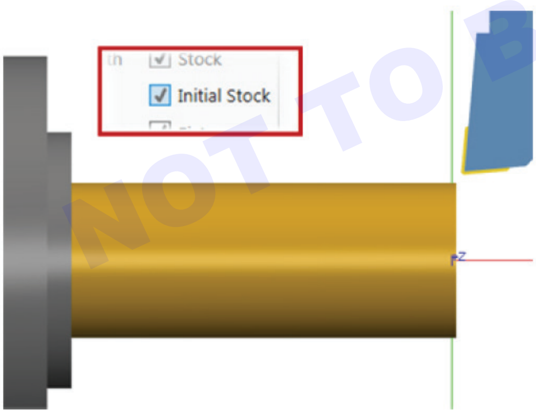


Fig 59

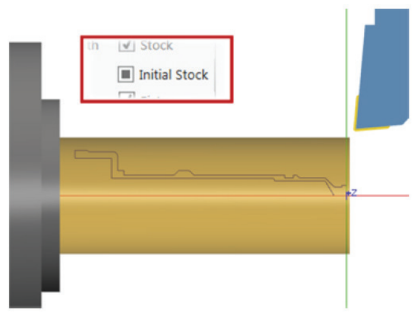


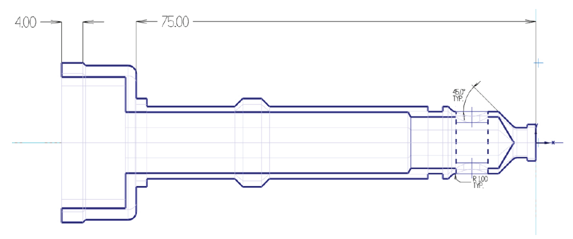
Fig 60

NC program for machining

Objectives: At the end of this lesson you shall be able to
- **define a NC Program by using Computer Aided Manufacturing Software**
- **explain about how to export the generated NC program for machining process.**

Once you have set up your job, you can begin creating toolpaths. Several toolpaths are normally involved in machining a lathe part. In this lesson you create the toolpaths necessary to shape the outer diameter (OD) of the part. The generated toolpaths will then get converted to NC programs. There are important parameters which we need to define in different type of machining techniques. In this exercise we will see the parameters of Facing, Roughing and Finishing. (Fig 1)

Fig 1



Facing the part: Face toolpaths prepare the face of the part for further machining. Once the face of the part is clean, you can use it to set tools or determine tool offsets.

You do not need to chain geometry to create a face toolpath. Mastercam can create the toolpath entirely from parameters you enter in the Lathe Face dialog box.

1 Choose Face from the Lathe Turning tab. (Fig 2)

2 From the Toolpath parameters tab, select the default OD roughing tool: T0101 R0.8 OD ROUGH RIGHT - 80 DEG.

The tool numbers called out in this tutorial are the defaults listed in the default library: Lathe_mm.Tooldb. Your tools may have different tool numbers.

- The tool has an orange insert when the insert faces away from you (insert down).
- The tool has a yellow insert when the insert faces towards you (insert up). (Fig 3)

Fig 2

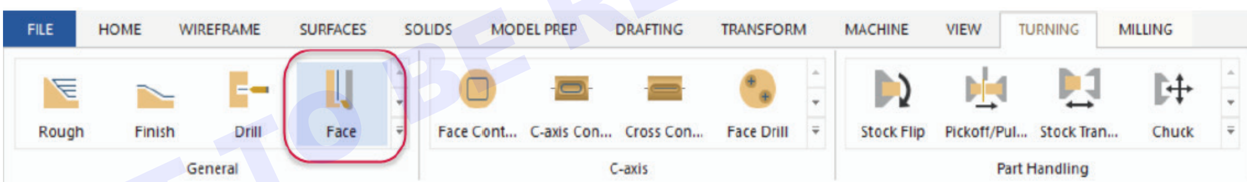
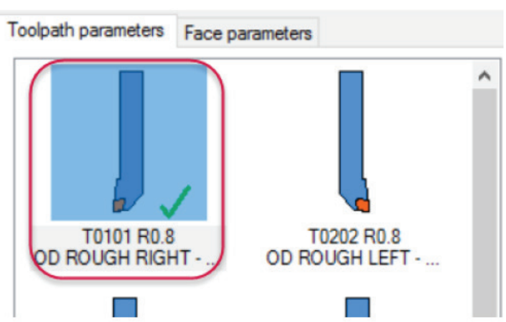


Fig 3



3 Keep all other parameters on this page at their default values.

Enter the cutting values.

Click the Face parameters tab. (Fig 4)

4 Confirm that Use stock is selected.

5 Use stock is available only if you have defined the stock boundary in Stock Setup.

6 If necessary, enter 0 in the text box, or click Finish Z and select the origin from the graphics window to place the finished face at the origin.

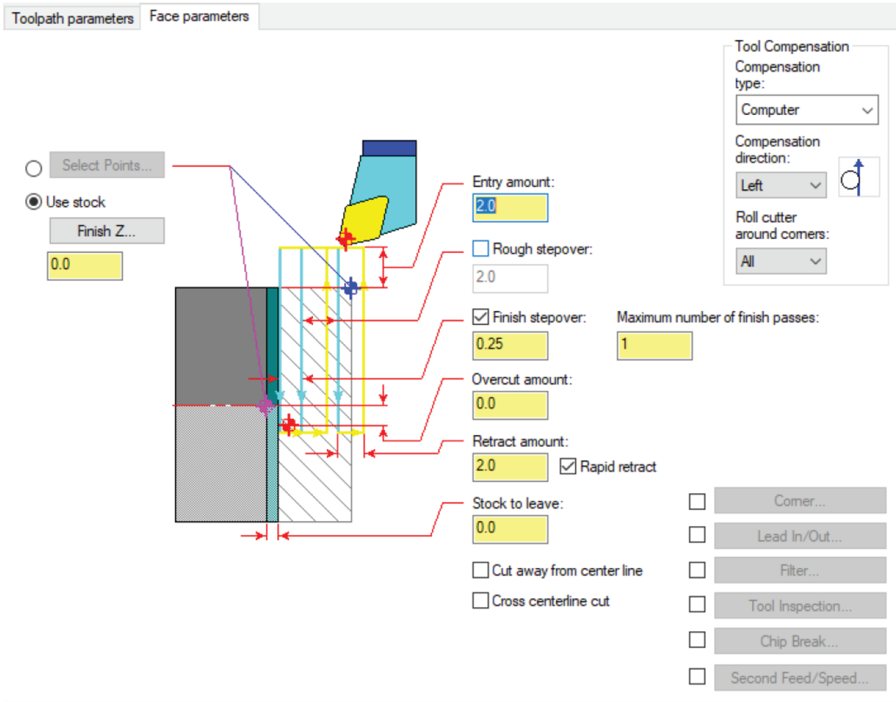
7 Keep all other parameters on this page at their default values.

8 Click OK to create the toolpath.

Roughing the Outer Diameter

Use Dynamic rough toolpaths to quickly remove large amounts of stock in preparation for a finish pass.

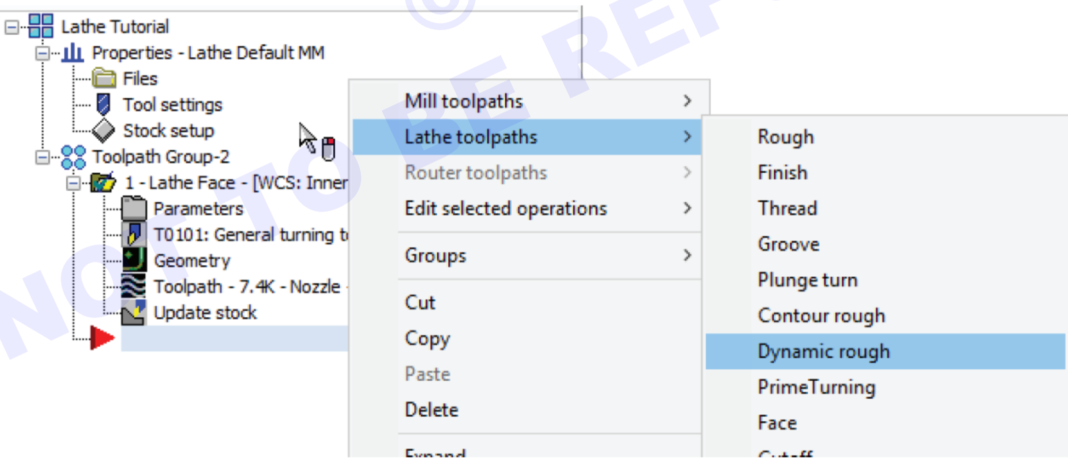
Fig 4



Chain the geometry

1 Right-click in the Toolpaths Manager. Select Lathe toolpaths, Dynamic rough. The Chaining dialog box displays, and you are prompted to select an entry point. Mastercam Lathe defaults to partial chaining. (Fig 5)

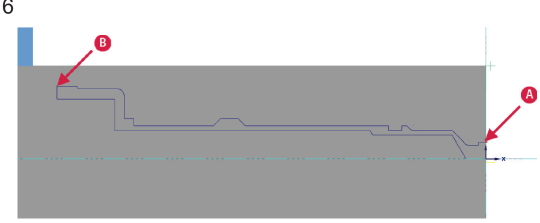
Fig 5



2 Click the geometry at the first entity (a) and then, at the last entity (b) to create a partial chain. Click OK in the Chaining dialog box to accept the chain.

The Lathe Dynamic Rough dialog box opens. (Fig 6)

Fig 6



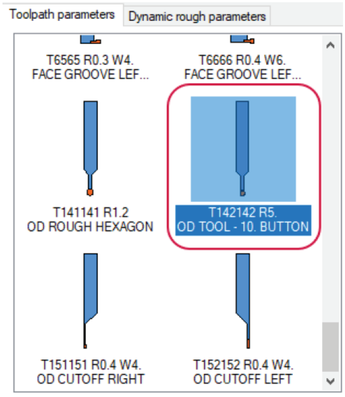
3 Enter the toolpath parameters.

Use the Toolpath parameters tab to select a tool, set feeds and speeds, and modify other general toolpath parameters. This tab is similar for most Lathe toolpaths.

The tool numbers called out in this tutorial are the defaults listed in the default library: Lathe_mm. Tooldb. Your tools may have different tool numbers. (Fig 7)

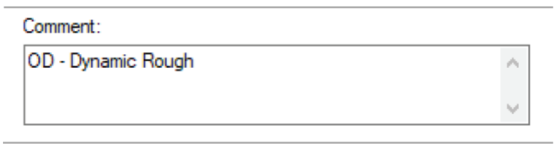
- From the Toolpath parameters tab, select the OD roughing tool: T142142 R5 OD TOOL - 10 BUTTON. Lathe dynamic rough only supports button tools.

Fig 7



- Type OD - Dynamic Rough in the Comment field.
- Comments help identify the operation in the Toolpaths Manager. They can be output to the NC file when you post. (Fig 8)

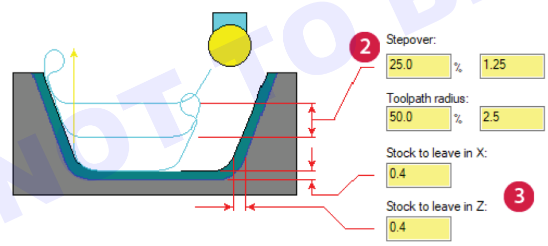
Fig 8



- Comments are optional. In this tutorial, not all operations will have comments.
- Keep all other parameters on this page at their default values.

4 Enter the cutting values. (Fig 9)

Fig 9

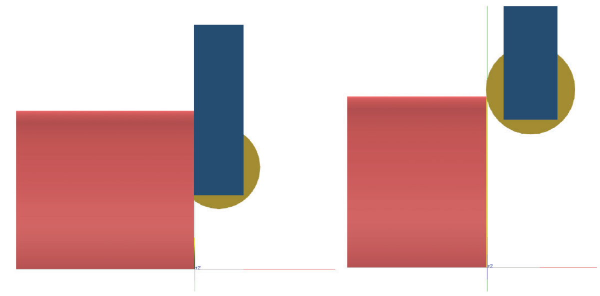


- Click the Dynamic rough parameters tab.
- Decrease the Stepover value to 25%.
- Increase the Stock to leave in X and Z to 0.4.

5 Select the Prevent up cutting option and click the Non-cutting Region button to view these parameters. (Fig 10)

- Prevent up cutting lets you specify the non-cutting portion of a round insert. When prevent up cutting is selected, and Angle is set to 0 degrees, Mastercam cuts only with the lower portion of the tool insert.
- The image on the left shows the part with Prevent up cutting turned off. The tool shank crashes into the material as it tries to cut the toolpath. The image on the right shows the part with Prevent up cutting turned on. The tool cuts only with the specified portion.

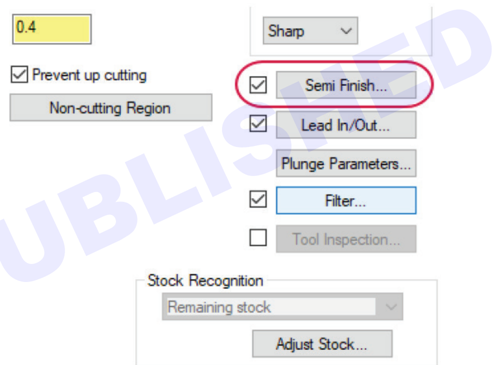
Fig 10



6 Close the Non-cutting Region dialog box after you have viewed it. If the Angle is set to 0, you do not need to modify any parameters.

7 Select the Semi Finish option and click the button to open the Semi Finish Parameters dialog box.(Fig 11)

Fig 11



8 The arc motion of the dynamic toolpath can leave an uneven surface. After the rough passes, the semi-finish pass follows the contour of the part with the roughing tool, smoothing and preparing it for the finish operation.

- Increase the semi-finish Feed rate to 0.8 mm/rev.
- Increase the semi-finish Spindle speed to 600 CSS.

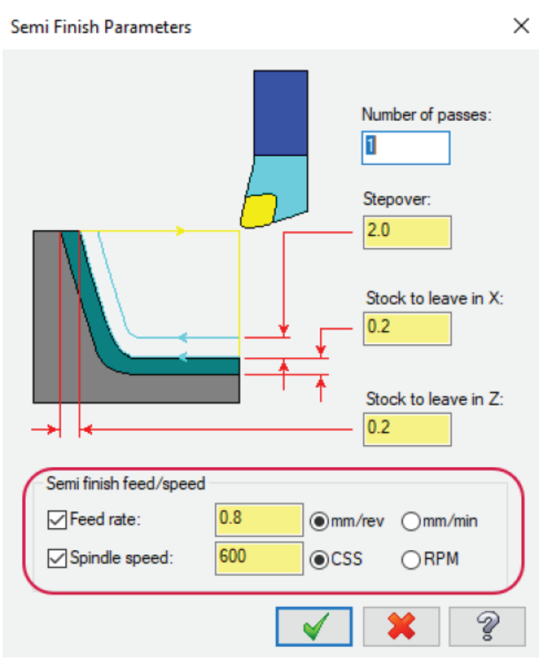
Click OK if the Tool Settings Modified message displays. (Fig 12)

9 Click OK to create the tool path. (Fig 13)

- Mastercam detects that the tool is approaching the stock closer than the specified clearance distance, stops the operation, and displays the Tool Clearance Violation warning. Additionally, the graphics window displays where the tool collides with the stock.

10 Select the option to quit the toolpath and choose to keep the operation when you are prompted. Mastercam marks the attempted Dynamic R Click the Parameters folder to reopen the Lathe Dynamic Rough dialog box and modify the operation. (Fig 14)

Fig 12



11 Click the Lead In/Out button.(Fig 15)

- The Lead In/Out dialog box opens.

Fig 13

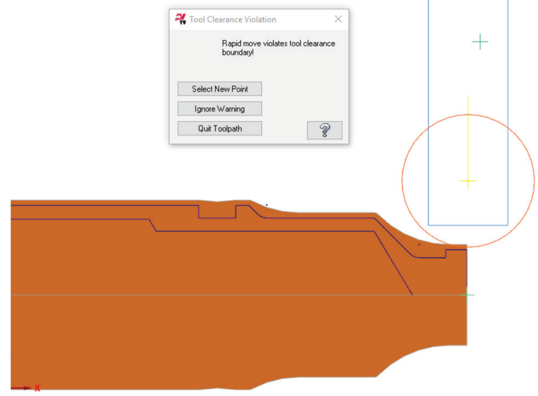


Fig 14

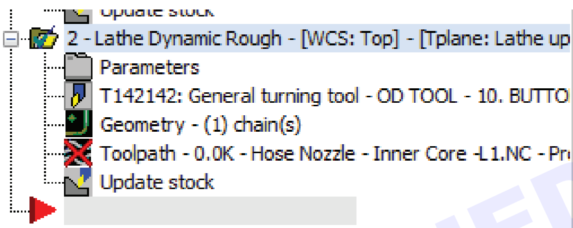
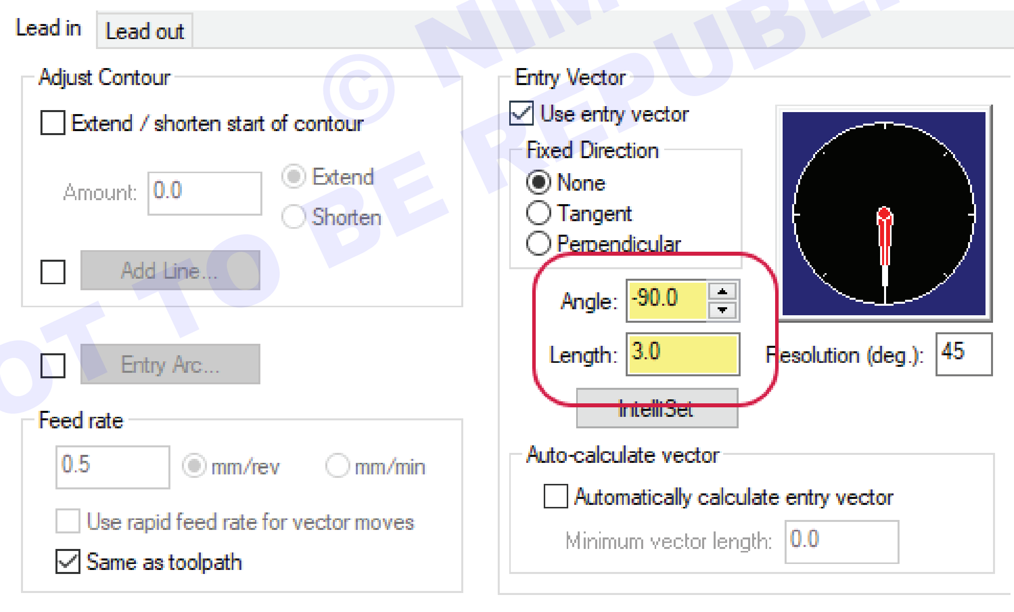


Fig 15



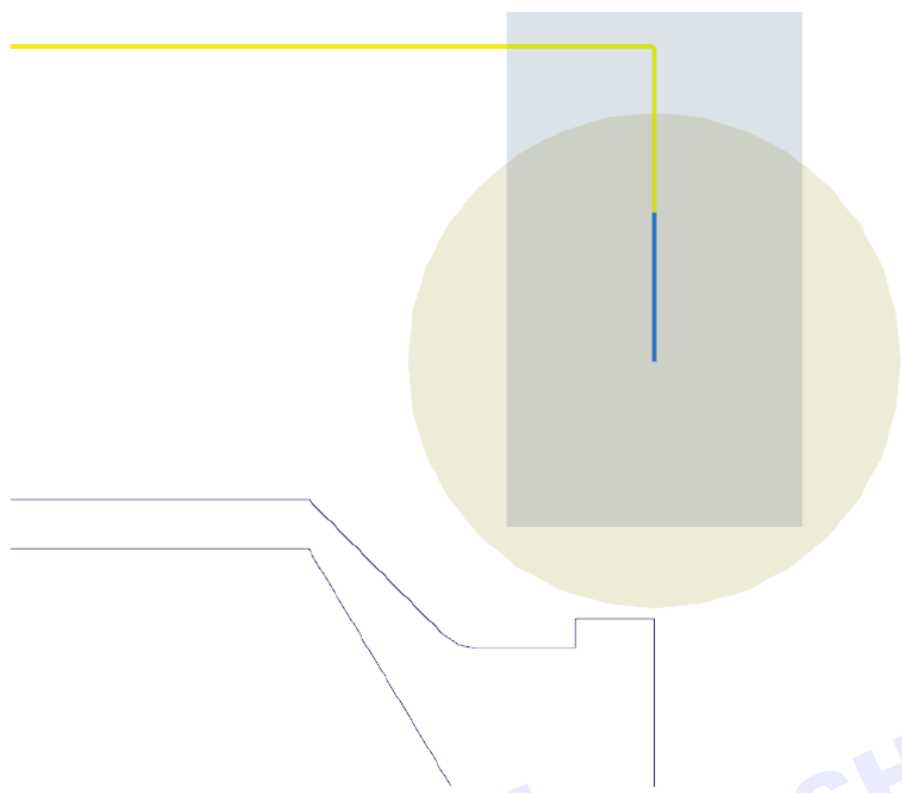
- The parameters in this dialog box control how the tool approaches and/or retracts from the part for each pass in the toolpath. This eliminates the need to create extra geometry for this purpose.
- Click the Lead in tab.
- The Lead in and lead out tabs have identical options for creating entry and exit moves. This allows you to set different values for each move and to combine different types of moves.

12 Enter -90 in the Angle field and increase the Length of the vector to 3.0. (Fig 16)

- This is the angle the tool follows as it approaches the part and the distance it travels as it moves towards the part. Modifying the entry vector will keep the rapid motion of the tool from violating the clearance as it begins the semi-finish.

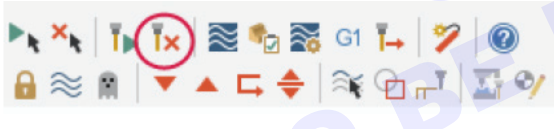
13 Click OK to return to the Lathe Dynamic Rough dialog box.

Fig 16



14 Regenerate the toolpath and save your part. (Fig 17)

Fig 17

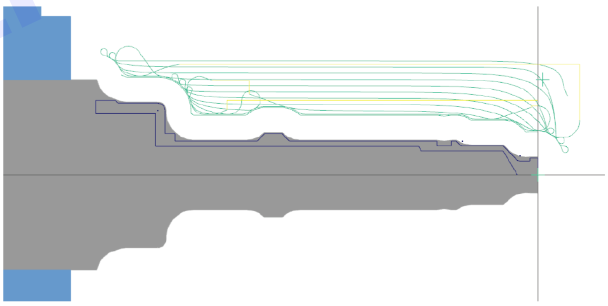


15 The graphics window displays the Dynamic Rough toolpath. (Fig 18)

16 As shown in 1-15 points, anyone can create the tool path for various process like finishing, facing etc. by controlling their respective parameters

Finishing: Use finish toolpaths to have the tool follow the contour of chained geometry. Typically, a finish toolpath follows a roughing toolpath.

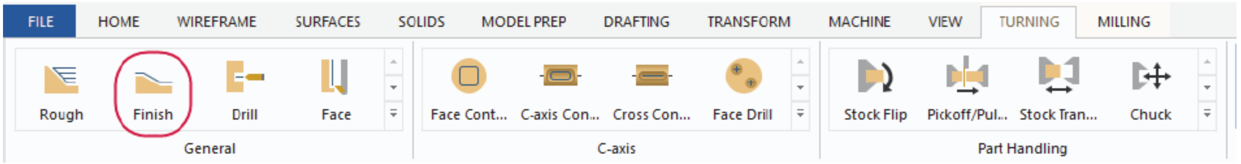
Fig 18



1 Choose Finish from the Lathe Turning tab. (Fig 19)

2 The Chaining dialog box displays, and you are prompted to select a point or chain a contour.

Fig 19

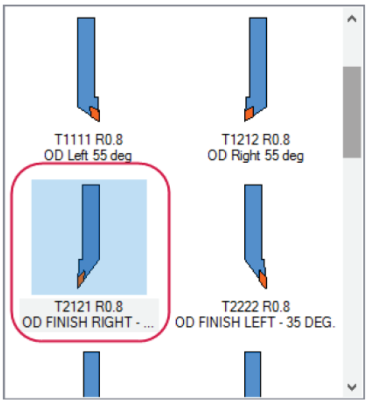


- Select the same geometry that you selected for the Dynamic Rough toolpath.
- Click OK in the Chaining dialog box to accept the chain.
- The Lathe Finish dialog box opens.

3 Enter the toolpath parameters.

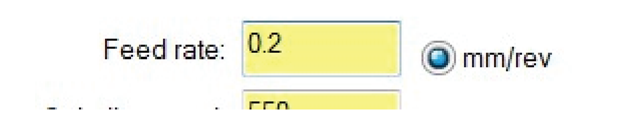
- Entering the toolpath parameters for any finish toolpath uses the same workflow as other lathe toolpaths: first you select the tool and tool options, and then you enter the cutting values.
- Select the finishing tool: T2121 R0.8 OD FINISH RIGHT - 35 DEG. (Fig 20)

Fig 20



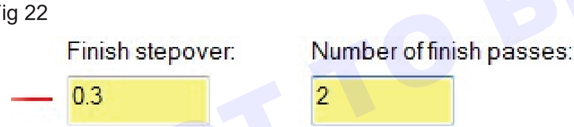
- Decrease the feed rate to 0.2. (Fig 21)

Fig 21



- Keep all other parameters on this page at their default values.
- Click the Finish parameters tab.
- Change the Finish step over to 0.3 and the Number of finishes passes to 2. (Fig 22)

Fig 22



- The operation makes two finer finish passes and avoids excessive cut depths.
- Leave all other parameters on this page at their default values.
- Click OK.
- Mastercam creates a finish operation on top of the previous roughing operation.
- Save the part.

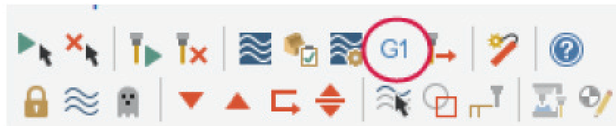
Export the generated NC program for machining process

1 In this exercise, you post the operations in the machine group.

If necessary, select all operations in the Toolpaths Manager.

2 Click the Post selected operations button in the Toolpaths Manager. (Fig 23)

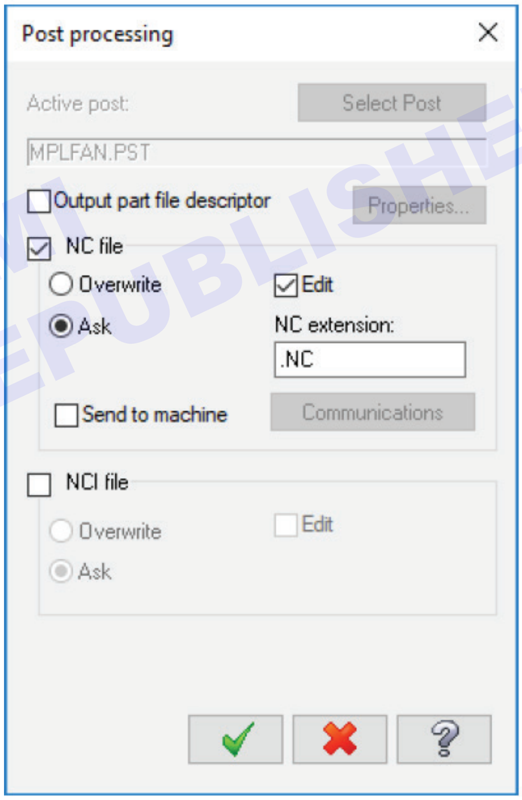
Fig 23



If you have not selected all the operations in a machine group, Mastercam will ask if you want to post all the operations.

The Post processing dialog box displays. Mastercam uses these settings to handle the files that are generated when posting. (Fig 24)

Fig 24



3 Click OK.

The Save As dialog box displays.

Rename the file or click Save to accept the default NC file name.

Mastercam posts the file, and it is opened in your default file editor. Examine the NC file before sending it to the machine.

Manufacturing processes

Objectives: At the end of this lesson you shall be able to
- **state what is manufacturing processes**
- **list the factors influencing manufacturing processes**
- **describe the concept of machining and its processes**
- **explain the benefit of tolerance in assembly drawing.**

Introduction to manufacturing processes: Manufacturing processes refer to the methods and techniques used to transform raw materials into finished goods. These processes are essential in producing products for a variety of industries, including automotive, aerospace, electronics, and consumer goods. The choice of manufacturing process depends on factors such as material properties, product design, required precision, and production scale.

Factors influencing manufacturing processes

1 **Material selection**
- Properties like strength, ductility and thermal resistance determine the appropriate process.

2 **Product design**
- Complex designs may require advanced processes like CNC machining or 3D printing.

3 **Production volume**
- Low-volume production favours flexible processes like 3D printing.
- High-volume production favours processes like casting and stamping.

4 **Cost and efficiency**
- Balancing material costs, tooling expenses, and labour requirements is critical.

5 **Precision and quality**
- Processes like CNC machining are chosen for high-precision parts.
- Casting or forming may be used for less critical tolerances.

Emerging trends in manufacturing

1 **Automation and robotics**
- Enhancing efficiency and consistency.

2 **Industry 4.0**
- Integrating IoT and AI for smart manufacturing.

3 **Sustainable manufacturing**
- Using eco-friendly materials and processes.

4 **Mass customization**
- Tailoring products to individual customer needs.

Manufacturing processes are the backbone of industrial production, evolving continually with technological advancements to meet the demands of modern industries. Let me know if you need a detailed discussion on any specific process.

Concept of machining components and its processes: Machining is a manufacturing process in which a component is shaped, sized, or finished by removing material from a workpiece using tools and machines. It is widely used to produce components with high precision and accuracy, often from metals, plastics, ceramics, or composites.

A **machined component** refers to the final product made through machining processes. These components are commonly found in industries like automotive, aerospace, electronics, and medical devices.

Design concept in assembly of parts: The design concept for the assembly of parts involves the process of creating and organizing components to ensure they fit together to form a functional product. This phase is critical in engineering and manufacturing to achieve efficiency, reliability, and ease of production.

Principles in assembly design

1 **Modularity**
- Design parts as independent modules that can be assembled easily.
- Simplifies maintenance and repair by allowing replacement of individual modules.

2 **Standardization**
- Use standardized parts (e.g., screws, fasteners) to reduce complexity and costs.
- Ensures compatibility with other systems and components.

3 **Minimizing parts**
- Reduce the total number of parts to simplify the assembly process.
- Achieve this by combining functions into single components when feasible.

4 **Design for assembly**
- Focus on ease of assembly during the design phase.
- Optimize part orientation, accessibility, and handling.

5 **Tolerance and fit**
- Ensure proper tolerances and fits (e.g., clearance, interference, and transition fits) for smooth assembly and operation.

- Use geometric dimensioning and tolerancing (GD&T) for precise specifications.

6 **Fastening methods**

- Choose appropriate joining techniques such as welding, bolting, riveting, or adhesive bonding based on application requirements.

7 **Accessibility**

- Design parts to allow tools and operators easy access during assembly or maintenance.

8 **Alignment features**

- Incorporate features like dowel pins, grooves, or locator tabs for precise alignment during assembly.

9 **Material compatibility**

- Select materials that are compatible to avoid corrosion, thermal expansion issues, or strength mismatches.

10 **Ergonomics**

- Ensure assembly processes are ergonomically designed for operators to reduce fatigue and errors.

Design process for assembly

1 **Conceptual design**

- Define the product's purpose and key functionality.
- Identify the major components and their relationships.

2 **Component design**

- Create detailed designs for individual parts, ensuring compatibility and functionality.

3 **Assembly layout**

- Develop an exploded view or layout showing how parts fit together.

4 **Prototyping**

- Create prototypes to test the assembly process and identify potential issues.

5 **Optimization**

- Refine the design to improve manufacturability, reduce assembly time, and ensure robustness.

6 **Validation**

- Conduct tests to ensure the assembled product meets functional, aesthetic, and durability requirements.

Tolerances in assembly drawings: Tolerances are detailed in engineering drawings to ensure proper communication between design and manufacturing teams.

1 **Dimensioning with tolerances**

- Clearly specify nominal dimensions and permissible variations.
- Example: 100±0.2 mm100 \pm 0.2 \, \text{mm}100±0.2mm

2 **GD & T symbols**

- Standardized symbols (per ISO or ASME) define geometric tolerances.
- Example
  - ⊥\perp⊥: Perpendicularity
  - ∥\parallel∥: Parallelism

3 **Material condition modifiers**

- Symbols like MMC (Maximum Material Condition) or LMC (Least Material Condition) define the tolerance zone based on material availability.

4 **Assembly notes**

- Include specific instructions, such as allowable fits or assembly procedures.

Design process for assembly with tolerances

1 **Define functional requirements**

- Understand the task the assembly must perform.
- Example: A piston must slide freely but not leak.

2 **Determine critical features**

- Identify dimensions and geometries critical to assembly and functionality.

3 **Specify tolerances**

- Assign tolerances based on functionality, manufacturing capabilities, and cost constraints.

4 **Simulate assembly**

- Use software tools to simulate how parts fit together under tolerances.

5 **Validate and iterate**

- Test prototypes to ensure the tolerances meet functional and assembly requirements.

Common tools for tolerance analysis

1 **Tolerance stack-up analysis**

- Ensures cumulative tolerances across multiple parts do not cause assembly issues.

2 **Simulation software**

- Tools like CATIA, SolidWorks, or AutoCAD for virtual assembly testing.

3 **GD&T checklists**

- Used to verify compliance with design requirements.

Importance of tolerances in assembly design

- **Ensures functionality:** Prevents fitment issues and ensures performance.
- **Improves manufacturability:** Balances design precision with production capabilities.
- **Reduces costs:** Avoids over engineering and unnecessary precision.

A well-thought-out design with appropriate tolerances ensures the assembled product functions as intended while remaining cost-effective and manufactural. Let me know if you'd like to discuss a specific example or aspect in more detail!

Introduction to CNC Turning, VMC and NC

Objectives: At the end of this lesson you shall be able to
- **state what is CNC turning, VMC and NC**
- **describe the features of CNC turning, VMC and NC**
- **compare CNC lathe, VMC and NC machines and its applications in industries.**

CNC Turning, VMC (Vertical Machining Center), and NC (Numerical Control) represent different levels of automation and machining technologies used in modern manufacturing. Each has its unique purpose and capabilities for precise and efficient machining operations.

1 CNC turning

CNC Turning is a machining process performed on a lathe, where the workpiece rotates while a stationary cutting tool removes material to achieve the desired shape. It is widely used for creating cylindrical or rotational parts.

Key features of CNC turning

- **Process:** The workpiece spins, and the cutting tool shapes the material by removing excess.
- **Machine used:** CNC Lathes or Turning Centers.
- **Axes:** Typically, 2 axes (X and Z); advanced machines may include Y-axis or live tooling.
- **Applications:** Shafts, bushings, pulleys, threaded components.

Advantages

- High precision and repeatability for cylindrical parts.
- Faster and more efficient than manual lathes.
- Suitable for mass production of rotational components.

2 Vertical Machining Center (VMC)

VMC refers to a CNC milling machine where the spindle is oriented vertically. It is used for operations such as drilling, tapping, milling, and contouring.

Key features of VMC

- **Process:** The cutting tool rotates vertically to remove material from a stationary or moving workpiece.
- **Machine used:** Vertical Machining Centers.
- **Axes:** Standard machines use 3 axes (X, Y, Z); advanced machines may use 4 or 5 axes.
- **Applications:** Creating intricate parts, molds, flat or contoured surfaces, and cavities.

Advantages

- Versatile for machining complex shapes.
- High precision and surface finish.
- Suitable for a wide range of materials, including metals, plastics, and composites.

3 NC (Numerical Control)

NC machines were the precursor to CNC machines. They operate based on pre-defined programs inputted through punched cards, tapes, or other non-digital means. Unlike CNC, NC lacks real-time computer control and automation.

Key features of NC

- **Process:** Numerical data controls machine operations.
- **Machine Used:** Early generations of milling, turning, and drilling machines.
- **Programming:** Requires manual programming using numeric codes.
- **Applications:** Simple and repetitive machining tasks.

Advantages

- Reliable for basic operations.
- Cost-effective for simple machining tasks.
- Easier to maintain than CNC machines.

Limitations

- No real-time control or automation.
- Limited flexibility and capability compared to CNC machines.
- Time-consuming setup and programming process.

Comparison of CNC Turning, VMC and NC

Aspect	CNC Turning	VMC	NC
Primary Motion	Workpiece rotates; tool moves linearly	Tool rotates; workpiece moves/held	Depends on machine type
Axes	2-3 axes (X, Z, optional Y)	3-5 axes (X, Y, Z, optional rotary)	Typically fewer axes

Aspect	CNC Turning	VMC	NC
Control	Computer-controlled in real-time	Computer-controlled in real-time	Pre-programmed numerical codes
Applications	Cylindrical and rotational parts	Intricate parts, molds, and cavities	Simple and repetitive tasks
Flexibility	Highly flexible for complex tasks	Highly flexible for complex tasks	Limited flexibility

Applications in industry

1 **CNC turning**

- Automotive shafts, bushings, and fasteners.
- Aerospace components with rotational symmetry.
- Medical implants and devices.

2 **VMC**

- Mold and die manufacturing.
- Precision components for aerospace and electronics.
- Intricate prototype and production machining.

3 **NC**

- Basic machining operations in small-scale industries.
- Production of simple parts in bulk quantities.

Conclusion

- **NC machines** introduced automation in machining, paving the way for CNC **(Computer Numerical Control)** systems.
- **CNC Turning** specializes in producing cylindrical parts, while, **VMC** offers versatilitiy for complex and intricate shapes.
- Together, these technologies revolutionized manufacturing by increasing precision, reducing manual intervention and enhancing productivity.

CNC machine controllers

Objectives: At the end of this lesson you shall be able to
- **state what is CNC controllers**
- **list the name of the commonly used CNC controllers**
- **describe CNC latest controllers and its capabilities.**

CNC machine controllers are the central processing units that interpret programing instructions(G-Code) and control the movement and operation of the machine. Below are some of the most widely used CNC controllers and their latest versions or advancements.

Fanuc CNC controllers

Latest version: Fanuc series 30i/31i/32i-B plus

Features

- Highs speed, high accuracy machining.
- Advanced 5 axis machining capabilities.
- Multi-channel and multi-path support for complex machining tasks.
- AI-based optimization for tool path and feed rate.
- Enhanced graphical interface for programming and monitoring,
- IoT ready with connectivity options for real time monitoring and data analysis.

Siemens CNC controllers

Latest version: Sinumerik one

Features

- Digital twin integration for real time simulation and planning.
- Advanced 5 axis and multi-tasking capabilities.
- High performance hardware for faster processing and precision.
- Intuitive HMI (Human Machine Interface) with touch screen support.
- Integrated CAD/CAM tools for streamlined work flows.
- IoT connectivity through Siemens Mind Sphere platform.

Mitsubishi CNC controllers

Latest version M800/M80 series

Features

- High speed processing with reduced cycle times.
- Multi axis and multi-tasking support for turn-mill centers.
- Advanced monitoring tools for predictive maintenance.
- Customizable user interface for ease of use.
- Smart machine optimization using AI and machine learning.
- Energy efficient operations.

Heidenhain CNC controllers

Latest version: TNC 640

Features

- Advanced contouring control for complex geometries.
- Conversational programming for easier operator training,
- 5 axis machining with dynamic collision monitoring.
- High-resolution touch screens for an intuitive interface.
- IoT ready with remote diagnostics and data logging.
- Integrated measurement system for real-time tool and workpeice monitoring.

HAAS CNC controllers

Latest Version: Next Generation Control (NGC)

Features

- Intuitive visual programming system (VPS)
- Enhanced tool management and probing systems.
- Real time performance monitoring via My Haas portal.
- High speed machining with improved accuracy.
- IoT and connectivity for remove machine monitoring.
- Built in diagnostics for maintenance and trouble shooting.

Mazatrol (Mazak CNC Controllers)

Latest version: Mazatrol SmoothAi

Features

- AI-powered programming for adaptive machining.
- Integrated CAD/CAM for simplified workflows.
- Multi-tasking and 5-axis machining capabilities.
- Advanced graphical simulation for toolpath verification.
- IoT-ready with connectivity to Mazak's Smooth Monitor software.
- Real-time tool wear monitoring.

NUM CNC Controllers

Latest version: Flexium+

Features

- Modular and scalable architecture for customized solutions.
- High-performance control for multi-axis machining.
- Integrated safety systems and diagnostics.
- Advanced 3D simulation and visualization tools.
- IoT integration for data-driven manufacturing.

Okuma CNC Controllers

Latest version: OSP-P300SA/P300LA

Features

- Seamless integration with Okuma hardware for optimized performance.
- Advanced collision avoidance systems.
- Adaptive machining with AI-based optimization.
- Customizable interface and built-in CAM support.
- IoT ready with Okuma's Connect Plan for real-time data insights.

Bosch Rexroth CNC Controllers

Latest version. IndraMotion MTX advanced

Features

- High-speed control for multi-axis CNC machines.
- Real-time IoT data logging and analysis.
- Modular design for flexibility in applications.
- Enhanced HMI for intuitive programming and operation.
- Integrated safety and energy efficiency features.

Delta CNC controllers

Latest version: ASDA-MS series

Features

- Cost-effective solution for 3- and 4-axis CNC applications.
- High-speed, high-precision control.
- Intuitive software for programming and monitoring.
- Compact design for small-scale applications.
- IoT-enabled for remote operation and monitoring.

Comparison of features of CNC controllers

Controller	High-speed machining	IoT ready	5-Axis capability	AI optimization	Integrated CAD/ CAM
FANUC 30i/31i/32i-B+	✓	✓	✓	✓	×
Siemens Sinumerik ONE	✓	✓	✓	✓	✓
Mitsubishi M800/M80	✓	✓	✓	✓	×
Heidenhain TNC 640	✓	✓	✓	×	×
Mazatrol moothAi	✓	✓	✓	✓	✓

Uses of important switches & buttons and controls

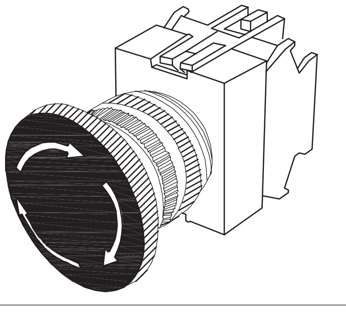
Objectives: At the end of this lesson you shall be able to
- **state the uses of emergency stop**
- **state the purpose of reset button**
- **state the uses of override of feed rate & spindle speed over ride**
- **list the functions of edit, lock and ON/OFF buttons.**

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.

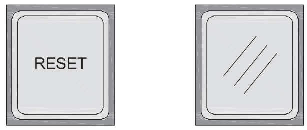
Fig 1



CN20N189411

Reset key (Fig 2)

Fig 2



CN20N189412

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.

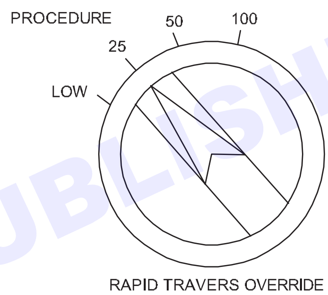
Rapid motion override

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 feed rates 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



CN20N189413

- 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 over ride

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.

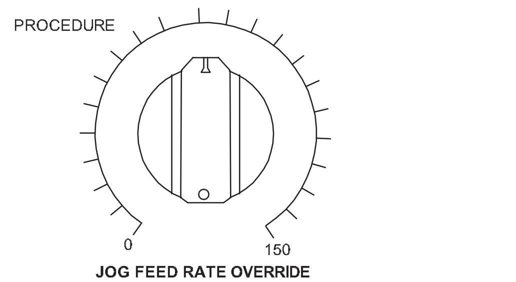
Feed rate can be over ridden with in a large range, typically from 0% to 200% or at least 0% to 150%. When the feed rate override switch is set to 0%, the CNC machine will stop the cutting motion. Some CNC machines do not have the 0% percent setting and start at 10%. This can be change by a system parameter. 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 feed rate override

- Set the feed rate 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 feed rate 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 feed rate remains as specified by program.

Machine lock and auxiliary function lock

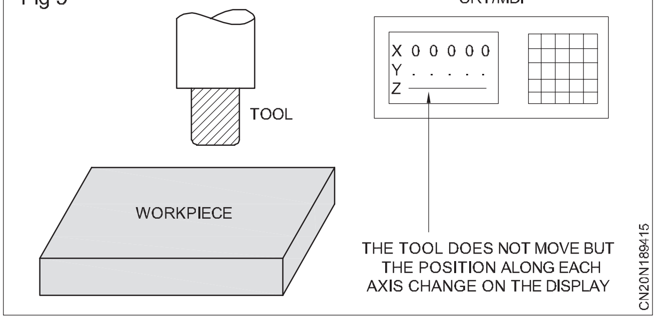
To display the change in the position without moving the tool, use machine lock. There are two types of machine lock: all–axis machine lock, which stops the movement along all axes, and Z–axis machine lock, which stops the movement along Z–axis only. In addition, auxiliary function lock, which disables 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).
- 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.

Fig 5



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

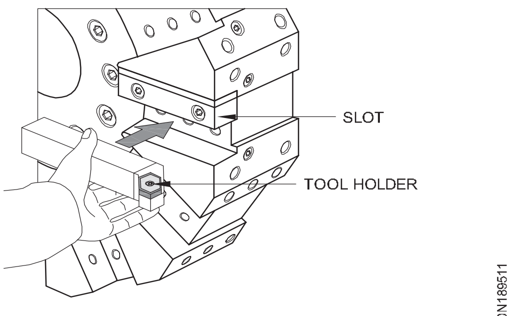
- M00, M01, M02, M30, M98, M99, M198, and M199 and commands are executed even in the auxiliary function lock state.

Tooling systems for CNC turning centres

Objectives: At the end of this lesson you shall be able to
- **state the tool setting in CNC turning centres**
- **describe the ISO standard for specifying inserts.**

A normal tool is lamped by inserting it in a slot in the turret and tightening screws to hold it firmly (Fig 1). While doing this, care must be taken to seat the holder properly against available butting surfaces. Finally, the tool offsets must be taken by touching the part or taking skin cuts.

Fig 1



NORMAL TOOLING

Rectangular or square type shank tools (Fig 2) are fitted in the tool holders (Fig 3) and the tool holders are fitted in turret.

Fig 2

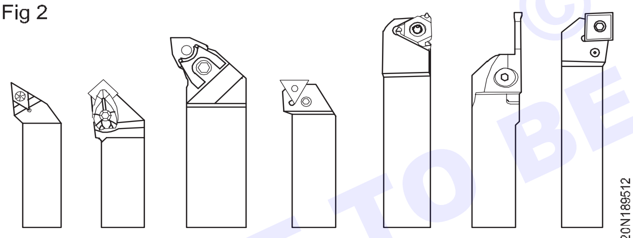
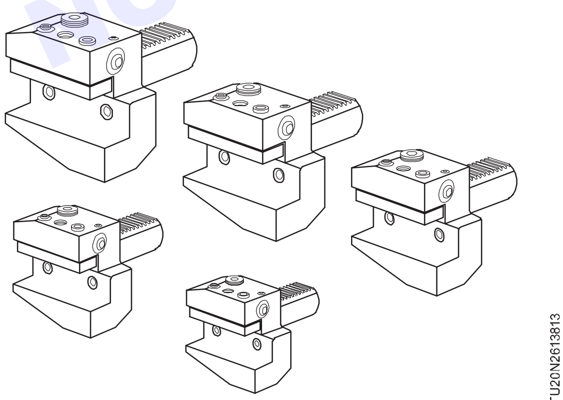


Fig 3



Whereas the round shank tools are fitted in round shank tool holders as shown in Fig 4 and the tool holders along with the tools are fitted in turret as shown in Fig 5.

Quick change tool holder systems on CNC lathes (Figs 6 & 7): A quick change tool holder, or quick change tooling, is a modular system that enables tools to be changed very fast on the machine during.

Fig 4

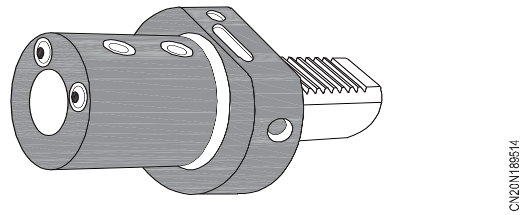
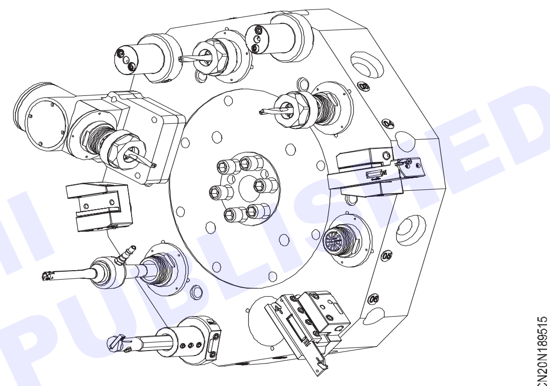


Fig 5



The system reduces machine downtime for tool change during setup changes, and for changing worn out inserts during machining.

With a quick change tool holder, you do not clamp the tool directly in the turret. The tool is replaced by an interchangeable cutting unit. You insert the cutting unit into a standardized locking unit. There are different locking units for internal and external tools, and the locking units remain on the turret forever - there is no need to keep removing them for every part. The interface between the units is such that the cutting unit will only sit in a particular position in the locking unit, and will sit accurately every time.

Fig 6

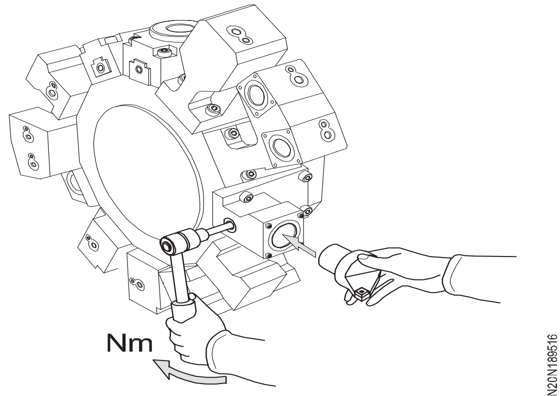
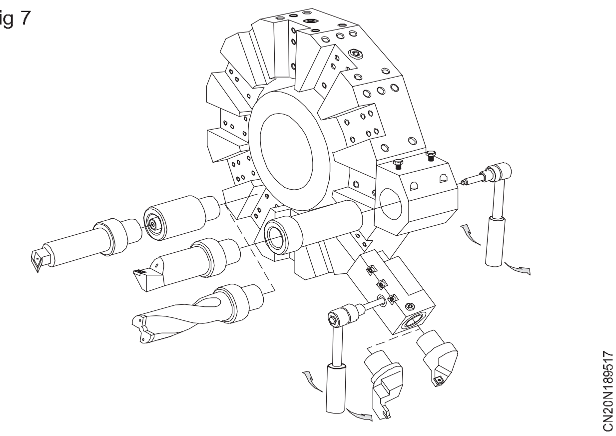


Fig 7

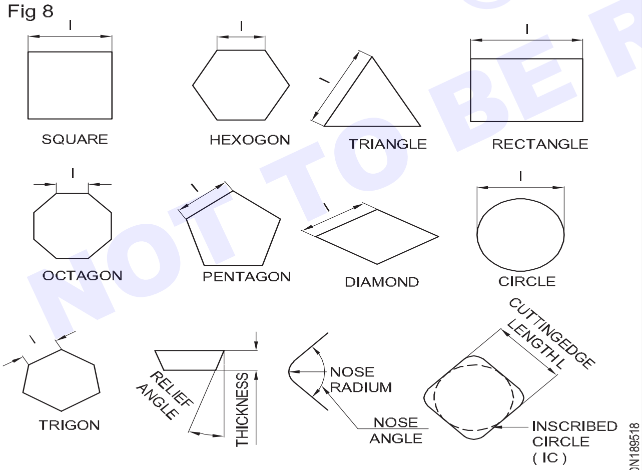


When doing the setup for a new part, you just quickly insert and clamp a new set of cutting units into the appropriate locking units. The tool offsets are pre-determined for each cutting unit, and remain the same every time that you put in the unit.

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 8)

Fig 8



INDEXABLE TOOL INSERTS

Inserts are produced in various sizes and thicknesses. Smallest possible size is chosen to produce the desired depth of cut. Thickness of an insert affects its strength. Hence, for a large depth of cut and feed, a thicker insert is chosen.

ISO standard is commonly followed for specifying inserts. An example is CNMG120408. The first letter, C in this case, indicates the shape of the insert. The common types are:

Symbol	Shape
S	Square
T	Triangular
H	Hexagonal
O	Octagonal
P	Pentagonal
L	Rectangular
R	Round
A, B, K	Parallelogram (nose angles 85°, 82° and 55° respectively
C, D, E, F, M, V	Diamond shaped or rhombic (nose angles 80°, 55°, 75°, 50°, 86°, 35° respectively
W	Trigon (nose angle 80°)

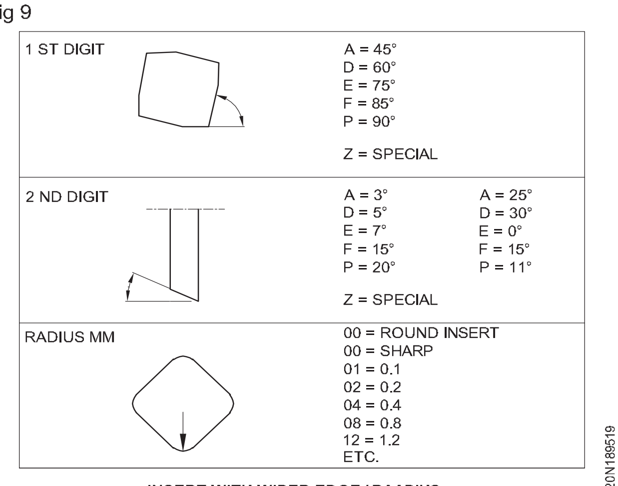
The second letter specifies the relief angles

Symbol	Relief Angle
N	0°
A	3°
B	5°
C	7°
P	11°
D	15°
E	20°
F	25°
G	30°

The third letter specifies tolerances on various dimensions (Fig 9) (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).

The value of first, second and radius are shown in Fig 9.

Fig 9

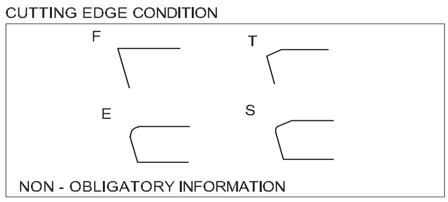


INSERT WITH WIPER EDGE / RAADIUS

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 10)**

Fig 10



F for sharp,

T for chamfered,

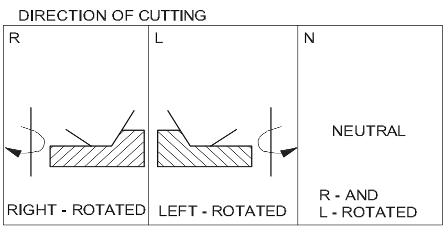
E for honed and

S for chamfered and honed.

This information, however, is non-obligatory.

- **Cutting direction (Fig 11)**

Fig 11



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 T Q		cylindrical with 40°-60° counter sink	without
			single-sided
B		cylindrical with 40°-60° double countersink	without
H			double-sided
C		cylindrical with 70°-90° counter-sink	without
J			single-sided
X		cylindrical with 70°-90° double countersink	without double-sided
		special shape	

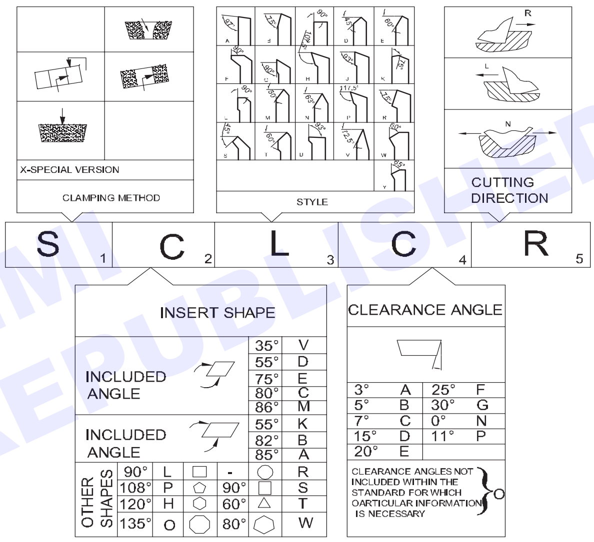
L for machining with left - rotated (CCW) spindle (M04),

R for machining with right - rotated (CW) spindle (M03) and N for both left-and right - rotated.

The appropriate character designations are appended to the right, after the radius specification.

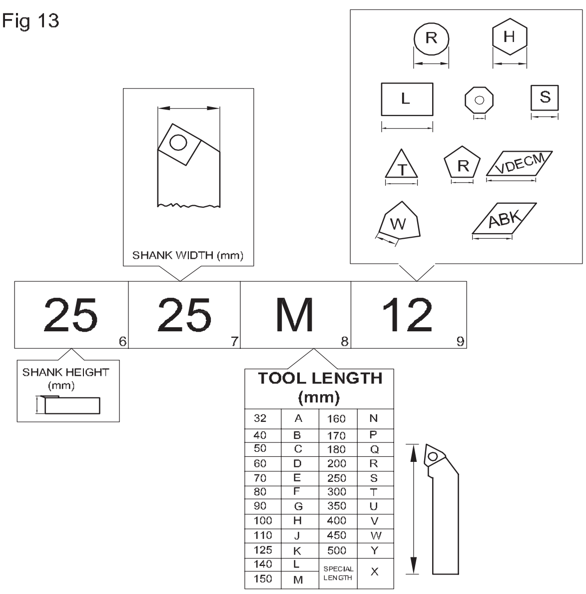
Tool holders for lathe: There is an ISO designation system for tool holders also, to suit various types of inserts. The first five characters describe insert clamping method, compatible insert shape, insert holding style of the tool holder (side cutting edge angle/end cutting edge angle and straight 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 12 & 13)

Fig 12



ISO DESIGNATION FOR LATHE TOOL HOLDERS

Fig 13

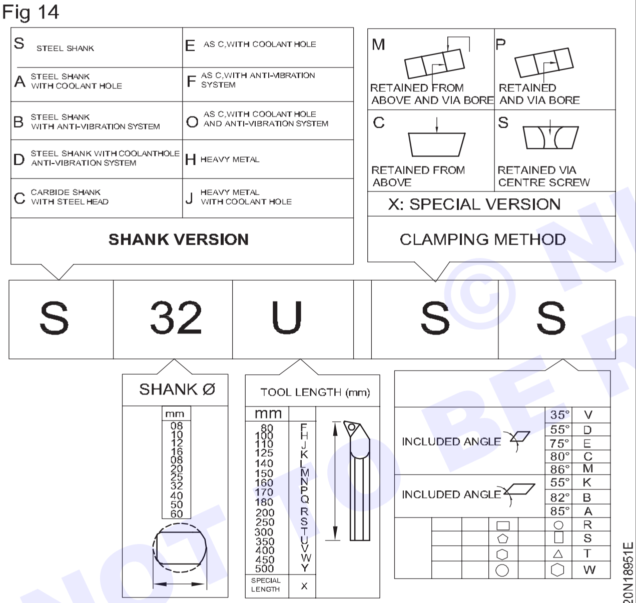


ISO DESIGNATIOM FOR LATHE TOOL HOLDERS

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 14 & 15)

Fig 14

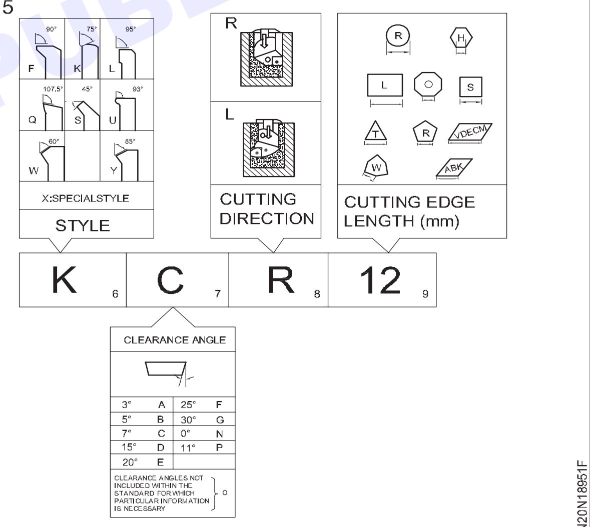


ISO DESIGNATION FOR LATHE BORING BARS

There is a similar ISO designation system for the internal tool holders 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)	ϕ32 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 15



ISO DESIGNATION FOR LATHE BORING BAR

Work holding for CNC turning

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

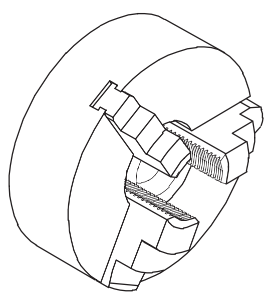
Now while gripping the stopper, bore out the soft jaws to the desired diameter and finish required to grip the work piece. Ideally the gripping pressure should be set to the same pressure, where the work piece will be clamped at.

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.

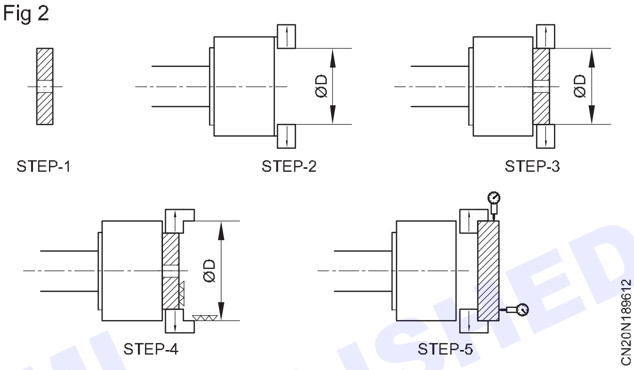
Fig 1



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.

Chuck only

Use it L/D is Less than 3 you can go up to L/D 5 with reduced cutting parameter which reduce the cutting force.

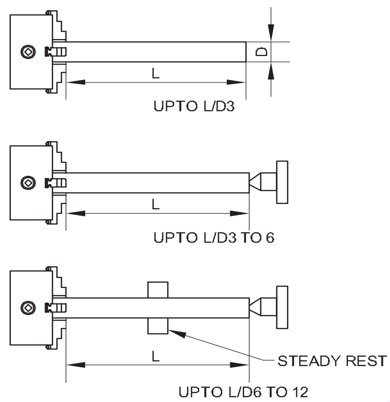
Tail stock

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.

Fig 1



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.

Introduction to turn mill centre

Objectives: At the end of this lesson you shall be able to
- **state what is turn mill centre**
- **list the features of turn mill centre**
- **list the application of turn mill centre**
- **describe the constructional details of turn mill centre.**

A Turn-Mill Centre is a highly versatile machining tool that combines the capabilities of a lathe and a milling machine in a single unit. This enables the simultaneous turning and milling operations, making it suitable for producing complex parts with high precision and efficiency. It is widely used in industries such as automotive, aerospace, and medical manufacturing.

Features of turn-mill centres

1 **Combined operations**

- **Turning:** For rotational machining (e.g., cylindrical and conical shapes).
- **Milling:** For non-rotational machining (e.g., flat surfaces, slots, and intricate shapes).

2 **Multi-axis configuration**

- Typically, these machines come with 3 to 9 axes, allowing for highly complex geometries.
- Axes include X, Y, Z (linear), and rotational axes (A, B, C).

3 **Live tooling**

- The machine uses live tooling heads to perform milling, drilling, or tapping without repositioning the workpiece.

4 **Integrated workpiece handling**

- Equipped with automated chucking systems, tailstocks, or sub-spindles for handling the workpiece.

5 **High precision**

- CNC (Computer Numerical Control) integration ensures precision and repeatability in machining operations.

6 **Efficiency**

- Eliminates the need to transfer the workpiece between machines, reducing setup time and potential errors.

Applications

- Production of components with complex geometries such as shafts, valves, impellers, and medical implants.
- Manufacturing parts requiring tight tolerances and high surface finishes.
- Ideal for small to medium batch production where versatility and quick setup are critical.

Advantages

- **Time-saving:** Performs multiple operations in one setup.
- **Space efficiency:** Combines two machine tools into one.
- **Improved accuracy:** Reduces handling errors.
- **Cost-effective:** Cuts down on tooling and labor costs.

Construction details of a turn-mill centre

A Turn-Mill Centre is a sophisticated CNC (Computer Numerical Control) machine that integrates turning and milling functionalities. Its construction is designed for versatility, precision, and efficiency in machining complex parts. Below are the key construction elements and their details:

1 **Machine bed**

- **Material:** Typically made of cast iron or composite materials to ensure rigidity and vibration damping.
- **Design:** Heavy and robust to minimize thermal expansion and vibration during machining.
- **Purpose**
  - Supports all components of the machine.
  - Provides a stable platform for high-precision machining.

2 **Spindle system**

Main spindle

- Drives the rotation of the workpiece during turning operations.
- Equipped with a chuck or collet for clamping the workpiece.
- High torque for turning operations and high RPM for milling and drilling.

Sub-spindle (Optional)

- Positioned opposite the main spindle for machining the back side of the workpiece.
- Works synchronously with the main spindle for seamless part transfer.

Dual spindle configuration

- Includes both main and secondary spindles for simultaneous front and back machining.

3 **Tool turret**

- **Multi-tool turret**
  - Houses multiple tools (turning and milling tools).
  - Can rotate to position the required tool in the machining area.
- **Live tooling**
  - Enables milling, drilling, and tapping operations while the workpiece is clamped.
  - Powered by servo motors or gear-driven systems.
- BMT (Bolt Mount Turret) or VDI (Quick-Change) tool mounting systems ensure rapid tool changes.

4 **Milling head**

- Mounted on a movable axis (e.g., Y-axis) to perform milling operations.
- Can tilt or rotate (in multi-axis configurations) for machining complex geometries.
- Driven by a high-speed motor for precision milling, drilling, and tapping.

5 **Guideways**

- **Linear guideways**
  - Ensure smooth and precise movement of the machine's components.
  - Common types: Ball screws, roller guideways.
- **Box Guideways**
  - Used in heavy-duty machines for enhanced rigidity and load-bearing capacity.

6 **CNC controller**

- The brain of the machine that executes the programmed instructions.
- Features
  - Multi-axis control for complex machining.
  - Real-time toolpath simulation.
  - Adaptive control for tool wear and thermal compensation.
- Common brands: FANUC, Siemens, Mitsubishi, Haas, etc.

7 **Axis configuration**

- X, Y, Z Axes
  - Control linear movements for turning and milling operations.
- C-Axis
  - Allows rotational control of the spindle for milling features like slots and holes.
- B-Axis (Optional)
  - Provides tilting capability for the milling head for angled operations.

8 **Chuck system**

- Holds the workpiece securely during operations.
- Types
  - Hydraulic chucks for automated clamping.
  - Pneumatic chucks for light-duty operations.

9 **Coolant system**

- Delivers coolant to the cutting zone to manage heat generation and tool wear.
- Features
  - High-pressure pumps for deep hole drilling.
  - Integrated filtration to remove chips and contaminants.

10 **Chip conveyor**

- Removes chips from the machining area automatically.
- Types
  - Belt-type or scraper-type conveyors based on the material and chip size.

11 **Automation features**

- **Robotic arms:** For loading/unloading workpieces.
- **Bar feeders:** Supply raw materials for continuous machining.
- **Part catchers:** Automatically collect finished parts from the sub-spindle.

12 **Safety enclosure**

- Protects operators from flying chips and coolant splashes.
- Includes a transparent window for visibility and lighting inside the machining area.

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

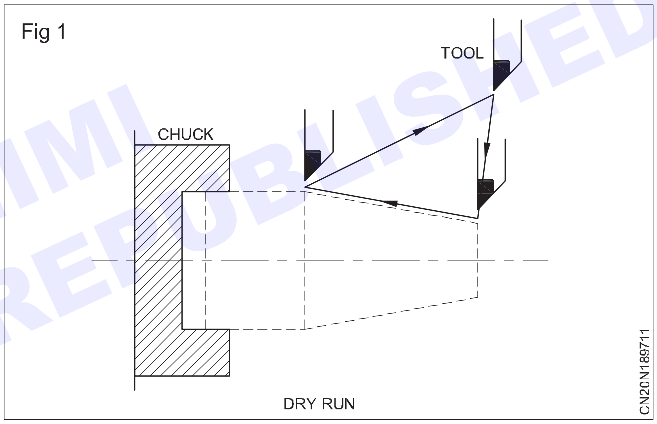
Set the tool offsets: Depending on the types of machine, get the tool geometry and wear offsets, tool length offsets and cutter radius offset. This setup includes the work offset setting.

Checking the part program

This step is the first evaluation of the part program. The part may be removed from the fixture temporarily. 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)



Step for dry run

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.

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

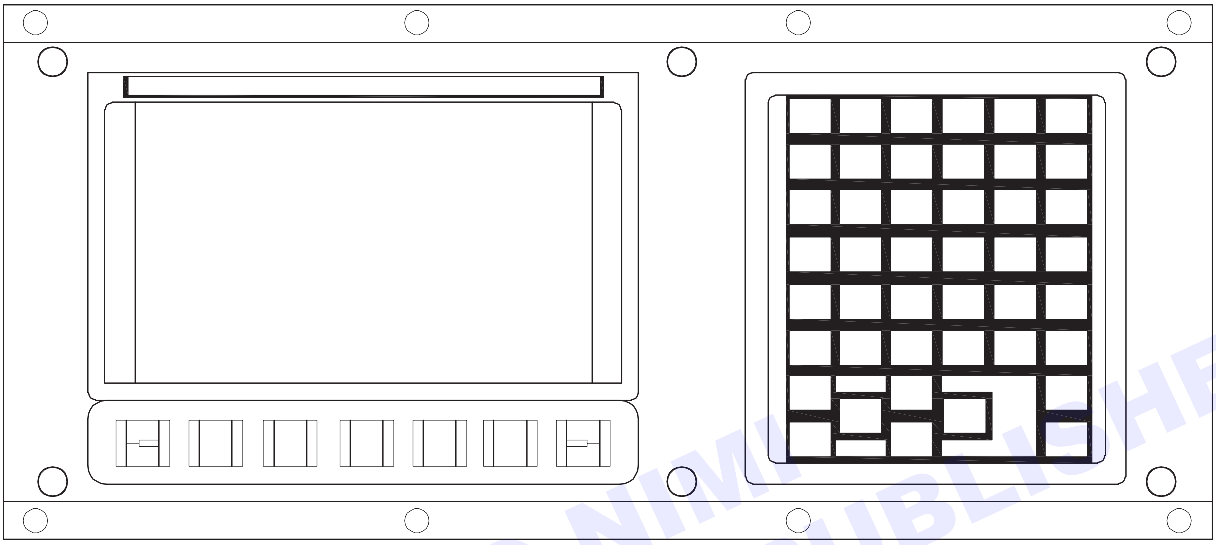
The tool path simulation shows the outline of finished part and the tool motions . The tool motions are identified by a dashed line(rapid motion) and a solid line(cutting motion).

Machine control and its functions

Objectives: At the end of this lesson you shall be able to
- **describe the machine control specification**
- **state the functions of CNC machine controls.**

Machine control functions (Fig 1)

Fig 1



CN20N189811

MACHINE CONTROL FUNCTIONS

The commonly used control functions of the FANUC OI mate - TB controller are explained below.

The control system has a 8.4” LCD monitor and the layout of the LCD screen and the alpha - numeric key pad is as given below.

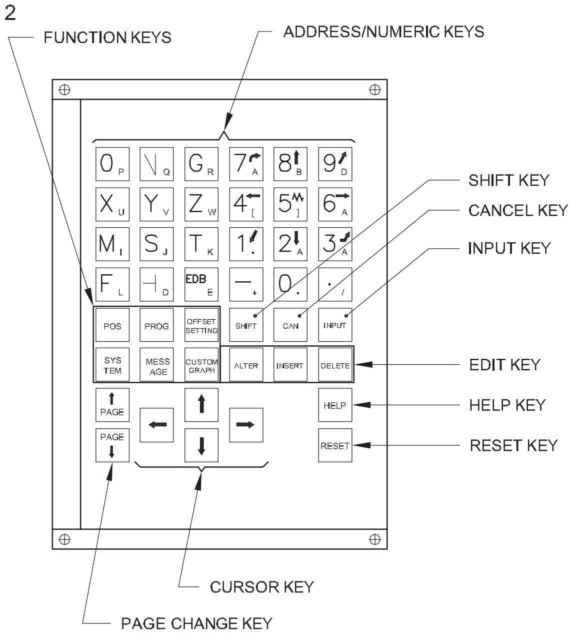
Monochrome

LCD/MDI unit

The layout of the various control keys in the MDI panel are as follows.

MDI key location (Fig 2)

Fig 2



CN20N189812

MDI KEY LOCATION

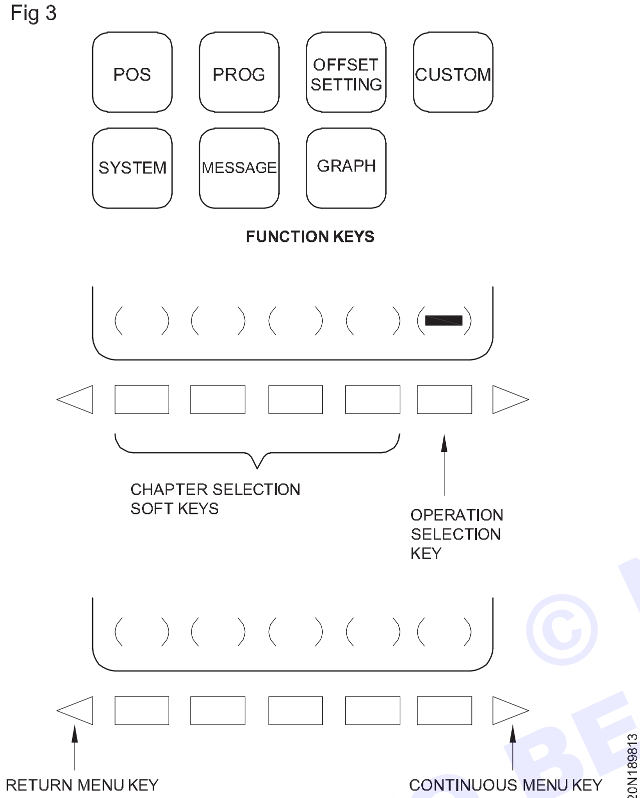
Explanation of the MDI keyboard

S No.	Name	Explanation
1	Reset key RESET	Press this key to reset the CNC, to cancel an alarm, etc.
2	Help key SYSTEM	Press this button to use the help function when uncertain about the operation of an MDI key (help function).
3	Soft key functions	The soft keys have various functions, according to the applications. The soft key are displayed at the bottom of the screen.
4	Address and numeric keys N (4 ←	Press these keys to input alphabetic, numeric, and other characters.
5	Shift key SHIFT	Some keys have two characters on their keytop. Pressing the <shift> key switches the characters. Special character E is displayed on the screen when a character indicated at the bottom right corner on the keytop can be entered.
6	Input key INPUT	When an address or a numerical key is pressed, the data is input to the buffer, and it is displayed on the screen. To copy the data in the key input buffer to the offset register, etc., press the input key. This key is equivalent to the [Input] key of the soft keys, and either can be pressed to produce the same result.
7	Cancel key CAN	Press this key to delete the last character or symbol input to the key input buffer. When the key input buffer displays >N001X100Z_ and the cancel can key is pressed, Z is cancelled and >N001X100_ is displayed.
8	Program edit keys ALTER RESET DELETE	Press these keys when editing the program. ALTER RESET DELETE ALTER INSERT DELETE
9	Function keys POS PROG	Press these keys to switch display screens for each function.
10	Cursor move keys ← ↑ ↓ →	There are four different cursor move keys. → This key is used to move the cursor to the right or in the forward direction. The cursor is moved in short units in the forward direction. ← This key is used to move the cursor to the left or in the reverse direction. The cursor is moved in short units in the reverse direction. ↓ This key is used to move the cursor in a downward or forward direction. The cursor is moved in large units in the forward direction. ↑ This key is used to move the cursor in an upward or reverse direction. The cursor is moved in large units in the reverse direction.

S. No.	Name	Explanation
11	Page change keys	Two kinds of page change keys are described below ↑ PAGE This key is used to changeover the page on the screen in the forward direction. PAGE ↓ This key is used to changeover the page on the screen in the reverse direction.

General screen operations (Fig 3)

Fig 3



GENERAL SCREEN OPERATIONS

Press a function key on the MDI panel. The chapter selection soft keys that belong to the selected function appear.

Press one of the chapter selection soft keys. The screen for the selected chapter appears. If the soft key for a target chapter is not displayed, press the continuous menu key (next - menu key). In some cases, additional chapters can be selected within a chapter.

When the target chapter screen is displayed, press the operation selection key to display data to be manipulated.

To redisplay the chapter selection soft keys, press the return menu key.

The general screen display procedure is explained above. However, the actual display procedure varies from one screen to another. For details, see the description of individual operations.

Function keys

Function keys are provided to select the type of screen to be displayed. The following function keys are provided on the MDI panel.

POS Press this key to display the position screen.

PROG Press this key to display the program screen.

OFFSET SETTING Press this key to display the offset/setting screen.

SYSTEM Press this key to display the system screen.

MESSAGE Press this key to display the message screen.

GRAPH Press this key to display the graphics screen.

CUSTOM Press this key to display the custom screen (conversational acro screen).

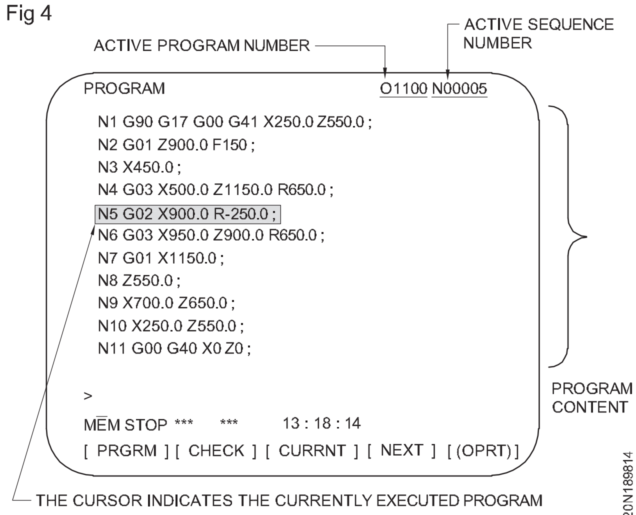
Program display

The contents of the currently active program are dis displayed. In addition, the programs scheduled next and the program list are displayed. (Figs 4 & 5)

The cursor indicates the currently executed location.

Current position display: The current position of the tool is displayed with the coordinate values. The distance from the current position to the target position can also be displayed. (Fig 6 & 7)

Fig 4



PROGRAM CONTENTS DISPLAY

Fig 5

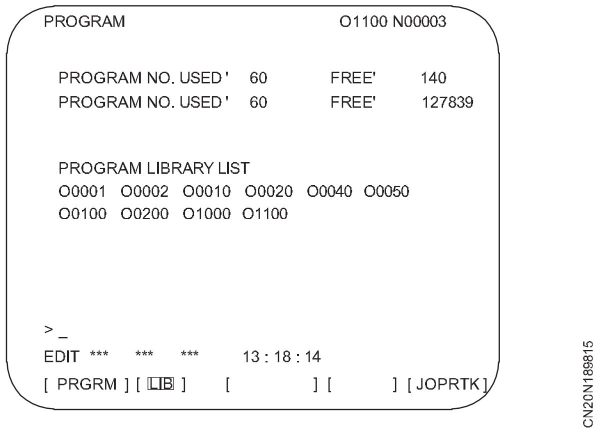
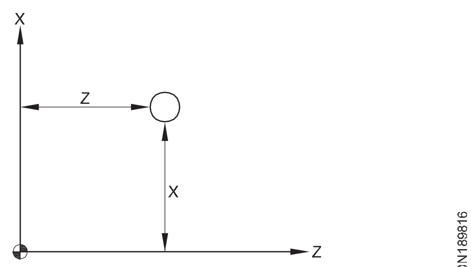
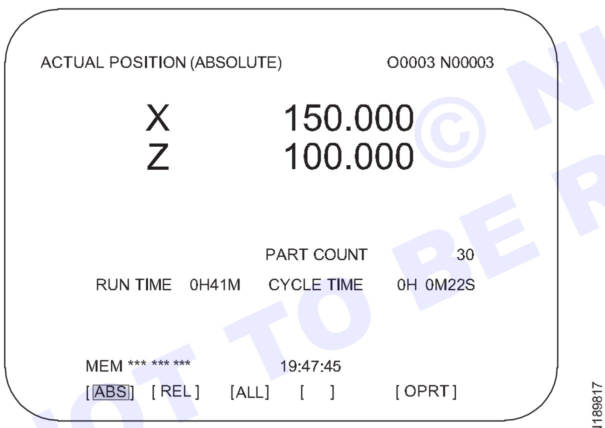


Fig 6



WORKPIECE COORDINATE SYSTEM

Fig 7



POSITION DISPLAY IN THE WORK (ABSOLUTE) COORDINATE SYSTEM

When a trouble occurs during operation, alarm numbers and alarm message are displayed on CRT screen. See APPENDIX G for the list of alarms and their meanings. (Fig 8)

Parts count display run time display: Two types of run time and number of parts are displayed on the screen. (Fig 9)

Graphic display: The graphic can be used to draw a tool path for automatic operation and manual operation, thereby indicating the progress of cutting and the position of the tool. (Fig 10)

Fig 8

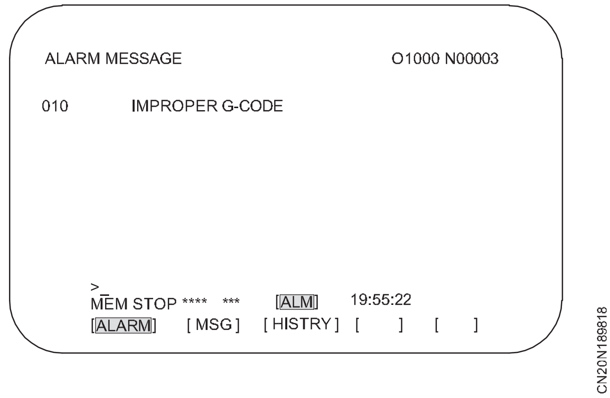


Fig 9

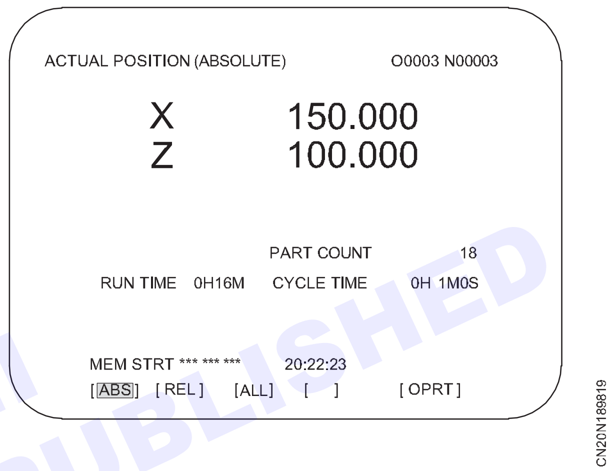
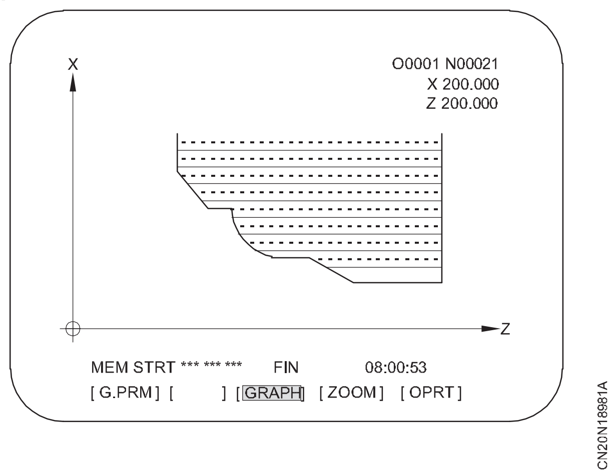


Fig 10



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.

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 1)

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 2)

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.

Fig 1

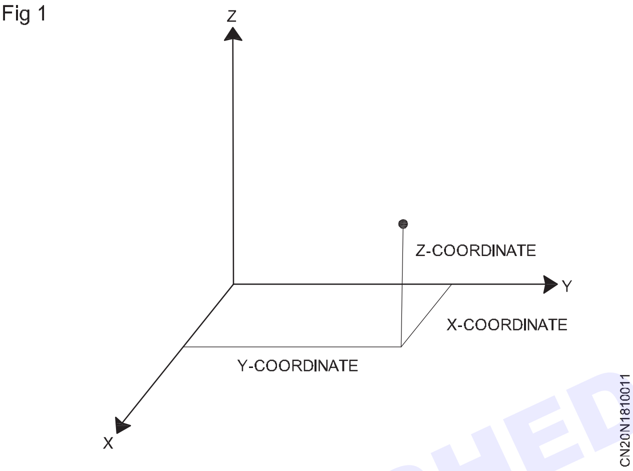
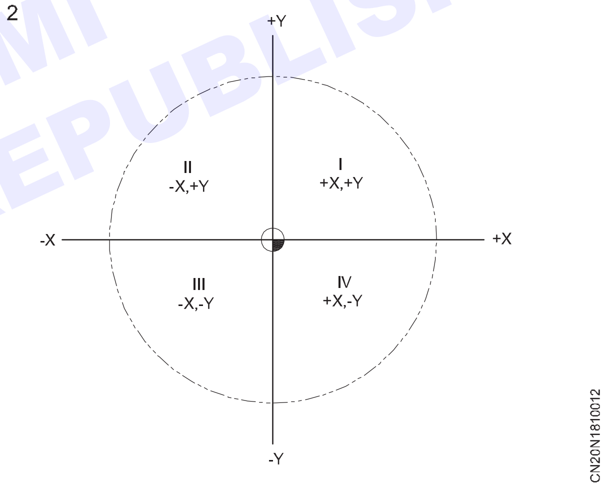


Fig 2



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-ordinate 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 3)

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 4)

Fig 3

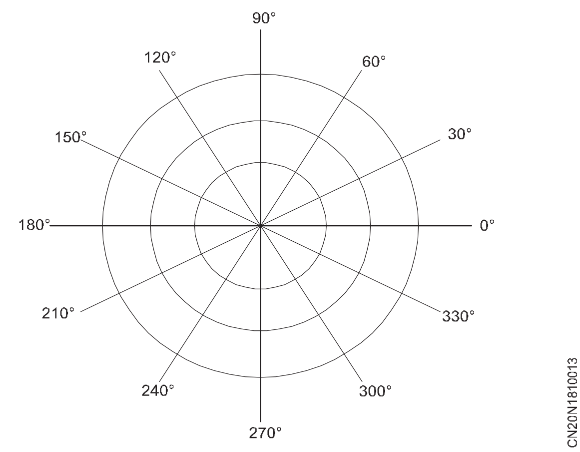
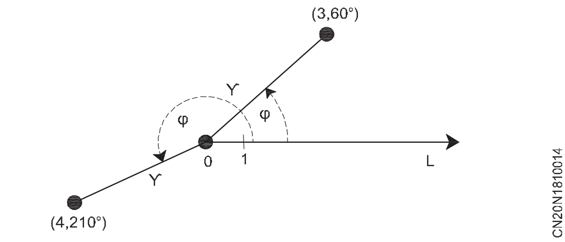


Fig 4



Points in the polar co-ordinate system with pole O and polar axis L.

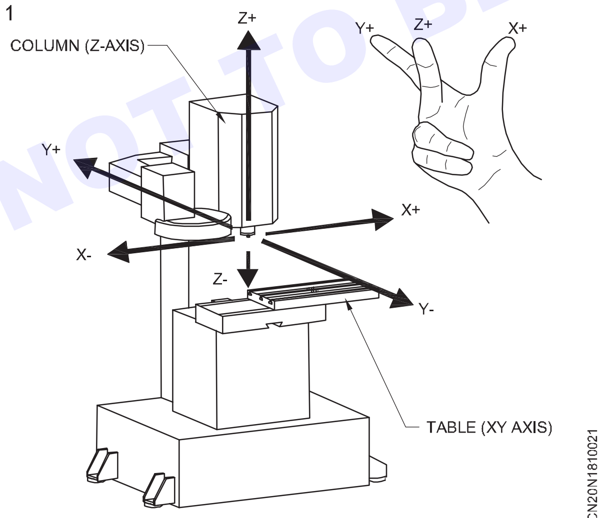
The point with radial coordinate 3 and angular coordinate 60 degrees or (3, 60°). The point (4, 210°).

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)

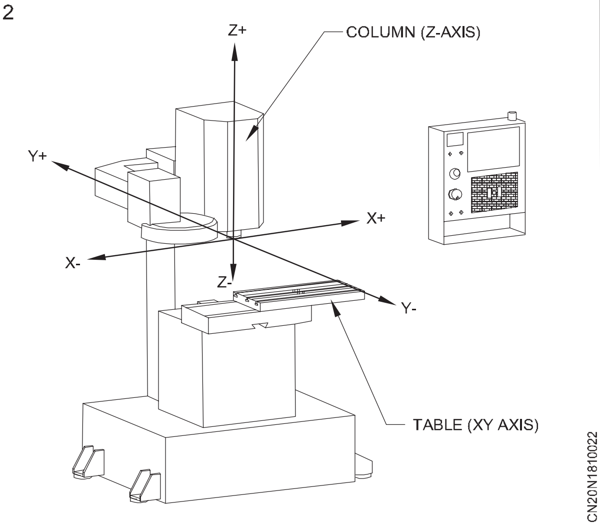
Fig 1



- 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)

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.

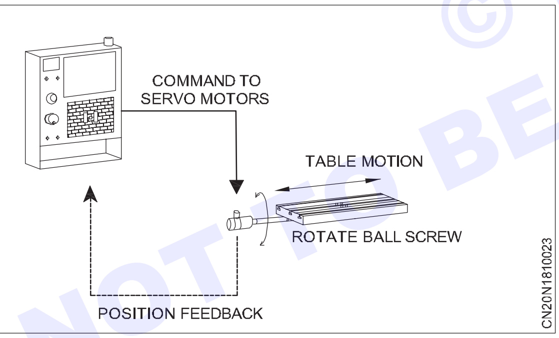
CNC Machine's Origin

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.

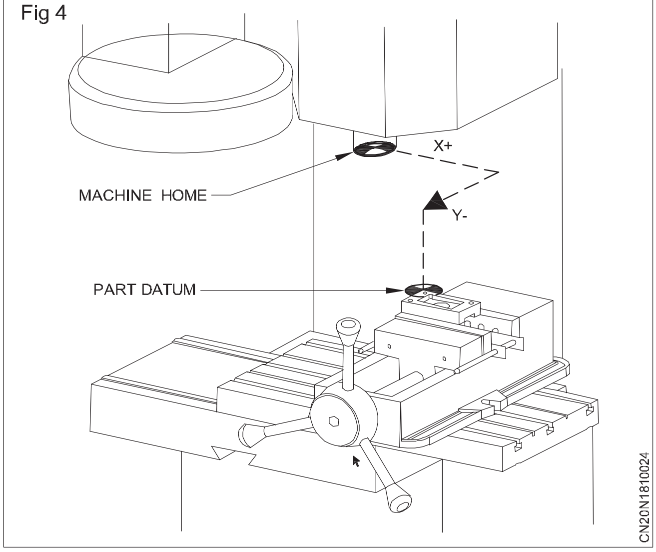
As we mentioned above, human operators will use a WCS, which provides an easy set of coordinates for writing a CNC program. However, these coordinates are always different from a machine's coordinates, so how does your CNC machine line the two up? With offsets.

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



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, radius and wear offsets are provided in geometry offset page. Enter the corresponding wear amount in respective tool number.

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.

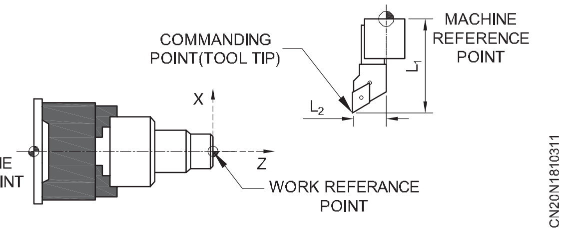
If you want or know more about the reference point then, there are limit switches for every axis in CNC machine to tell the reference point position. So when we take the CNC machine to the reference point those limit switches got active and tell the CNC machine control that the specified axis is reaching 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 θ50.04 mm instead of θ50.00What 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 gains 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 = 47.15

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	57.732mm

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

Reduce 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

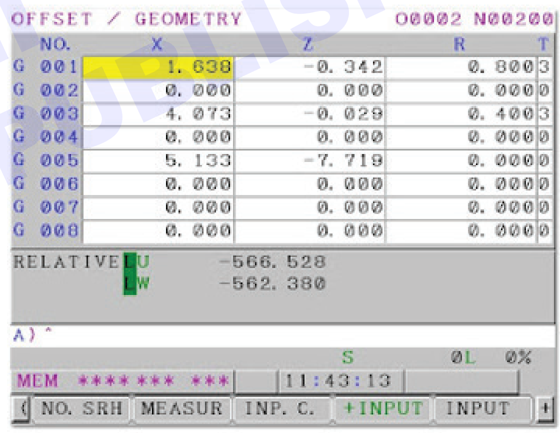
New tool offset in Z 40.00 - 0.50 = 40.50

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

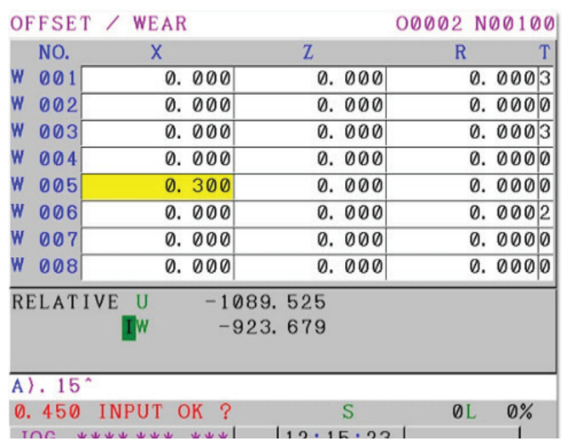
Press offset page /geometry and position the cursor at appropriate tool number and enter the tool data's. (Fig 2)

Fig 2



Similar go to the wear offset page and enter the data's (Fig 3)

Fig 3



Concept of Sub programming in CNC Machining

Objectives: At the end of this lesson you shall be able to
- **state what is sub program in CNC machining**
- **list the features of sub program**
- **explain the structure and its code for sub program.**

Sub programming in CNC machining refers to the use of modular and reusable code blocks within a CNC program. Instead of writing repetitive code for similar operations, a subprogram (or subroutine) is defined separately and can be called multiple times from the main program. This practice enhances code efficiency, reduces errors, and simplifies program management.

Sub programming is a powerful technique that simplifies CNC programming, improves efficiency, and ensures consistency in machining operations

Features of sub programming

1 Reusability

- Subprograms can perform repetitive operations like drilling, pocketing, or contouring without rewriting the same code.

2 Modularity

- Makes CNC programs easier to understand and maintain by dividing the operations into smaller logical units.

3 Parameterization

- Allows passing variables (like position or depth) to subprograms for flexibility in machining similar features at different locations.

4 Nested calls

- A subprogram can call another subprogram, enabling complex operations through hierarchical structuring.

Benefits of Sub programming

- **Efficiency:** Reduces programming time by reusing code.
- **Error reduction:** Minimizes the risk of errors in repetitive operations.
- **Compact code:** Main program remains concise and easier to debug.
- **Flexibility:** Quickly adapt to changes by modifying the subprogram instead of the entire code.

Structure of sub programming

1 Main program

The main program is the primary sequence of commands and contains calls to subprograms as needed.

- **Example**

O1000 (Main Program)

G21 (Metric Units)

G90 (Absolute Positioning)

M98 P2000 L3 (Call Subprogram O2000, Repeat 3 Times)

M30 (End of Program)

2 Subprogram

The subprogram contains the sequence of operations to be executed when called by the main program.

- **Example**

O2000 (Subprogram)

G0 X50 Y50 (Move to Position)

G1 Z-10 F100 (Linear Cut)

G0 Z10 (Retract Tool)

M99 (Return to Main Program)

Codes in sub programming

1 M98

- Used to call a subprogram.
- Syntax: M98 Pxxxx Lyy
  - Pxxxx: Subprogram number.
  - Lyy: Number of times the subprogram is repeated (optional).

2 M99

- Marks the end of the subprogram and returns control to the main program.

3 Oxxxx

- Defines the subprogram, where xxxx is the subprogram number.

Example of a CNC turning machine program that uses a subprogram for machining multiple identical steps, such as facing and turning operations.

Main program (O1000)

O1000 (Main program for turning operations)

G21 (Metric unit)

G90 (Absolute Positioning)

M6 T1 (Select Tool 1 for Facing)

G96 S200 M3 (Constant Surface Speed, Spindle Start)

G0 X50 Z2 (Rapid Move to Safe Position)

M98 P2000 L1 (Call Sub program O2000 for Facing Once)

M6 T2 (Select Tool 2 for Turning)

G96 S300 M3 (Set New Spindle Speed)

G0 X50 Z0 (Rapid Move to Start Position)

M98 P3000 L3 (Call Sub program O3000 for Turning, Repeat 3 Times)

G0 X100 Z100 (Return to Safe Position)

M30 (End of Program)

Sub program for facing (O2000)

plaintext

Copy code

O2000 (Subprogram for Facing Operation)

G0 X50 Z0.2 (Rapid Move to Start Position)

G1 X0 Z0 F0.2 (Face the Surface with Feed Rate 0.2 mm/rev)

G0 Z2 (Retract Tool to Safe Position)

M99 (Return to Main Program)

Sub program for turning (O3000)

plaintext

Copy code

O3000 (Subprogram for Turning Operation)

G0 X50 Z0 (Rapid Move to Start Position)

G1 X40 Z-20 F0.3 (Turn Diameter to 40 mm for 20 mm Length)

G1 X30 Z-40 (Reduce Diameter to 30 mm for 20 mm Length)

G0 X50 Z2 (Retract Tool to Safe Position)

M99 (Return to Main Program)

Explanation of the code

Main program (O1000)

1 Setup

- G21: Set units to millimeters.
- G90: Set absolute positioning.

2 Facing operation

- Tool 1 is selected for facing.
- M98 P2000 L1: Calls sub program O2000 once for facing.

3 Turning operation

- Tool 2 is selected for turning.
- M98 P3000 L3: Calls sub program O3000 three times for repeated turning operations.

4 End of program

- M30: Marks the end of the main program.

Facing sub program (O2000)

1 Moves the tool to the safe position near the workpiece.

2 Executes a facing cut from X50 Z0.2 to X0 Z0.

3 Retracts the tool to a safe height.

Turning sub program (O3000)

1 Moves the tool to the starting position for turning.

2 Executes two turning operations:

- Reduces the diameter from 50 mm to 40 mm over 20 mm length.
- Further reduces the diameter from 40 mm to 30 mm over another 20 mm.

3 Retracts the tool to a safe position.

Output action

1 The machine first faces the part using the facing subprogram.

2 Then it performs three turning passes to machine the required profile.

Introduction to CNC turning centre program using CAM

Objectives: At the end of this lesson you shall be able to
- **describe the fundamentals of CNC turning programs**
- **state the key operations performed on CNC turning centers.**

Introduction

CNC (Computer Numerical Control) Turning Centers are advanced machines designed for precision manufacturing of cylindrical parts, such as shafts, bolts, rings etc. The process of programming CNC Turning Centers has been greatly simplified and enhanced with the use of CAM (Computer-Aided Manufacturing) software. CAM enables machinists to create detailed machining programs that control the CNC machine, automating processes and ensuring accuracy. By integrating CAD (Computer-Aided Design) files with CAM software, operators can design and execute multiple operations on a workpiece, such as facing, roughing, grooving, threading, drilling, and finishing.

The CNC Turning program is generated in CAM software through a systematic approach, which involves importing the design of the component, selecting machining operations, defining parameters such as cutting speed and feed rate, and simulating the entire process before generating the final program. This workflow ensures that the machine performs the required operations seamlessly, without the need for manual coding or trial-and-error methods.

CNC Turning program using CAM

One of the standout features of CAM is its ability to handle a wide range of machining operations in CNC Turning Centers. These operations are essential for shaping raw materials into finished components with intricate features. Below are some key operations that can be performed using CAM for CNC turning

1 Facing

Facing is the first operation performed in most turning processes, where material is removed from the end of the workpiece to create a flat surface. CAM simplifies this operation by allowing machinists to define the area, tool approach, and cutting parameters in detail. Accurate facing ensures a clean starting surface for subsequent operations, improving the overall quality of the finished part.

2 Roughing

Roughing is used to remove the bulk of material quickly, shaping the workpiece close to its final dimensions. CAM software optimizes roughing operations by generating toolpaths that maximize material removal while maintaining tool longevity. This operation is particularly important for reducing machining time, especially when dealing with larger components or complex geometries.

3 Grooving

Grooving involves creating channels or recesses on the workpiece. CAM makes it easy to program the toolpaths for grooves with specific depths, widths, and positions. These grooves may be functional, such as providing seating for O-rings, or purely aesthetic. With CAM, machinists can preview the groove operation in a simulated environment to ensure it meets design specifications.

4 Threading

Threading is one of the most intricate operations in CNC turning, requiring high precision to create internal or external threads. CAM software allows users to select thread types (metric, imperial, or custom), pitches, and profiles. It calculates the required tool movements, ensuring that the threading operation is accurate and consistent, even for complex thread geometries.

5 Drilling and boring

CAM enables the programming of drilling and boring operations with a high degree of accuracy. Users can define the location, diameter, and depth of holes, ensuring they meet the design requirements. Boring operations, which refine and enlarge existing holes, are also easily programmed using CAM, enabling precise control over internal dimensions.

6 Finishing

The finishing operation refines the surface of the workpiece to achieve the desired dimensions and surface finish. CAM provides detailed control over the tool's cutting parameters, ensuring a smooth and polished final product. This step is critical in industries where surface quality is of utmost importance, such as aerospace and medical device manufacturing.

7 Parting or Cut-off

Parting separates the finished part from the remaining stock material. CAM ensures a clean and efficient cut-off operation, minimizing tool wear and preventing damage to the workpiece during the separation process.

Benefits of CAM for CNC turning program creation

1 Automation of complex processes

CAM automates the generation of toolpaths for various turning operations, reducing the need for manual programming. This saves time and minimizes errors, especially for intricate designs.

2 Toolpath optimization

CAM software ensures that toolpaths are optimized for efficiency, reducing machining time while maintaining precision. This is particularly useful for roughing and finishing operations.

3 Simulation and verification

Before running the program on the machine, CAM provides a simulation of the entire machining process. This helps identify potential issues like tool collisions, inefficient paths, or machining errors.

4 Consistency and reusability

CAM-generated programs can be reused for batch production, ensuring consistent quality across all parts. Once a program is created, minor modifications can be made to accommodate design changes without starting from scratch.

5 Enhanced safety

CAM reduces manual intervention, minimizing the risk of human error. The simulation feature also helps detect and eliminate potential hazards, making the machining process safer.

6 Ease of customization

CAM allows machinists to customize machining parameters and toolpaths to meet specific design requirements. For example, if a customer requires a different thread size or groove design, the program can be easily updated.

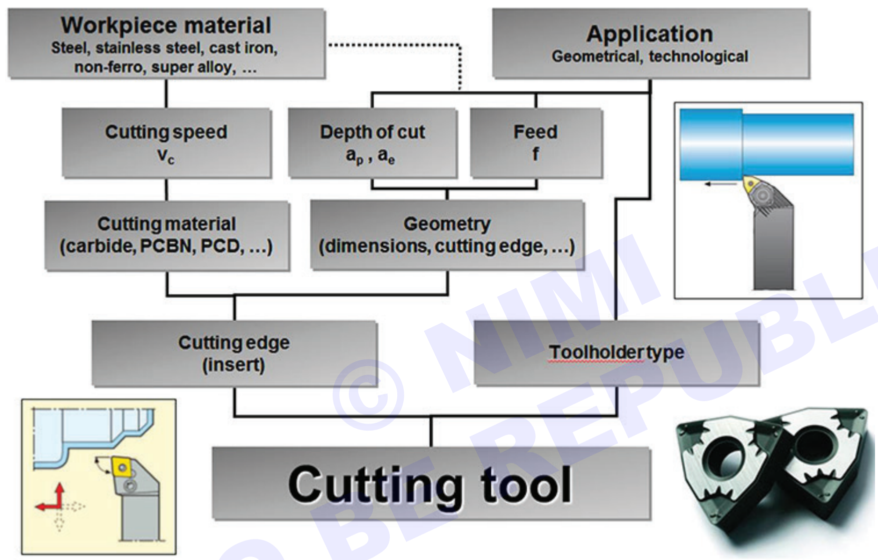
Selection of tools depending on material to be cut

Objectives: At the end of this lesson you shall be able to
- **describe tool materials and their applications**
- **selection of cutting tools in CAM software.**

Cutting tools may not demand high investment in overall production process but it plays a vital role in productivity of the product. Many people do not pay much attention in choosing right cutting tool and incur loss. Cutting tools play an important role while defining a strategy for machining process. It becomes more important when all the new materials like Ni and Ti based alloys, composites and other difficult to cut materials are coming in.

The right selection of cutting parameters, grades and geometries not only do the efficient machining, but also helps to maintain the geometrical tolerances of work piece, improves productivity and maintains the cost per component ratio. The selection of right cutting tools is based on the material to machine, power available on machine, fixture stability, machine dynamics that leads to the right selection of cutting tool material, grades, geometries, depth of cuts and the feeds. (Fig 1)

Fig 1



Cutting tool selection has a direct impact on the proper programming of feeds and speeds at the machine.

However, many other variables that affect feeds and speeds are:

- Work piece material class and condition
- Work piece diameter
- Cutter material
- Cutter geometry
- Type of cut
- Depth of cut
- Condition of the machine

Cutting tool manufacturers publish the general feeds and speeds and recommended usage for the application. Cutting tool manufacturers are often a good place to start for recommendations on tool selection and feeds/ speeds since they rely on customer loyalty. The customer (or potential) should select an insert and grade based on the vendor's recommendation. (Fig 2)

Workpiece material	Tool material	Characteristics
Aluminum, Copper	High-Speed Steel (HSS)	High toughness, withstands moderate cutting speeds, low wear.
Stainless Steel	Carbide	High hardness, heat resistance, suitable for abrasive materials.
Hardened Alloys	Ceramic	Ultra-hard, brittle, requires stable machining conditions.
Titanium	Cubic Boron Nitride (CBN)	Excellent for hardened materials, ideal for finishing operations.

Fig 2

Group	Material	Unit	Continuous Cutting & Finishing		Continuous Cutting & Light Intermittant Cutting		Light Intermittant to Roughing	
			High Speed		Medium Speed		Low Speed	
P	Steel		NC3010	NC3215	NC3220	NC3225	NC3030	PC5300
	Low carbon/mild steel	Surface Speed (m/min)	250-400	100-320	100-280	100-285	50-150	100-170
	High Carbon steel 10-30HRc		220-340	80-270	80-245	80-245	50-150	80-160
	Alloy steel <27HRc		200-330	70-250	70-220	70-220	50-120	80-150
	Pre-hardened steel (30-50HRc)		200-250	50-220	50-160	50-160	50-100	50-130
	Chipbreakers Negative		VQ, VF, MP		MP, VM		VM, HR	
	Chipbreakers Positive		VF, C25, MP		C25, MP, HMP		HMP	
M	Stainless		PC5040	NC9115/PC8105	NC9125/PC8110	PC8115	NC9135	PC5300/PC9030
	Austenitic (303,304, 316, 321)	Surface Speed (m/min)	80-160	160-220	150-200	140-190	100-135	100-135
	Ferritic & martensitic (400 series)		100-180	150-250	120-220	130-200	100-135	100-135
	Duplex / Superduplex		60-140	120-160	100-140	90-130	60-100	60-100
	PH stainless (e.g. 17-4PH)		50-100	50-110	40-110	40-100	30-100	30-100
	Chipbreakers Negative		-	VP,MM	MM, RM, HS, VP		RM, GS, MM	
	Chipbreakers Positive		AK	MM	MP		HMP	
S	Exotics		H01/H05	PC8105	PC8110	PC8115	PC5300	PC5400
	Hastelloy/Inconel/Stellite	(m/min)	-	50-70	45-65	35-55	20-40	20-35
	Titanium alloy		50-90 / 45-70	-	-	-	40-60	30-50
	Chipbreakers Negative		VP1, VP2, HA		HA, VP2, VP3, HS		VP3, HS	
	Chipbreakers Positive		MP, HMP		MP, HMP		HMP	
K	Cast Iron		NC6205		NC6215		NC6215	
	Grey Cast Iron	(m/min)	300-500		150-400		120-300	
	Ductile Cast Iron		220-450		140-380		110-350	
	Chipbreakers Negative		HM, B25		HM, VR		VR, VM	
	Chipbreakers Positive				C25, HMP		HMP	
K	Aluminium		DP150	H01				
	Aluminum	Surface Speed (m/min)	1000-3000	300-1000				
	Aluminum alloy (medium silicon)		600-2500	250-500				
	Aluminum alloy (high silicon)		300-700	-				
	Chipbreakers Negative		-	HA				
	Chipbreakers Positive		-	AK, AR				
H	High Hardened Steel		DNC250		KB420	KB320	KB330	
	50-70 HRc hardened steel	(m/min)	120-220		130-170	120-150	80-110	

Mechanical properties and Thermal conditions

The hardness, toughness, and thermal conductivity of the workpiece material are pivotal in tool selection:

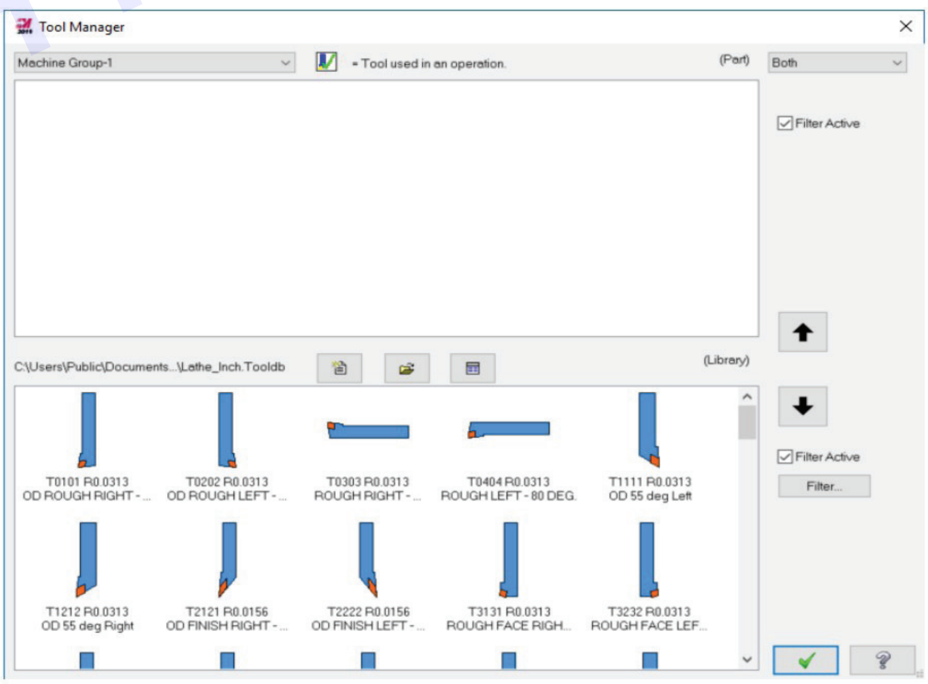
1 **Hardness:** As workpiece hardness increases, the tool must have a correspondingly higher hardness to resist wear and deformation. For example, machining hardened steel requires tools with exceptional hardness, such as CBN or polycrystalline diamond.

2 **Toughness:** Materials prone to chipping or fracturing, like cast iron, demand tools with sufficient toughness to withstand intermittent cutting forces.

3 **Thermal conductivity:** Materials with low thermal conductivity, such as stainless steel, generate high heat at the cutting zone. Tools with high heat resistance, such as those with carbide or ceramic compositions, are necessary to prevent thermal degradation.

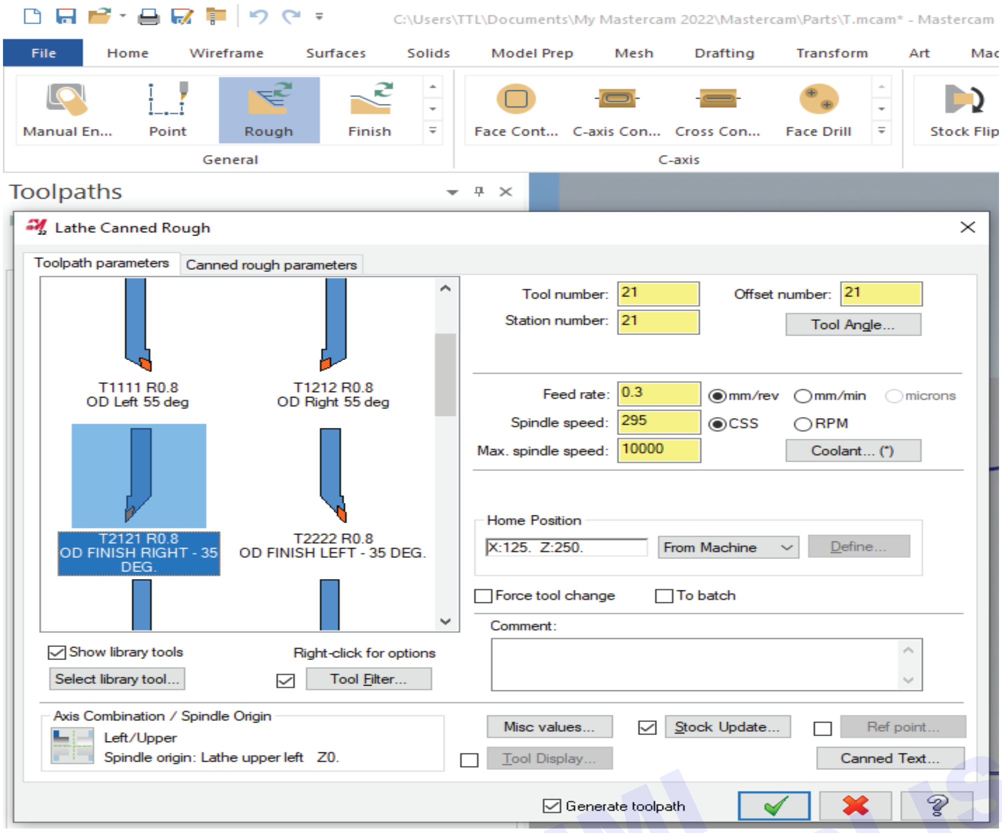
Tool selection in CAM software

Fig 3



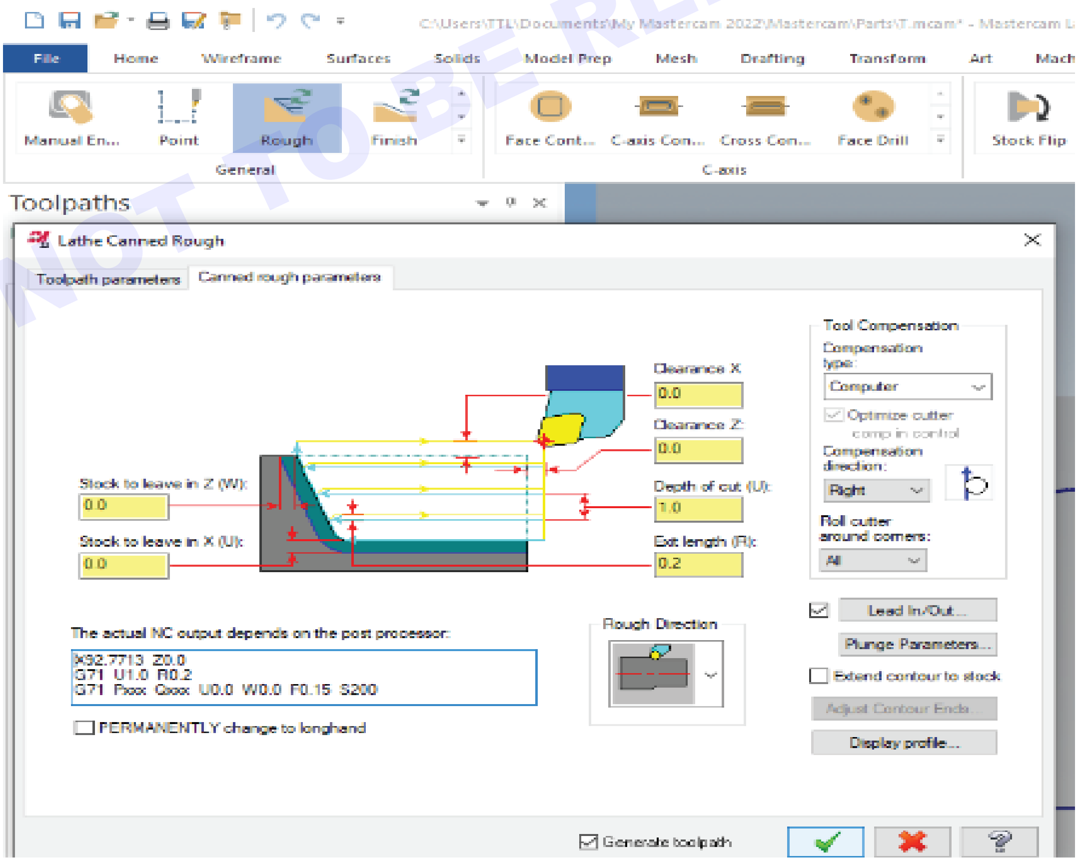
Tool selection for CNC lathe roughing operation

Fig 4



Tool cutting parameters

Fig 5



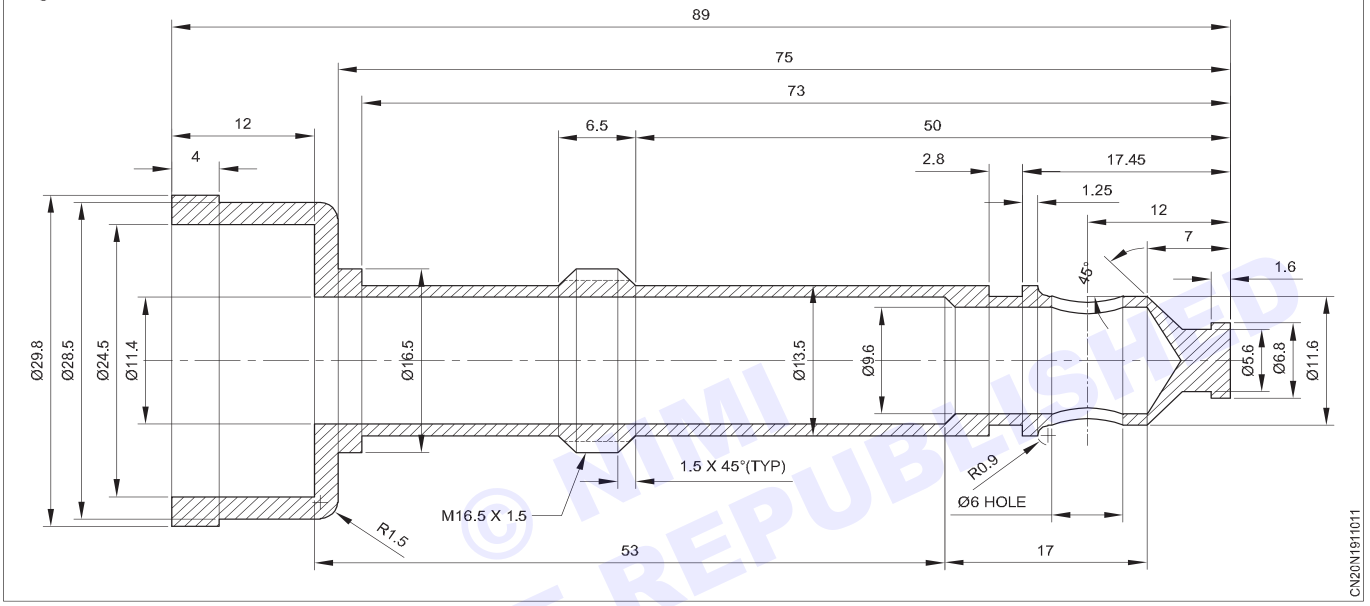
Program creation tool and technique

Objectives: At the end of this lesson you shall be able to
- **creation of CAM program**
- **creation of toolpath and machine group.**

General setup (Fig 1)

Before generating toolpaths for the part, you must prepare Mastercam and the part file. This preparation includes such tasks as selecting a machine definition and defining the stock.

Fig 1



Various tool and techniques are followed to create CAM Program. Some of them are as follow.

1 Setting working environment units
2 Defining the machine setup
3 Orient the Part
4 Using a Lathe Coordinate System
5 Creating a 2D geometry
6 Setting up stock
7 Define the chuck.

Getting ready to work

Select configuration file (Fig 2)

1 Start Mastercam using your preferred method:
- Double-click Mastercam's desktop icon. (or)

Fig 2

- Launch Mastercam from the Windows Start menu.

2 Select the default metric configuration file: (Fig 3)
- Click the File tab.
- Choose Configuration from Mastercam's Backstage View to open the System Configuration dialog box.

Fig 3

a Choose ...\mcamxm.config <Metric> from the Current drop-down list. (Fig 4)

b Click OK.

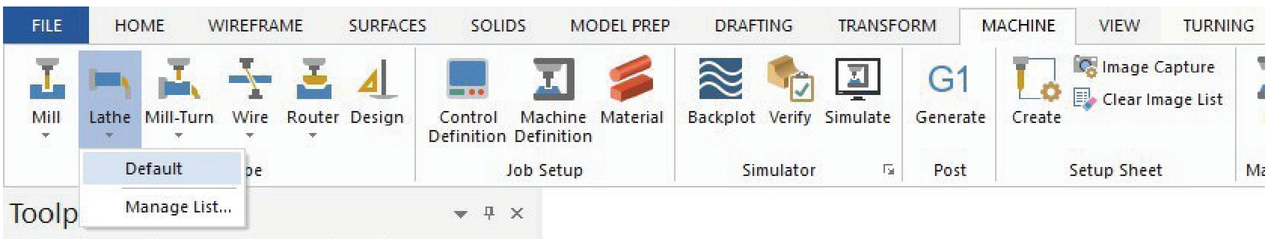
Fig 4

Loading a machine definition

Your part file needs at least one machine group before you can create toolpaths. Mastercam creates a machine group whenever you select a machine. To see how machine selection works, complete the following exercise.

1 From the Machine tab, choose the Lathe machine type, and select Default. (Fig 5)

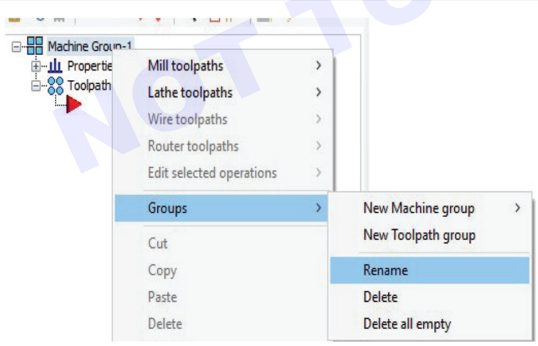
Fig 5



The menu lists all available machine definitions. This tutorial uses Mastercam's default Lathe machine. Normally, you would select the machine on which you plan to cut the part from the list displayed here.

2 **Rename the machine group**

Machine groups store complete jobs for a specific machine. For example, if some toolpaths will be cut on a lathe, and other toolpaths on a mill, you can simply create a second machine group. Each machine group can store its own job setup information and tools and use a different set of toolpath defaults. The toolpaths from each group will post to separate NC files.

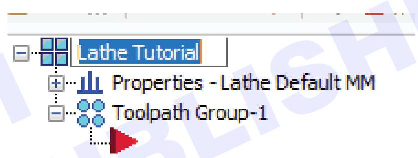


Mastercam lets you create as many machine groups as you need to organize your work.

- Right-click the machine group, and select Groups, Rename from the pop-up menu.
- Mastercam highlights the current group name.
- Type a new machine group name.
- The machine group name can be anything you want, but it is best to choose a name that describes the machine and its operations.

3 Select File, Save to save the file. (Fig 6)

Fig 6

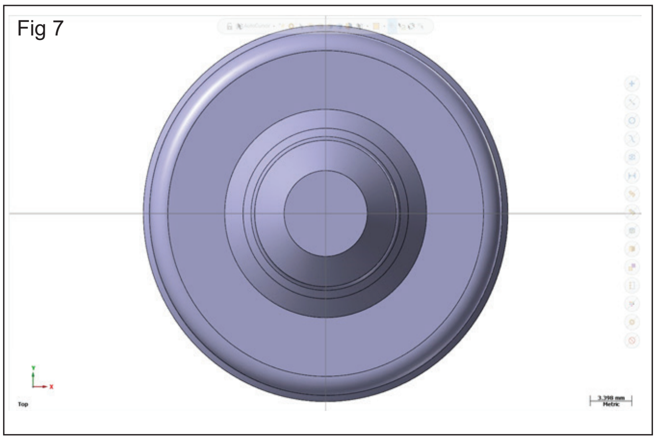


Orienting the part

Before you can machine the part, you need to move the geometry to its machine orientation. In this exercise, you use Mastercam's dynamic gnomon to re-align the part geometry to a typical horizontal lathe.

1 If necessary, press [F9] to display axes. (Fig 7)

Fig 7



2 Right-click and change the G View to Isometric to view the entire part from another angle. You may wish to unzoom to get a clearer view of the origin.

3 Select Dynamic from the Transform tab. (Fig 8)

Fig 8

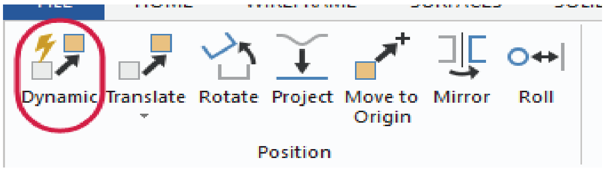
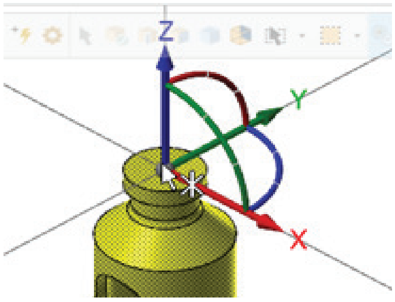


Fig 9



The Dynamic function panel displays.

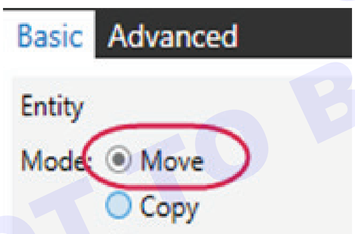
4 Select the part and press [Enter], or click the End Selection button.

5 Use the Auto Cursor to place the dynamic gnomon on the origin and click.

6 See Mastercam Help for more information on the Auto Cursor.

7 Confirm that Move is selected in the function panel. (Fig 10)

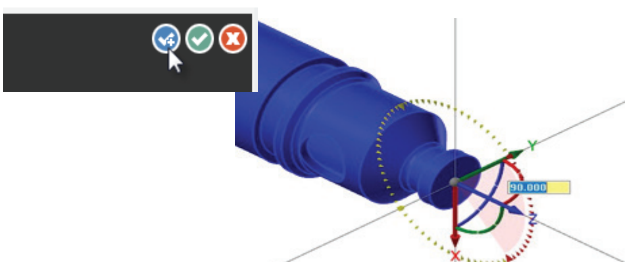
Fig 10



8 Select the center segment of the green arc to rotate the part 90 degrees so that its length lies on the X axis. Use your mouse to rotate the gnomon or enter 90 into the on-screen input box.

1 On the function panel, click the blue button (OK and Create New Operation). (Fig 11)

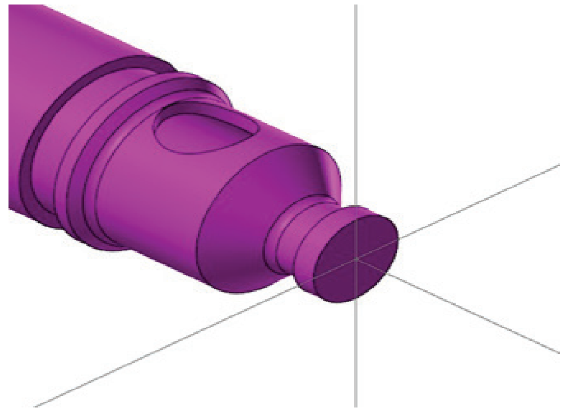
Fig 11



2 Select the part again and rotate it 90 degrees so that the hole aligns with the Z axis.

3 Use the same method you used previously to rotate the part from its original orientation. (Fig 12)

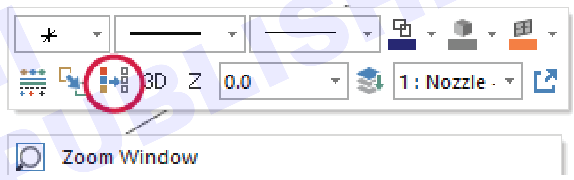
Fig 12



4 Click OK to accept the transformation.

5 Right-click and choose the Clear Colors command from the mini-toolbar to remove the result colors from the translated geometry. (Fig 13)

Fig 13



6 Save the file.

Using a lathe coordinate system

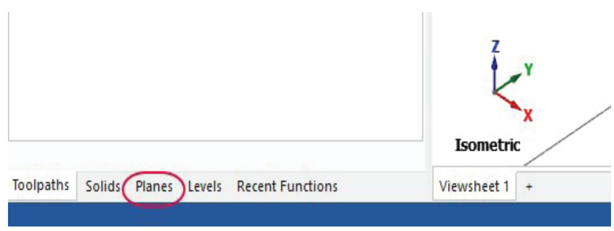
The coordinate system for a traditional lathe turning application differs from the coordinate system for milling applications. Instead of a 3D/XYZ space, a lathe coordinate system for typical turning applications is 2D, with the tool axis perpendicular to the spindle (Z axis) instead of parallel to it. In other words,

The D (diameter) axis in Lathe is equal to the Y axis in Mill.

The Z (length) axis in Lathe is equal to the X axis in Mill.

1 Select the Planes tab to open the Planes Manager. Fig 14)

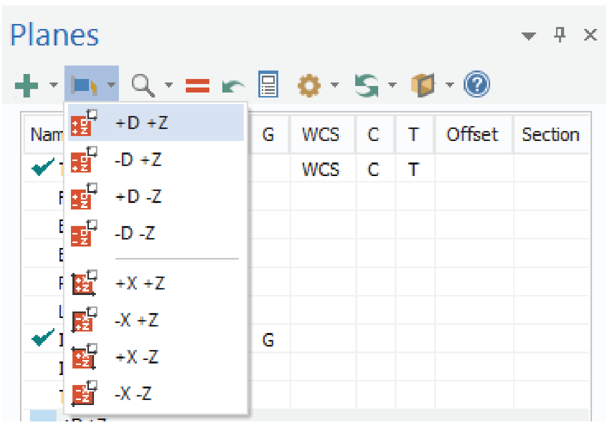
Fig 14



2 If a managers tab is not available, select it from the Managers group on the View tab.

Click Select lathe plane from the Planes Manager toolbar, and select +D +Z from the drop-down menu. (Fig 15)

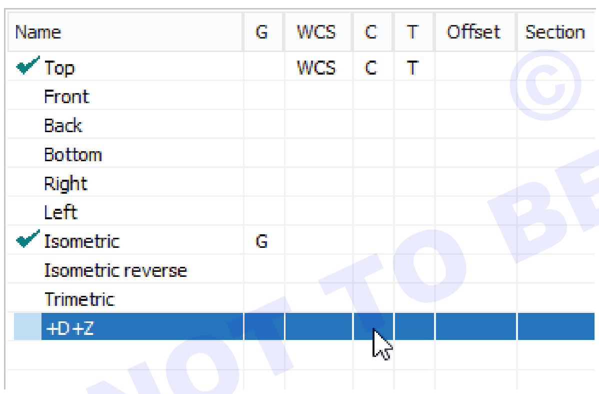
Fig 15



3 When working with a lathe, you can orient the construction plane to work in radius (X/Z) or diameter (D/Z) coordinates.

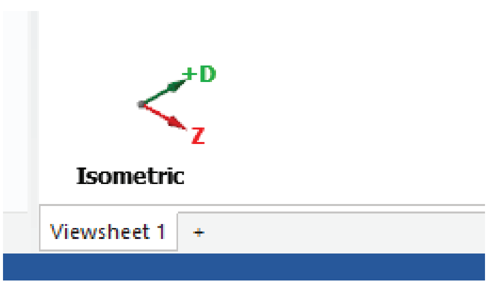
4 Click in the +D +Z row of the C column to set +D +Z as your construction plane. (Fig 16)

Fig 16



The Status bar indicator displays the new settings. The gnomon in the lower-left corner of the graphics window displays the current coordinate system. (Fig 17)

Fig 17



5 By default, the T plane is the same as the C plane. For 2D turning, you can leave the WCS as the Top plane

Creating 2D geometry

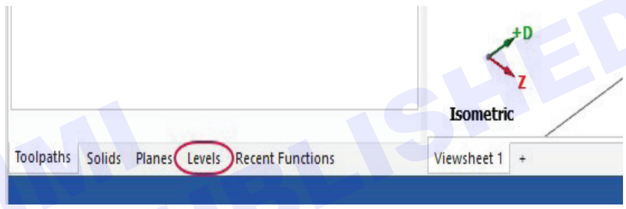
- Turn Profile generates a two-dimensional profile by creating wireframe geometry. We use this method in this tutorial and use the Levels Manager to organize the entities.
- Solid chaining eliminates the need to create wireframe geometry. It spins a profile around the selected axis "on the fly" when you select geometry for a toolpath. See Mastercam Help for more information about chaining solids.

Create a level for the new profile geometry

Use levels to organize part geometry. By organizing your files into levels, you can more easily control which areas of the drawing are visible at any time and what is selectable. In this section, you create a level specifically for the 2D profile geometry that you will create in the next section.

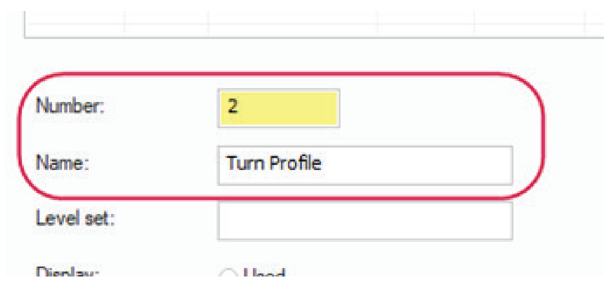
1 Click the Levels tab to open the Levels Manager. (Fig 18)

Fig 18



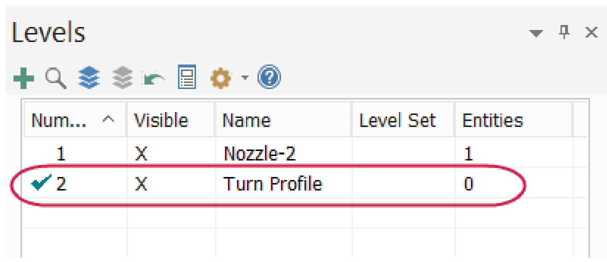
2 Create a new level by typing 2 and Turn Profile in the Number and Name fields. (Fig 19)

Fig 19



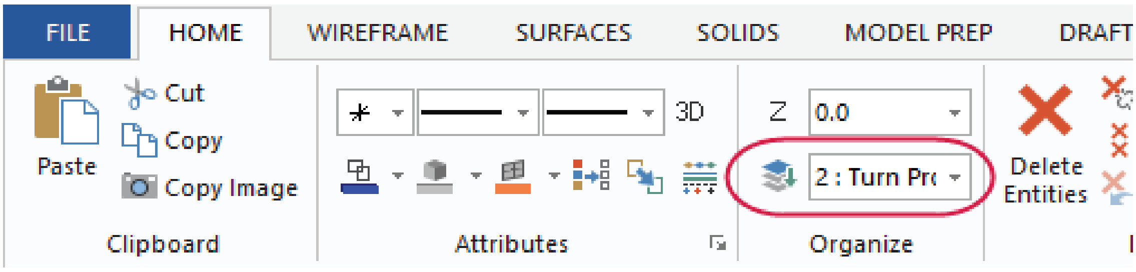
3 The new level displays in the table with zero entities. A blue check indicates that it is the current level. (Fig 20)

Fig 20



4 The current level displays on the Home tab.(Fig 21)

Fig 21

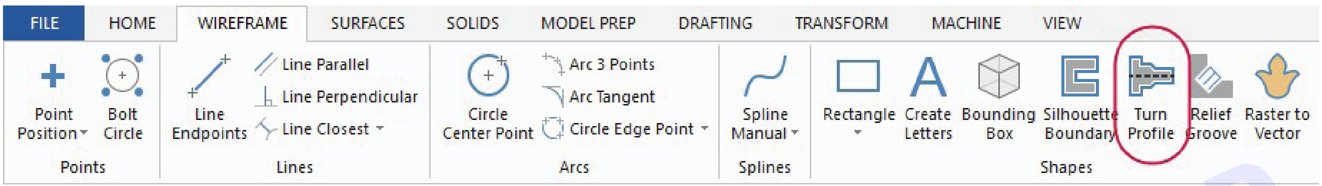


Create the profile

1 From the Wireframe tab, select Turn Profile. (Fig 22)

2 Select the part as prompted. Press [Enter], or click the End Selection button.

Fig 22



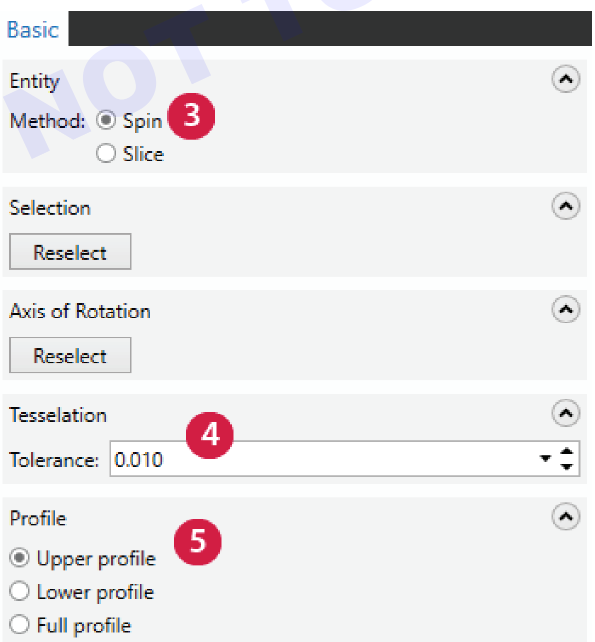
3 Use the Spin method. (Fig 23)

- The Spin method creates a profile by spinning the geometry about an axis.
- The Slice method produces a profile by creating a cross-section through the geometry on the XY plane.

4 Reduce the Tessellation tolerance to 0.01.

5 If necessary, choose the Upper profile option.

Fig 23



- Click OK.
- Return to the Levels Manager & hide Level 1 (click the X in the Visible column) to view the profile. (Fig 24)

Fig 24

The illustration above shows the part from the TOP view.

- Save the file

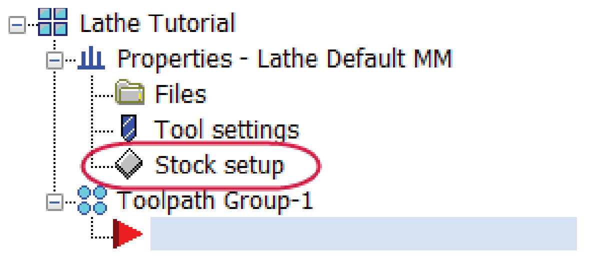
Setting up the stock in the left spindle

This exercise teaches you to create the stock for the part, set the stock margins and grip length, and place the stock at the correct Z position.

Creating the stock before creating the chuck jaws makes it easier to locate the stock relative to your part. Then, when you create the jaws, you can choose to automatically position them relative to the stock.

1 Open the Toolpaths Manager. Expand the Properties group and click Stock setup to open the Stock Setup tab of the Machine Group Properties dialog box. (Fig 25)

Fig 25

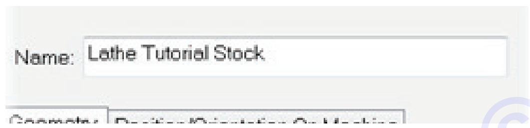


2 In the Stock section, select Left Spindle and click Properties. The Machine Component Manager - Stock dialog box opens. (Fig 26)

Fig 26

3 In the Name field, type Lathe Tutorial Stock to name the stock setup for the left spindle. (Fig 27)

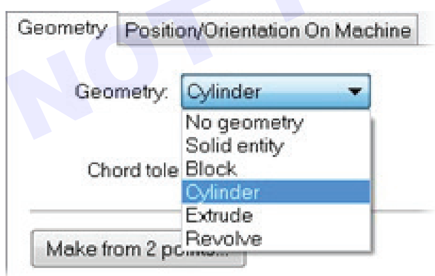
Fig 27



4 Choose Cylinder from the Geometry drop-down selections.

5 Cylinder lets you create 3D bar stock. This is suitable for most turning operations. (Fig 28)

Fig 28



6 Click the Make from 2 points button. (Fig 29)

Fig 29

7 In the graphics window, select the origin and the top of the part's back face. If you have not already done so, change the GView to TOP. (Fig 30&31)

Fig 30

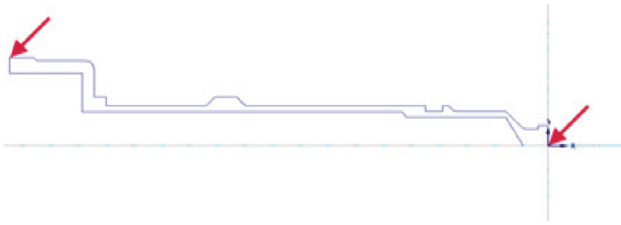
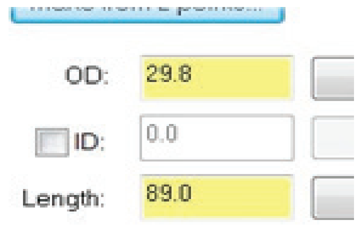


Fig 31



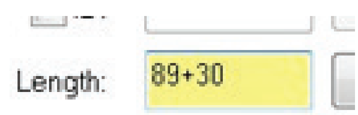
- Add an additional 30 mm to the Length field.

  The additional stock (about 30 percent) provides enough material for the left spindle chuck to hold the stock in place during machining. (Fig 32)

Fig 32

Fields that take number values have a built-in calculator that let you enter simple formulas directly into the field. (Fig 33)

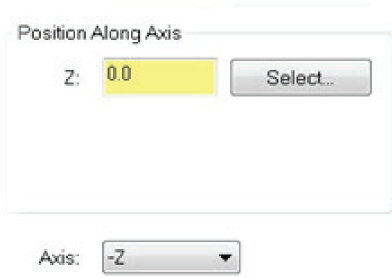
Fig 33



- Confirm that the Position Along Axis is set to 0.0, and the Axis field has -Z for the stock direction.

  These two values determine the location and orientation of the cylinder. The center of the part's face lies on the origin, and the part is positioned along -Z. ((Fig 34)

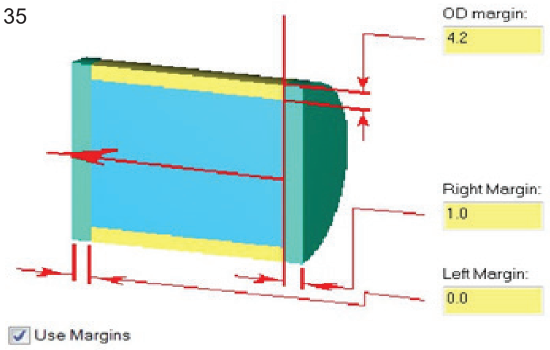
Fig 34



- Select the Use Margins option to activate the stock margin fields.

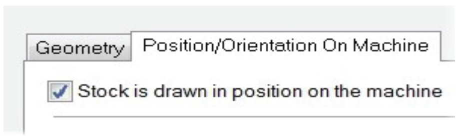
  Enter 4.2 mm in the OD margin field and 1.0 mm in the Right Margin field. Mastercam adds the margin to the stock boundary. (Fig 35)

Fig 35



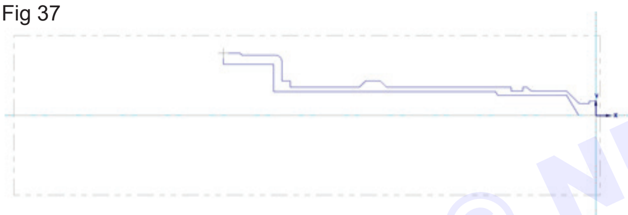
- Click the Position/Orientation On Machine tab, and make sure the option, Stock is drawn in position on the machine is selected. (Fig 36)

Fig 36



- Return to the Geometry tab and click the Preview Lathe Boundaries button to view your results. (Fig 37)

Fig 37



- Press [Enter] to return to the Machine Component Manager - Stock dialog box. Click OK to accept your lathe stock setup settings. The left spindle stock is defined. (Fig 38)

Fig 38

Defining the chuck jaws

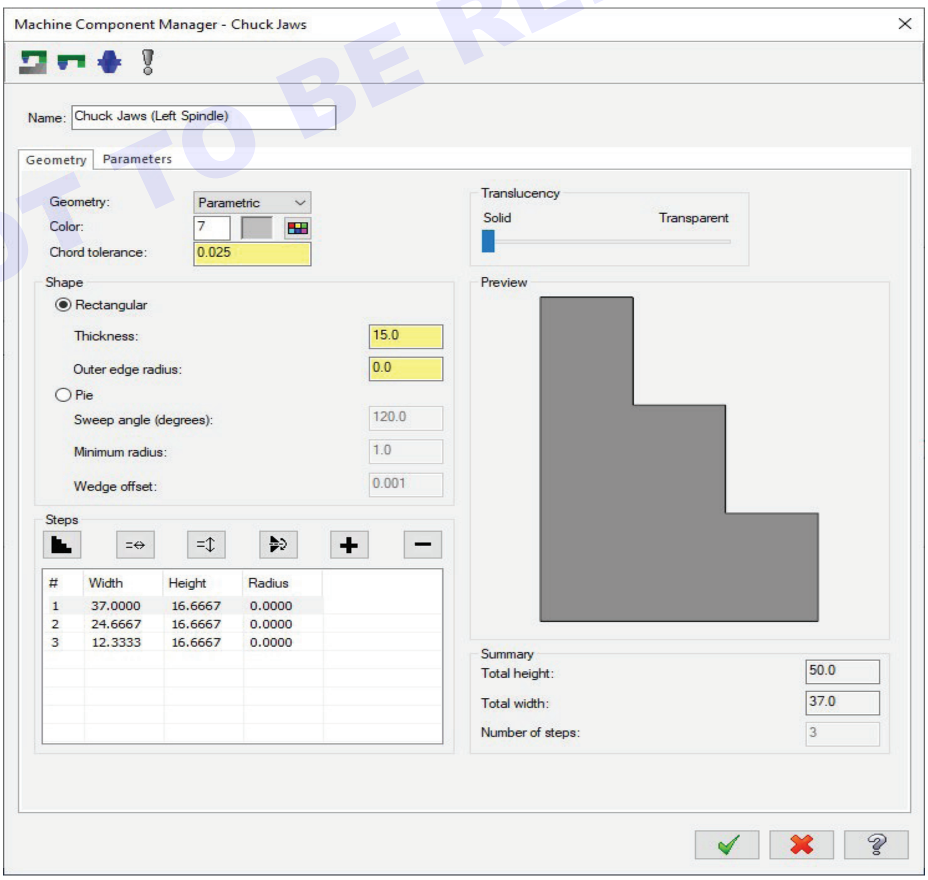
In this exercise you will define the position and grip length for the chuck jaws. You can only use the method described below after the stock has been set up.

1 In the Chuck Jaws section, select Left Spindle and click Properties. (Fig 39)

Fig 39

The Machine Component Manager - Chuck Jaws dialog box opens to the Geometry tab. (Fig 40)

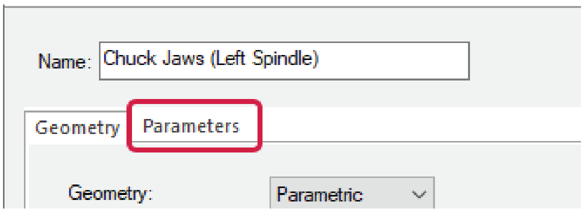
Fig 40



Confirm that your settings match those in the picture above. Although, we use a simple definition for this tutorial, Mastercam allows you to define your chuck jaws by selecting a solid model or a chained profile. See Mastercam Help for more information about enhanced support for modeling chucks and chuck jaws.

2 Click the Parameters tab. (Fig 41)

Fig 41

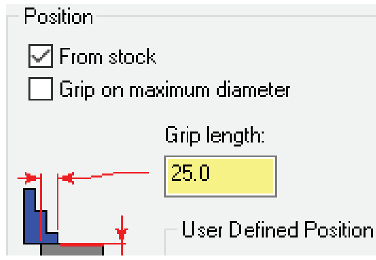


3 In the Position section:

- Select from stock.
- Enter 25 mm in the Grip length field.

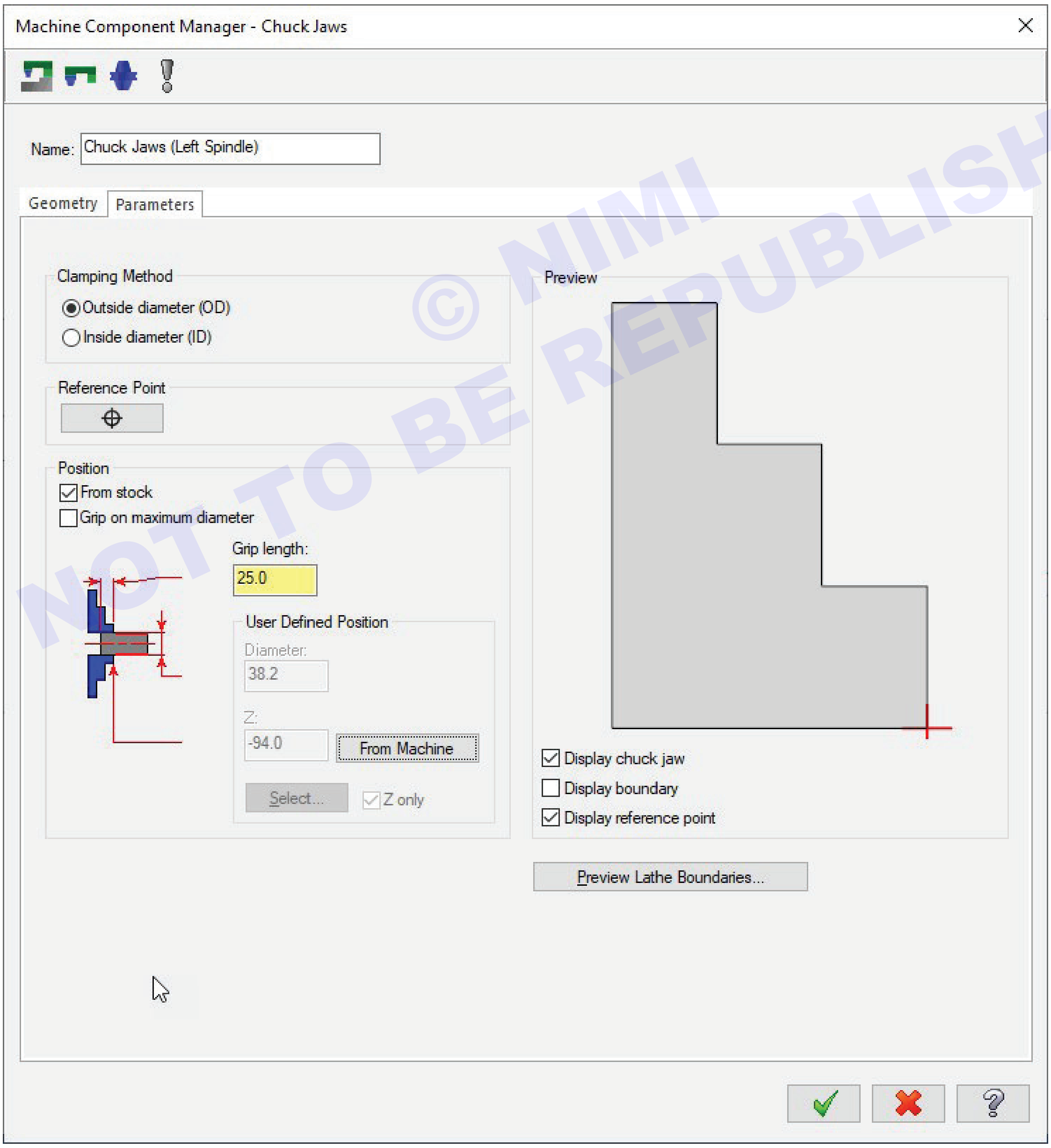
Mastercam uses these settings to calculate the position of the chuck jaws relative to the stock, and to determine how much stock is being held by the chuck jaws. Since you added 30 mm of extra stock in Exercise 6 on page 18, a grip length of 25 mm gives enough clearance to cut off the part. (Fig 42)

Fig 42



4 Confirm that the other values on the Parameters tab match those in the graphic below. (Fig 43)

Fig 43



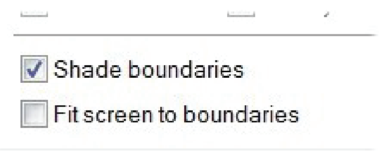
5 Click OK to accept these settings.

The Stock Setup tab displays with the left spindle's chuck jaws defined. (Fig 44)

Fig 44

6 Select the Shade boundaries option to more easily see the stock boundaries and chuck jaws you have created. (Fig 45)

Fig 45

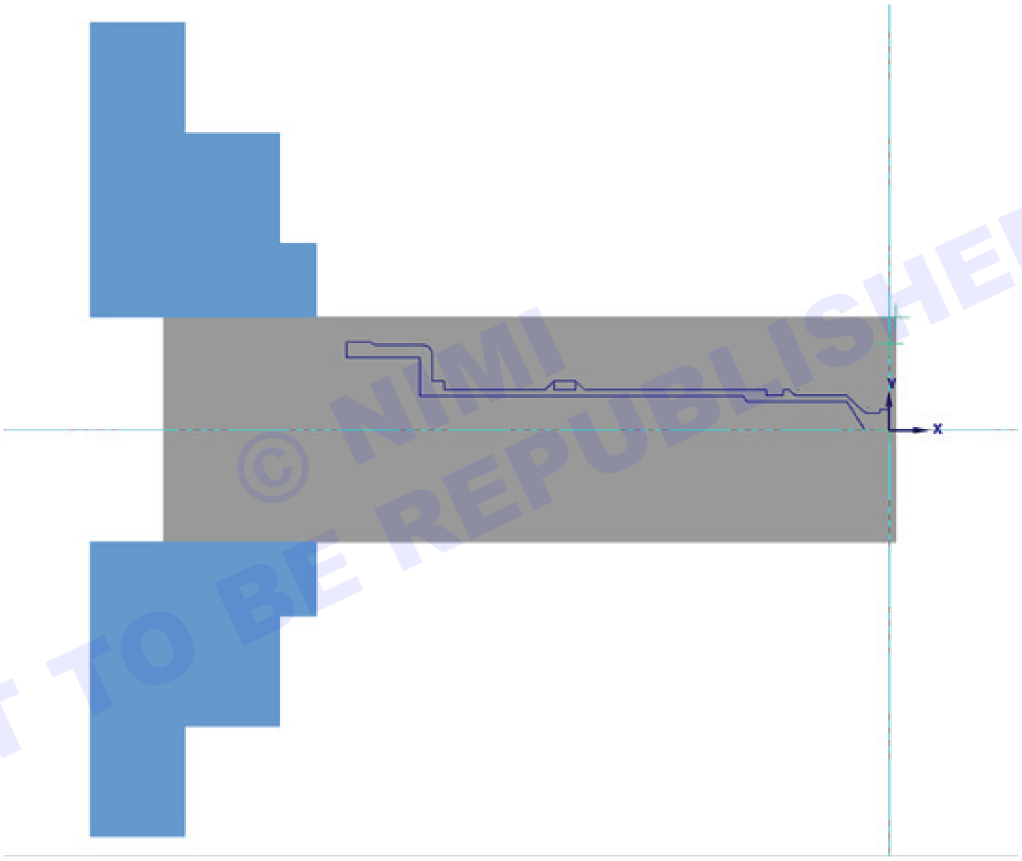


Select the Shade boundaries option to more easily see the stock boundaries and chuck jaws you have created.

7 Click OK to close the Machine Group Properties dialog box.

8 Save the file. (Fig 46)

Fig 46



You have prepared your part. Now you can create toolpaths.

Generation of complex machining part program with the help of CAM software

Objectives: At the end of this lesson you shall be able to
- **describe the tool path in lathe operation**
- **state about the use back plot option.**

Once you have set up your job, you can begin creating toolpaths. Several toolpaths are normally involved in machining a lathe part. In this lesson you create the toolpaths necessary to shape the outer diameter (OD) of the part. Then, you backplot the operations you created to check your work.

Facing, roughing and finishing

Facing the part: Face toolpaths prepare the face of the part for further machining. Once the face of the part is clean, you can use it to set tools or determine tool offsets.

You do not need to chain geometry to create a face toolpath. Mastercam can create the toolpath entirely from parameters you enter in the Lathe Face dialog box.

1 Choose Face from the Lathe Turning tab. (Fig 1)

2 From the Toolpath parameters tab, select the default OD roughing tool: T0101 R0.8 OD ROUGH RIGHT - 80 DEG. (Fig 2)

Fig 1

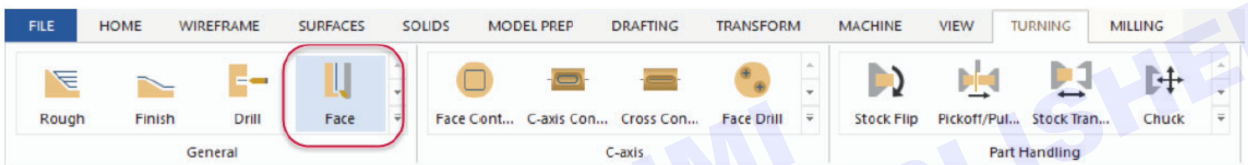
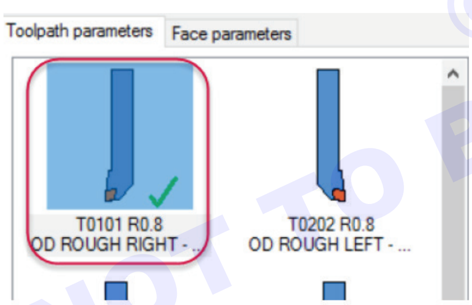


Fig 2



Your tools may have different tool numbers.

3 Keep all other parameters on this page at their default values.

Enter the cutting values

1 Click the Face parameters tab. (Fig 3)

2 Confirm that Use stock is selected.

Use stock is available only if you have defined the stock boundary in Stock Setup. (Fig 4)

3 If necessary, enter 0 in the text box, or click Finish Z and select the origin from the graphics window to place the finished face at the origin.

4 Keep all other parameters on this page at their default values.

5 Click OK to create the toolpath.

Roughing a part

Use rough toolpaths to quickly remove large amounts of stock in preparation for a finish pass. Roughing passes are typically straight cuts parallel to the Z-axis.

Mastercam includes several types of roughing toolpaths:

- Standard rough toolpaths, which let you access all of Mastercam's roughing options.
- Canned rough toolpaths, which use your machine tool's canned cycles to create the most efficient code (however, these do not offer as many options as the standard rough toolpaths).
- Canned pattern repeat toolpaths, which create roughing passes in the shape of the part contour, rather than cutting parallel to the Z-axis.
- Dynamic rough toolpaths, which remain engaged in the material more effectively, and use more of the surface of your insert, extending the tool life and increasing the cutting speed.
- Contour rough toolpaths, which are useful for parts where the initial stock shape is similar to the final part shape, such as using a casting for stock.

Fig 3

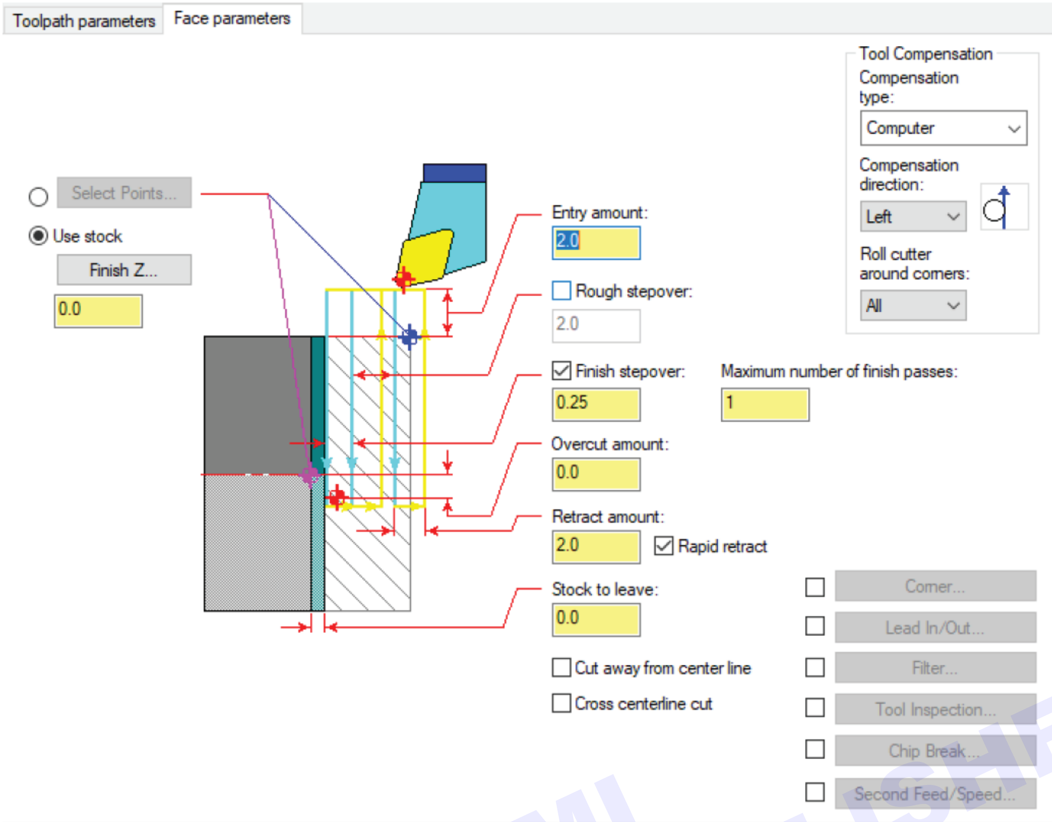
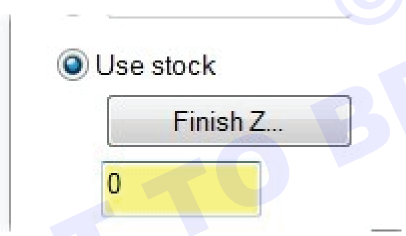
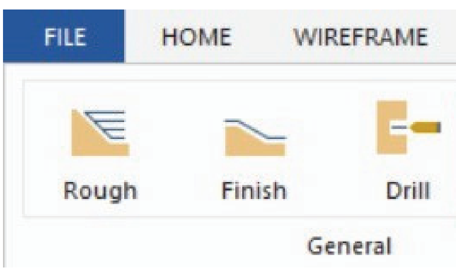


Fig 4



1 Choose Rough from the Lathe Turning tab.(Fig 5)

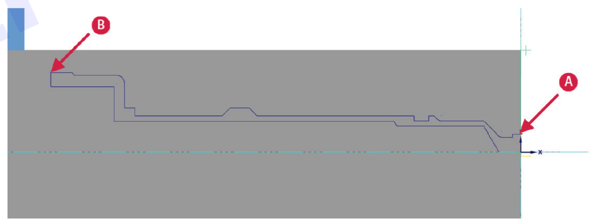
Fig 5



2 The chaining dialog box displays, and you are prompted to select an entry point. Mastercam Lathe defaults to partial chaining.

3 Click the geometry at the first entity (a) and then, at the last entity (b) to create a partial chain. (Fig 6)

Fig 6



4 Click OK in the Chaining dialog box to accept the chain.

5 The lathe rough dialog box opens.

6 Use the Toolpath parameters tab to select a tool, set feeds and speeds, and modify other general toolpath parameters. This tab is similar for most Lathe toolpaths. From the Toolpath parameters tab, select the default OD roughing tool.

7 Click the rough parameters tab. (Fig 7)

8 Enter the cutting values like depth of cut, stock to leave X and Z for finishing etc.

9 Click OK to create toolpath.

10 Click the Parameters folder in toolpath manager to reopen the Lathe Rough dialog box and modify the operation. (Fig 8)

Fig 7

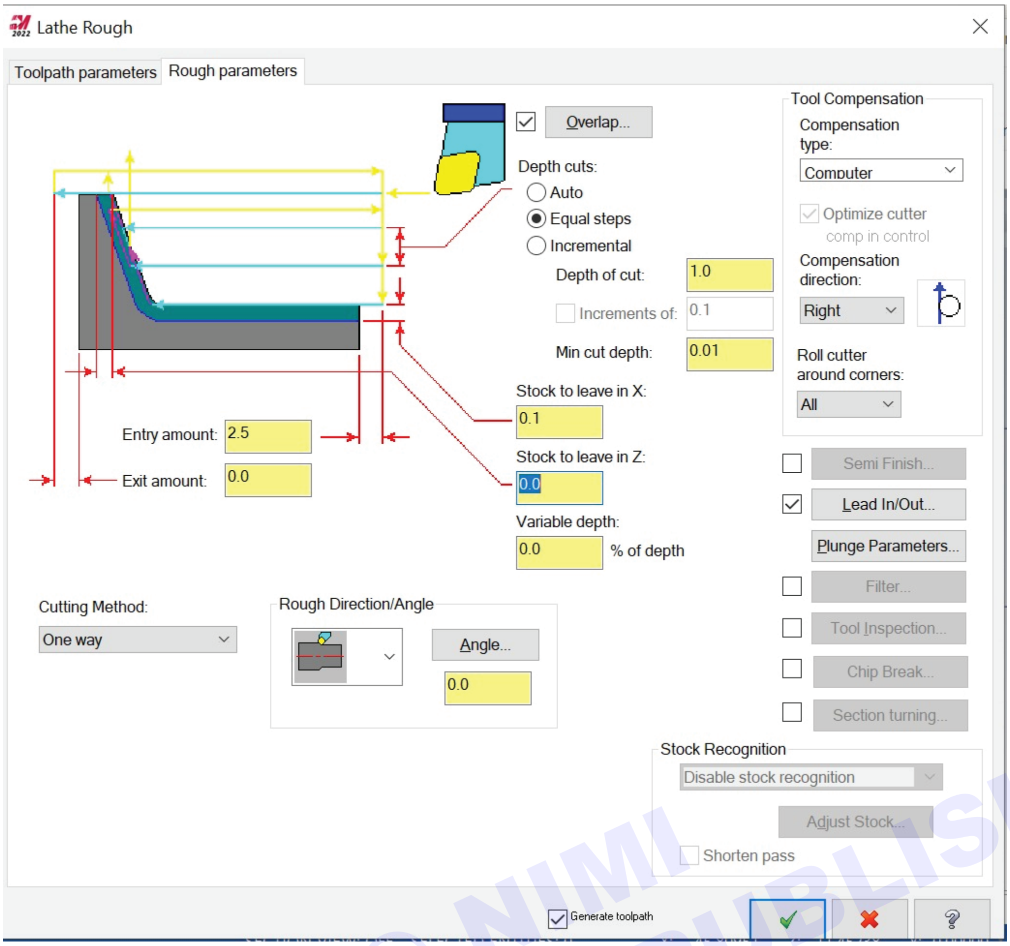
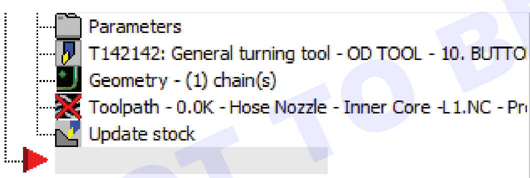


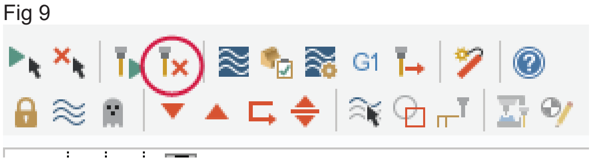
Fig 8



After modifying the parameters click OK.

11 Regenerate the toolpath and save your part.

Fig 9



Finishing a part: Use finish toolpaths to have the tool follow the contour of chained geometry. Typically, a finish toolpath follows a roughing toolpath.

1 Choose Finish from the Lathe Turning tab. (Fig 10)

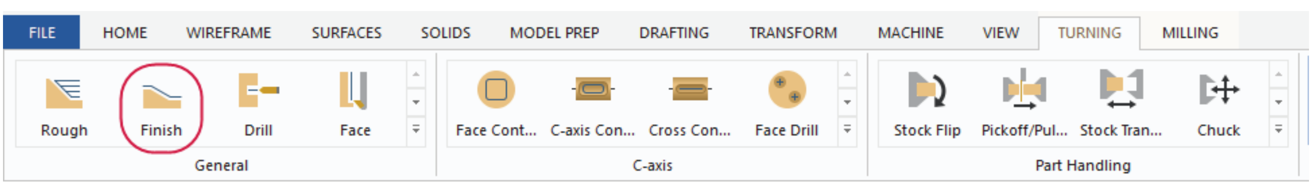
The Chaining dialog box displays, and you are prompted to select a point or chain a contour.

2 Select the same geometry that you selected for the Roughing toolpath.

3 Click OK in the Chaining dialog box to accept the chain. The Lathe Finish dialog box opens.

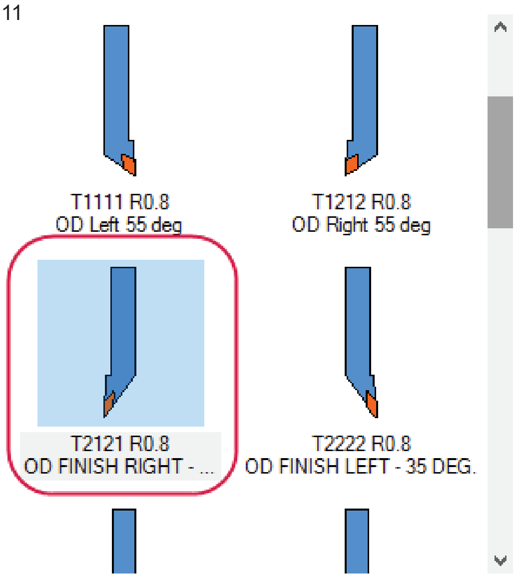
Entering the toolpath parameters for any finish toolpath uses the same workflow as other lathe toolpaths: first you select the tool and tool options, and then you enter the cutting values.

Fig 10



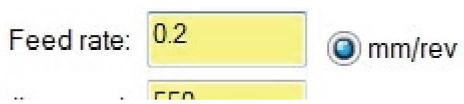
4 Select the finishing tool. (Fig 11)

Fig 11



5 Decrease the feed rate to 0.2. (Fig 12)

Fig 12

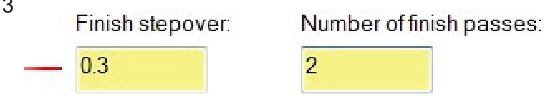


6 Keep all other parameters on this page at their default values.

7 Click the Finish parameters tab.

8 Change the Finish step over and the Number of finish passes. (Fig 13)

Fig 13



9 Leave all other parameters on this page at their default values.

10 Click OK.

Mastercam creates a finish operation on top of the previous roughing operation.

Back plot: Back plotting shows the path the tools take to cut your part and lets you spot errors in the program before you send it to the machine.

1 Click the Select all operations button in the Toolpaths Manager. (Fig 14)

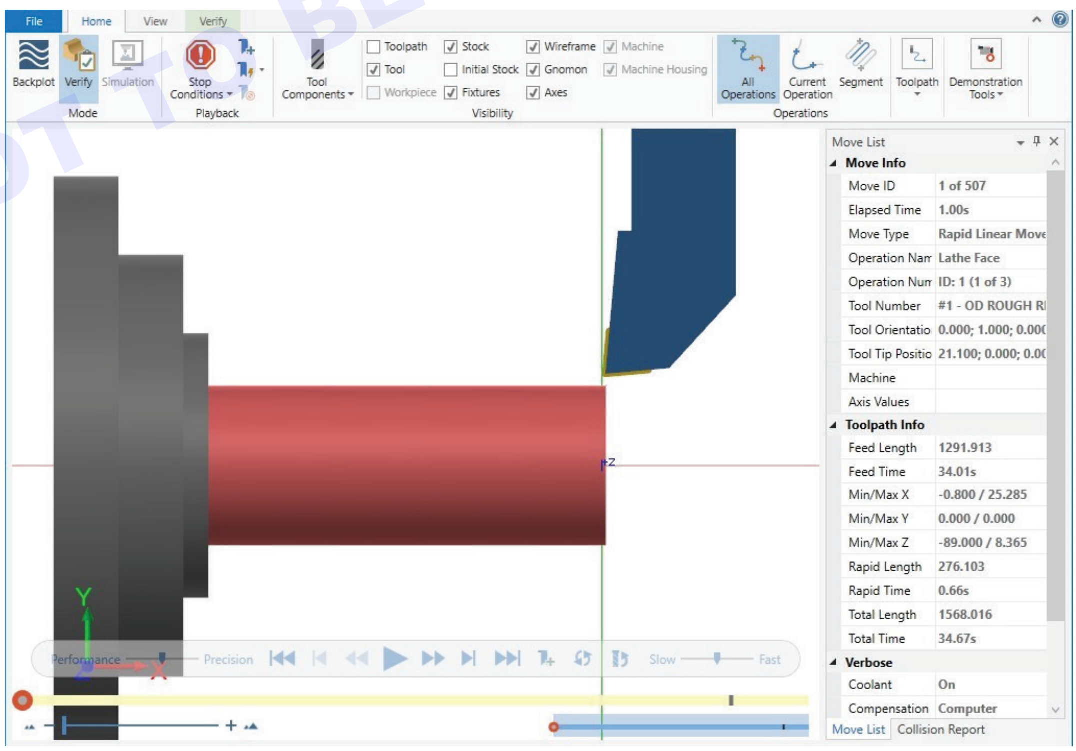
Fig 14

2 Click the Verify selected operations button. (Fig 15)

Fig 15

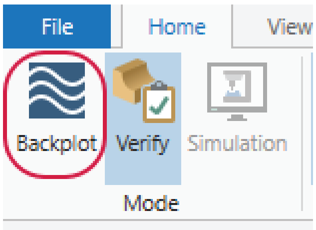
Mastercam Simulator opens a view of the part in Verify mode in a separate window. (Fig 16)

Fig 16



3 Click Backplot on the Home tab. (Fig 17)

Fig 17



The Simulator window changes to Backplot mode. (Fig 18)

4 Select the Initial Stock option in the ribbon bar's Visibility group to see the stock before machining. (Fig 19)

5 Select the Initial Stock option again to view the part profile against a translucent display of the initial stock. (Fig 20)

Fig 18

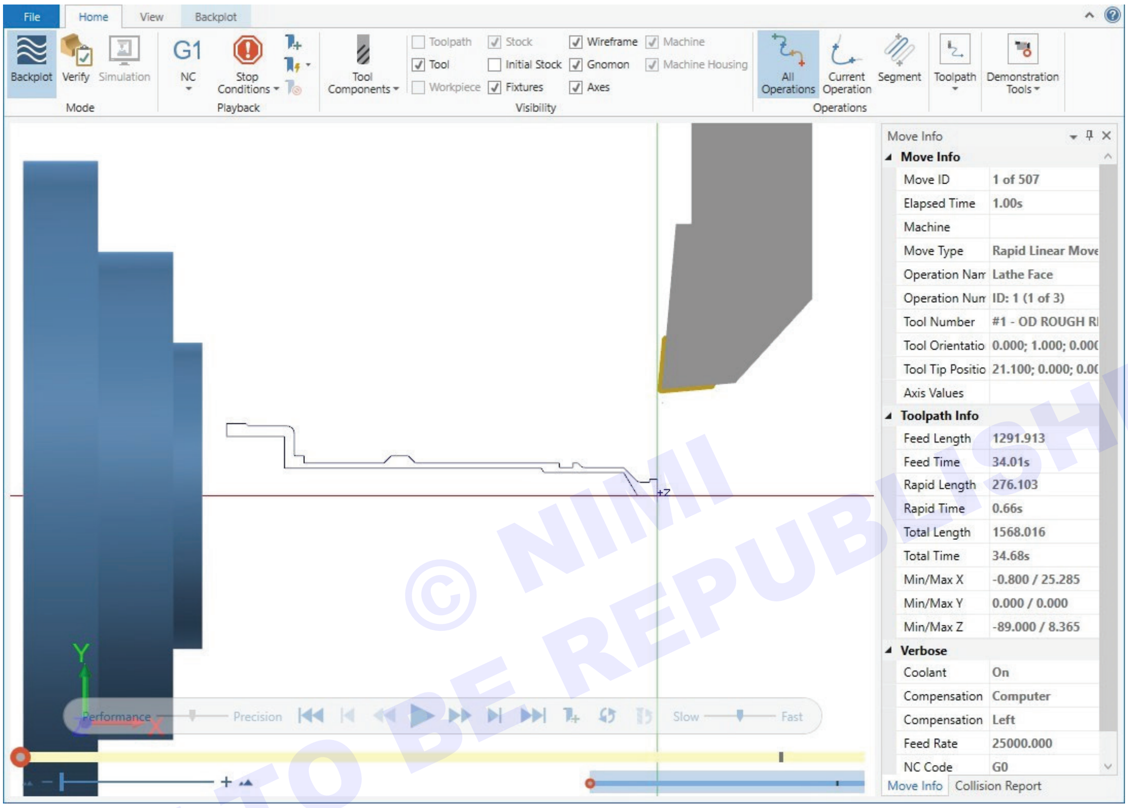


Fig 19

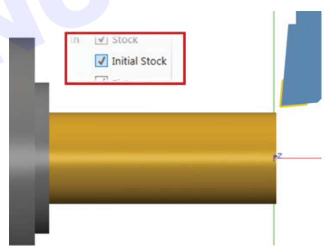
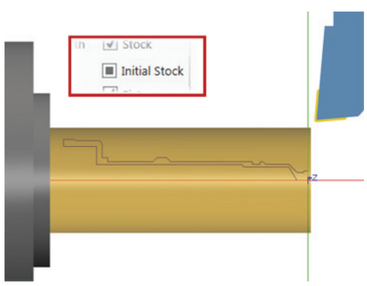


Fig 20



6 For a better view of the operations in the Mastercam Simulator, use the Page Up key to zoom into the part and [Ctrl + Arrows] to pan the view.

The views in Mastercam and Mastercam Simulator windows can be synchronized. Turn this option on or off from the Viewtab of Mastercam Simulator.

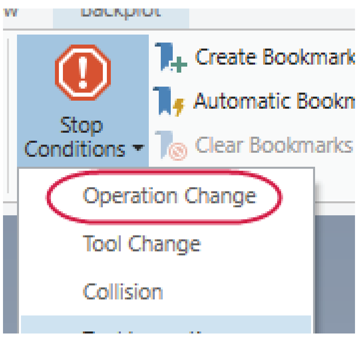
7 Click the Play button at the bottom of the Simulator screen. (Fig 21)

Fig 21

Simulator backplots all three toolpaths with information about the current toolpath motion displayed in the Move List on the right-side of the screen.

8 To pause the simulator at the end of each operation, select Operation Change from the Stop Conditions drop- down. (Fig 22)

Fig 22



9 Use the Play [R], Step Forward [S], and Step Backward [B] keys to view the operations again at your own pace.

10 Minimize the Mastercam Simulator, or move it to another monitor.

Program generation

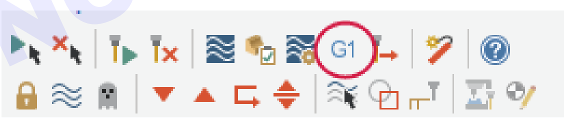
Program generation in CAM (Computer-Aided Manufacturing) refers to the process of creating a detailed set of instructions (usually in the form of G-code) that a CNC (Computer Numerical Control) machine can follow to manufacture a part. This process involves converting a CAD (Computer-Aided Design) model into machine-readable commands that dictate the movements and operations of the CNC machine.

Posting the operations in the machine group

1 If necessary, select all operations in the Toolpaths Manager.

2 Click the Post selected operations button in the Toolpaths Manager. (Fig 23)

Fig 23

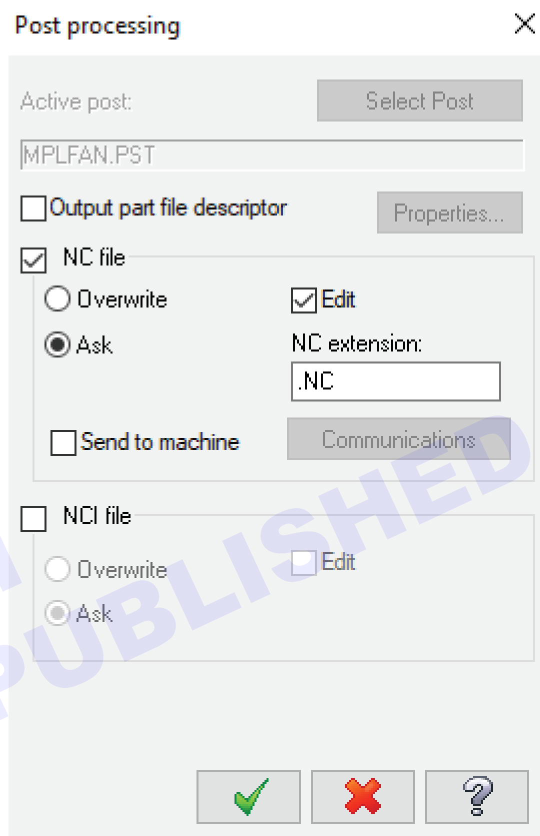


If you have not selected all the operations in a machine group, Mastercam will ask if you want to post all the operations.

The Post processing dialog box displays. Mastercam uses these settings to handle the files that are generated when posting.

3 Click OK. (Fig 24)

Fig 24



The Save As dialog box displays.

4 Rename the file, or click Save to accept the default NC file name.

Mastercam posts the file, and it is opened in your default file editor. Examine the NC file before sending it to the machine.

Tool path optimization and cycle time calculation

Objectives: At the end of this lesson you shall be able to
- **explain the need of toolpath optimization**
- **define cycle time**
- **explain how to calculate cycle time for CNC turning.**

Toolpath Optimization

Toolpath optimization focuses on refining the path that the cutting tool follows during the turning process to enhance efficiency, reduce machining time, improve surface quality, and extend tool life. In CNC turning, this involves minimizing unnecessary movements of the tool and maximizing material removal rates while maintaining optimal cutting conditions. To do this, we use techniques like adjusting the tool's path based on the shape of the material, ensuring the tool always cuts the right amount of material to avoid damage, and simulating the whole process to catch any mistakes before they happen. By optimizing the toolpath, we can save time, reduce costs, and produce higher-quality parts, making the whole process more effective and easier to manage.

For facilitating high speed machining, following are the requirements for a CAM system:

- It should maximize program processing speed.
- It should minimize feed rate losses.
- It should maintain a constant chip load.
- It is essential to note that a number of CAM programs approach the problem, assuming that various specialized tool path strategies may be used to manufacture the part as professionally as possible.
- There are different types of tool path strategies that are used to optimize CNC machining services. Here are a few of them:

1 **Constant stepover:** A tool path wherein the tool follows the shape of the pocket using parallel paths that are separated by a constant stepover.

Pros

- Produces a very consistent and regular looking finish.
- This is the simplest tool path strategy. It is a default approach, and may not even be given a name in the CAD program.

2 Constant "Z" Machining

This strategy is particularly used for finishing, where the tool path tracks at a constant Z around the profile that is being machined. It is normally used for steep walls, while in other situations, another strategy is applied. The areas that are not very steep are avoided by limiting the path to contact angles that range from 30 to 90 degrees.

Pros

- Produces a pretty finish because the scallops are all the same height.

3 Plunge roughing

A roughing technique where cutting occurs through the motion of only Z-axis much like plunging a drill repeatedly into the work piece. It takes advantage of the fact that most machines are rigid in the Z-axis, and can take a higher feed rate and/or a larger cutter when used in this way. Plunging works best if the tool path is orchestrated to ensure climb milling.

Pros

- May result in higher performance when roughing. By implementing dynamic tool path strategies it will not only increase the tool life, surface finish, and spindle life, but also the total cost efficiency and cycle time. You can implement any of the strategies explained above based on your requirement.

Cycle time

Cycle time calculation in refers to determining the total time required to complete an operation from start to finish, including all the steps involved in machining a part. It encompasses various components such as the time taken for loading and unloading the workpiece, tool changes, actual cutting operations, idle times during rapid movements, and setup time required to configure the machine for the specific job. The core of cycle time calculation lies in estimating the machining time, which depends on parameters like cutting speed, feed rate, depth of cut, workpiece diameter, and length of the cut. Accurate cycle time calculation is crucial for effective production planning, cost estimation, and improving overall efficiency in manufacturing. By understanding and optimizing these factors, manufacturers can reduce waste, enhance productivity, and better manage resources in CNC turning operations.

Factors affecting cycle time

- **Machine capabilities:** Faster machines can reduce machining and idle times.
- **Tooling:** High-performance tools may reduce the number of passes and increase cutting speeds.
- **Workpiece material:** Harder materials may require slower speeds and feeds, increasing cycle time.

- Process Parameters: Optimizing cutting parameters like feed rate, cutting speed, and depth of cut can significantly reduce cycle time.

In CNC turning, calculating the cycle time involves understanding the length of the job, including all movements of the tool necessary to complete the work. The formula for determining the length includes the job length, the additional travel needed beyond the job dimensions (tool over travel), the number of passes the tool makes, and the initial approach of the tool to the workpiece.

Simple Formula for Cycle Time Calculation:

1 Calculate Total Length (L)

Total Length = Job Length + (Tool Over Travel x Number of Passes) + Tool Approach Length

2 Determine Average RPM (N):

RPM = (1000 x Cutting Speed) / (3.14 x Average Diameter of the workpiece)

Using these calculations helps in determining how long it will take for the turning operation to complete, which is crucial for scheduling and productivity analysis.

Example calculation: Suppose we have a job where the tool needs to travel a total job length of 200 mm, the tool over travels by 5 mm for each of three passes, and the tool approach length is 10 mm. The average diameter of the workpiece is 50 mm, and we aim for a cutting speed of 100 meters per minute.

1 Total length calculation

Total Length = 200 mm + (5 mm x 3) + 10 mm = 200 mm + 15 mm + 10 mm = 225 mm

2 RPM calculation

RPM = (1000 x 100) / (3.14 x 50) = 100000 / 157 = 637 RPM approximately

Feed per Revolution (f)

Feed = 3 mm per revolution

Using the above information, the cycle time can be calculated by dividing the total length by the product of RPM and feed per revolution.

$$\text{Cycle} = \frac{\text{Total Length}}{\text{RPM x Feed per revolution}} = \frac{\text{225 mm}}{\text{637 rev/min x 3 mm/rev}}$$

$$= \frac{\text{225 mm}}{\text{1911 mm/min}} = 0.118 \text{ minutes}$$

Machine offset and cutter tool nose radius compensation

Objectives: At the end of this lesson you shall be able to
- **describe about machine offset importance and concept**
- **state tool nose radius compensation.**

Machine offset and cutter tool nose radius compensation are essential concepts in CNC machining to ensure precision and accuracy.

Machine offset

Machine offset refers to the adjustments made to the machine's coordinates to account for the position of the tool relative to the workpiece. This ensures that the tool moves to the correct location during machining. There are different types of offsets:

- **Work offset**

Work offset is a crucial concept in CNC machining that helps in accurately positioning the workpiece relative to the machine's coordinate system. It ensures that the machine knows the exact location of the part to be machined, allowing for precise and repeatable operations.

Explanation

Work offsets are used to set the zero point (origin) of the workpiece in the machine's coordinate system. This is essential for ensuring that all machining operations are performed at the correct locations on the workpiece. The most common work offsets are G54 to G59, but many machines support additional offsets.

Steps to set work offset

a Zeroing the Machine: Establish the machine's home position, which serves as the reference point for all movements.

b Locating the Part Zero: Determine the part zero (origin) on the workpiece. This involves measuring the shift in X, Y, and Z coordinates from the machine zero to the part zero.

c Entering the Offset Values: Input the measured offsets into the CNC machine's control system. This tells the machine where the part zero is located.

- **Tool offset:** Compensates for the length and diameter of the tool.

Tool offset is a critical concept in CNC machining that ensures the machine accurately accounts for the tool's dimensions, such as length and diameter, during the machining process. This helps in achieving precise and consistent results.

Explanation

Tool offsets are adjustments made to the machine's coordinates to compensate for the tool's physical dimensions. There are two main types of tool offsets:

- **Tool length offset (H Offset):** Compensates for the length of the tool.
- **Tool diameter offset (D Offset):** Compensates for the diameter of the tool.

Tool length offset (H Offset)

This offset adjusts for the difference in length between the spindle and the cutting tool. It ensures that the tool tip is positioned correctly relative to the workpiece. The G-code used for tool length compensation is G43 (to apply the offset) and G49 (to cancel the offset).

Tool diameter offset (D Offset)

This offset compensates for the tool's diameter, ensuring that the tool path is adjusted so the edge of the cutter follows the programmed path. The G-codes used for diameter compensation are G41 (left compensation) and G42 (right compensation), with G40 used to cancel the compensation.

- **Wear offset**

Wear offset is used to make fine adjustments to the tool's position to compensate for tool wear and ensure that the machined parts remain within the desired tolerance. This is crucial for maintaining precision and quality throughout the production run.

Explanation

Wear offsets are typically small adjustments made to the tool's position in the X, Y, or Z axes. These adjustments account for the gradual wear of the cutting tool, which can affect the dimensions of the machined part. By using wear offsets, operators can correct for these changes without reprogramming the entire tool path.

How Wear Offsets Work

a **Initial Setup:** During the initial setup, the geometry offset is used to set the tool's position relative to the workpiece.

b **Wear Compensation:** As the tool wears down, the wear offset is adjusted to compensate for the change in tool dimensions. This ensures that the tool continues to cut accurately.

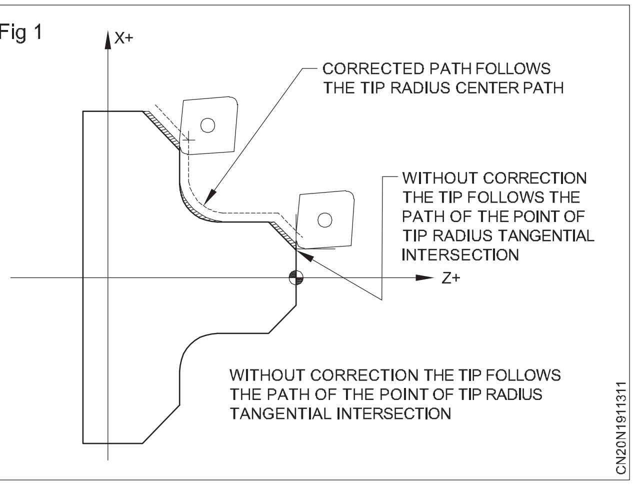
Cutter Tool Nose Radius Compensation

IT (TNRC) is used to account for the radius of the tool's cutting edge, especially in CNC turning operations. This compensation ensures that the tool path is adjusted to produce the correct part dimensions. The key G-codes used are

- **G41:** Compensates for the tool radius to the left of the cutting path.
- **G42:** Compensates for the tool radius to the right of the cutting path.
- **G40:** Cancels the compensation. (Fig 1)

Using TNRC is crucial for achieving accurate profiles, especially when dealing with chamfers and radii. It helps in avoiding undercuts or overcuts by adjusting the tool path based on the tool's nose radius.

Fig 1



Concept of interpolation and canned cycles

Objectives: At the end of this lesson you shall be able to
- **describe importance and concept interpolation in CNC machine**
- **state importance and concept of canned cycles.**

Interpolation in CNC machining

Interpolation is a fundamental concept in CNC (Computer Numerical Control) machining. It involves calculating intermediate points between known positions to create smooth and accurate motion along a path. This allows the CNC machine to move seamlessly from one point to another, which is crucial for creating complex shapes, curves, and contours. There are different types of interpolation:

- **Linear interpolation**

Linear interpolation is a fundamental concept in CNC (Computer Numerical Control) machining. It involves moving the cutting tool in a straight line between two points. This type of interpolation is essential for creating precise linear cuts and is widely used in various machining operations.

How linear interpolation works

When a CNC program specifies the start and end coordinates for a movement, the machine's control system calculates the intermediate points along a straight path. This ensures smooth and accurate motion from one point to another. The G-code used for linear interpolation is G01.

Key features

a **Straight line movement:** The tool moves in a straight line between two specified points.

b **Feed rate control:** The speed of the tool movement is controlled by the feed rate, specified with the F command.

c **Modal command:** Once activated, the G01 command remains in effect until another movement command is issued. eg:

G01 X10.0 Y20.0 Z-5.0 F100

- **Circular interpolation**

Circular interpolation in CNC machining

Circular interpolation is a technique used in CNC (Computer Numerical Control) machining to move the cutting tool along a circular arc. This is essential for creating curved surfaces and circular features on a workpiece. The G-codes used for circular interpolation are G02 (clockwise) and G03 (counterclockwise).When a CNC program specifies a circular movement, the machine's control system calculates the intermediate points along the arc to ensure smooth and accurate motion. The movement can be defined using either the radius of the arc or the coordinates of the arc's center.

Key features

Clockwise (G02) and Counterclockwise (G03): Specifies the direction of the circular movement.

a **Feed rate control:** The speed of the tool movement is controlled by the feed rate, specified with the F command.

b **Radius or center coordinates:** The arc can be defined using the radius ® or the center coordinates (I, J, K). e:g

c **G01 X10.0 Y10.0 F100 ; Move to starting point**

G02 X20.0 Y20.0 R10.0 F50 ; Clockwise arc to (20,20) with radius 10

G03 X30.0 Y30.0 I5.0 J5.0 F50 ; Counterclockwise arc to (30,30) with center offset (5,5)

The tool moves to the starting point (X10.0, Y10.0).

It then moves along a clockwise arc to (X20.0, Y20.0) with a radius of 10 units.

Finally, it moves along a counterclockwise arc to (X30.0, Y30.0) with the center of the arc offset by (I5.0, J5.0) from the starting point.

- **Spline interpolation**

Spline interpolation is an advanced technique used in CNC (Computer Numerical Control) machining to create smooth and precise curves. Unlike linear and circular interpolation, which are limited to straight lines and circular arcs, spline interpolation can handle complex, free-form shapes. This makes it particularly useful in industries requiring high precision and intricate geometries, such as aerospace and automotive manufacturing.

How spline interpolation works

Spline interpolation involves fitting a smooth curve through a series of points (known as control points) using mathematical functions called splines. The most common types of splines used in CNC machining are B-splines and Bezier splines. These splines ensure that the curve passes smoothly through or near the control points, providing a continuous and smooth tool path.

Key features

a **Smooth curves:** Spline interpolation creates smooth and continuous curves, avoiding the sharp transitions that can occur with linear interpolation.

b **High precision:** It allows for precise control over the shape of the curve, making it ideal for complex geometries.

c **Flexibility:** Spline interpolation can be used to create a wide variety of shapes, from simple arcs to intricate free-form surfaces.

Example

Here's a simplified example of how spline interpolation might be used in a CNC program.

Interpolation ensures that the cutting tool follows the desired trajectory with high precision, maintaining consistent speed and accuracy throughout the machining process.

```
G01 X0 Y0 F100      ; Move to starting point
G05 P1 X10 Y10      ; Define control point for spline
G05 P1 X20 Y15      ; Define next control point
G05 P1 X30 Y10      ; Define next control point
G05 P1 X40 Y0       ; Define end point of spline
```

In this example, the G05 command is used to define the control points for the spline. The CNC machine will interpolate a smooth curve through these points. Spline interpolation is essential for achieving high-quality finishes and precise geometries in CNC machining. It allows for the creation of complex shapes that would be difficult or impossible to achieve with linear or circular interpolation.

Canned cycles

Canned cycles are pre-programmed sequences of machine operations used in CNC (Computer Numerical Control) machining to automate repetitive tasks. These cycles simplify the programming process by condensing multiple lines of G-code into a single command, making them efficient for tasks like drilling, boring, and threading.

Key features

- **Automation:** Canned cycles automate repetitive machining operations, reducing the need for manual programming.
- **Efficiency:** They save time and reduce the risk of errors by providing a concise way to program complex operations.
- **Modal commands:** Canned cycles remain active until cancelled, allowing for continuous operation without reprogramming.

Common types of canned cycles

1 **Drilling cycles**
   - **G81:** Standard drilling cycle for simple holes.
   - **G83:** Peck drilling cycle for deep holes, retracting periodically to clear chips.

2 **Boring cycles**
   - **G85:** Boring cycle for enlarging existing holes.
   - **G86:** Boring cycle with spindle stops.

3 **Tapping cycles**
   - **G84:** Tapping cycle for threading holes.

4 **Reaming cycles**
   - **G85:** Reaming cycle for finishing holes to precise dimensions.

Example

Here's a simple example of a G-code program using a drilling canned cycle:

```
G90 G17 G21       ; Absolute positioning, XY plane selection, metric units
G00 X10 Y10       ; Rapid move to starting position
G81 Z-5.0 R1.0 F100; Drilling cycle: drill to Z-5.0, retract to R1.0, feed rate 100
G80               ; Cancel canned cycle
```

In this example

- G81 initiates the drilling cycle, drilling to a depth of Z-5.0, retracting to R1.0, and using a feed rate of 100.
- G80 cancels the canned cycle.

Importance of program exchange between system and machine

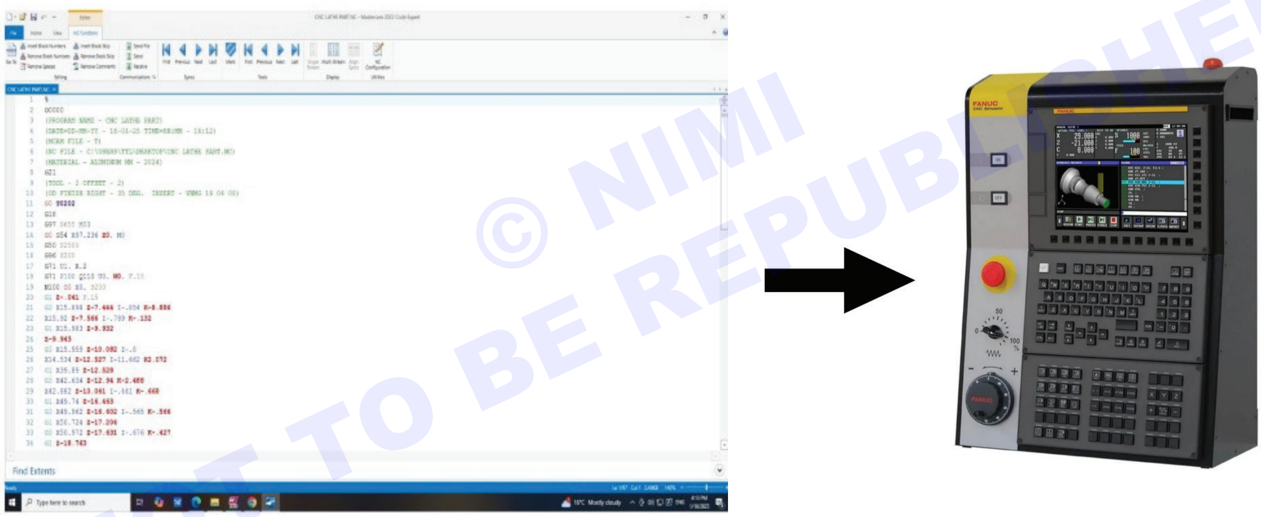
Objectives: At the end of this lesson you shall be able to
- **define program exchange**
- **state the importance of program exchange.**

Program exchange

Program exchange refers to the process of transferring machining programs, typically in the form of G-code or M-code files, from a computer system to a CNC machine. These files are generated using CAM software based on the design specifications created in CAD (Computer-Aided Design) software. The transfer can occur through various methods, such as USB drives, network connections, or direct serial communication. This process bridges the gap between virtual design and physical machining, ensuring that the CNC machine executes the exact instructions required to produce a part.

For instance, when a CAM system generates a program for a specific part, this program needs to be transferred to the CNC machine for execution. The accuracy and reliability of this transfer are crucial for achieving the desired manufacturing outcomes. (Fig 1)

Fig 1



Importance of program exchange between computer system and machine

In the modern era of manufacturing, Computer Numerical Control (CNC) machines have become indispensable for achieving precision, efficiency, and repeatability in production processes. One of the most critical aspects of CNC operations is the seamless exchange of programs between the computer system and the CNC machine. This process ensures that the design intent, as captured in a CAM (Computer-Aided Manufacturing) software, translates into precise machining operations on the shop floor.

1 Accuracy in manufacturing

The program exchange process involves transferring G-code or M-code files generated by CAM software to the CNC machine. These codes dictate the tool paths, spindle speeds, feed rates, and other parameters necessary for machining a part. Accurate transfer of these programs eliminates the risk of errors that might occur if the programs were manually inputted. This precision ensures that parts are manufactured to exact specifications, reducing the likelihood of defects and waste.

For example, consider a scenario where a complex 3D part with intricate geometries is being machined. The CAM software generates a detailed G-code program based on the design, and its accurate transfer ensures that every curve, hole, and slot is machined as intended. Any discrepancies during program exchange could result in deviations from the design, leading to costly rework or scrap.

2 Efficiency in production

A seamless program exchange between the computer system and the CNC machine significantly enhances production efficiency. Manual programming of CNC machines is a time-consuming process that is prone to human errors. By utilizing program exchange, operators can quickly upload pre-verified programs to the machine, saving valuable time and effort.

Furthermore, modern CNC machines often support networked environments where multiple machines can receive programs directly from a central computer system. This capability allows manufacturers to streamline workflows, minimize machine downtime, and optimize resource utilization. In high-volume production environments, such efficiency gains can translate into substantial cost savings.

3 Flexibility in operations

Program exchange enables greater flexibility in manufacturing operations. With the ability to transfer programs electronically, operators can quickly switch between different machining tasks. This adaptability is particularly valuable in industries where product customization or small-batch production is common.

For instance, in aerospace or medical device manufacturing, where each component may have unique specifications, program exchange allows for rapid reconfiguration of CNC machines. Instead of spending hours manually reprogramming the machine, operators can simply upload the required program and begin machining the next part.

4 Integration with industry 4.0

The concept of Industry 4.0 emphasizes smart manufacturing systems where machines, software, and humans work collaboratively. Program exchange between computer systems and CNC machines is a fundamental aspect of this paradigm. Modern CNC machines are equipped with communication protocols such as Ethernet or wireless connectivity, enabling them to interface seamlessly with CAM systems and other digital tools.

5 Error reduction and quality assurance

Manual entry of machining programs is not only time-intensive but also susceptible to errors such as incorrect toolpath data, missing commands, or improper syntax. These errors can lead to tool breakage, damage to the workpiece, or even accidents. Program exchange eliminates these risks by automating the transfer of verified programs.

Additionally, CAM systems include simulation features that allow operators to visualize the machining process before transferring the program to the CNC machine. This pre-validation step helps identify and correct potential issues, ensuring that the program runs smoothly on the machine. Such measures contribute to higher product quality and increased customer satisfaction.

Concept of importing & exporting of CNC program

Objectives: At the end of this lesson you shall be able to
- **define the concept of importing and exporting CNC programs**
- **state best Practices for program transfer.**

Concept of importing & exporting of CNC program

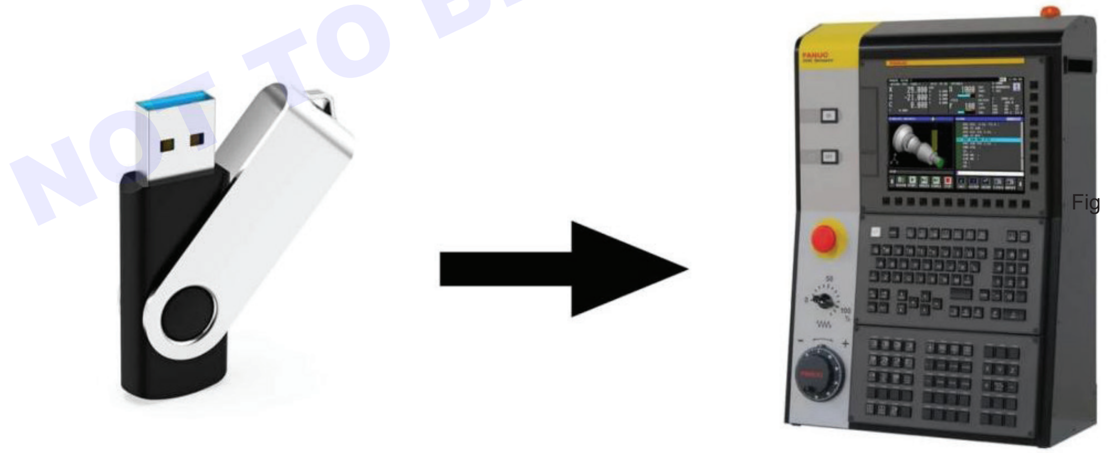
CNC (Computer Numerical Control) machines rely on precise instructions in the form of machining programs to execute manufacturing operations. Importing and exporting CNC programs are critical processes that allow for seamless communication between external storage devices, computer systems, and CNC machines. This ensures flexibility, efficiency, and accuracy in modern manufacturing environments. Understanding the concept of program import and export facilitates the smooth transfer of vital data required for machining operations.

Importing CNC programs

Importing CNC programs refers to the process of transferring machining programs, usually in the form of G-code or M-code, from external sources, such as USB drives, computers, or networked systems, into the CNC machine's memory. This process enables the CNC machine to execute the program for manufacturing operations. Importing programs allows CNC machines to work with designs created on various CAD/CAM software systems. Furthermore, programs can be edited externally and re-imported to the CNC machine to accommodate design modifications or optimizations. These processes also enable CNC machines to integrate seamlessly into Industry 4.0 environments, where multiple devices and systems communicate in real-time.

There are several methods for importing CNC programs. One common method involves using USB drives. USB devices are popular due to their simplicity and portability. Technicians can connect a USB drive to the CNC machine, switch the machine to “Edit” mode, and access the file system. The desired program file is then selected and imported into the machine’s memory, where it is assigned a program number. Another method is network-based transfer, where modern CNC machines equipped with Ethernet or Wi-Fi connectivity allow direct communication with computers or servers. Programs can be transferred using FTP (File Transfer Protocol) servers, which provide a reliable method for large-scale data exchange. For machines without advanced networking capabilities, a direct cable connection, such as an RS232 cable, can be used for transferring programs. Although slower, this method remains effective for older CNC systems. (Fig 1)

Fig 1

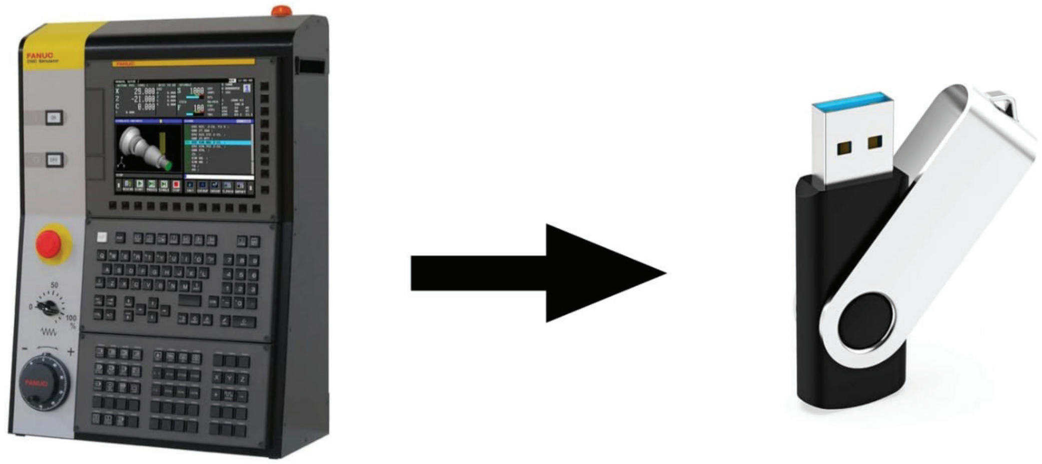


Exporting CNC programs

Exporting CNC programs involves transferring machining instructions stored in the CNC machine's memory to external storage devices or computer systems. This ensures that programs can be backed up, edited, shared, or reused for future operations. Exporting programs ensures that critical machining data is securely backed up, reducing the risk of data loss due to machine errors or memory limitations. Exporting also enables program sharing for collaboration or future use.

As with importing, USB drives are a common method for exporting CNC programs. In this process, the technician accesses the CNC memory, selects the program to be exported, assigns a file number, and saves the program onto the USB drive. Network-based exporting involves connecting the CNC machine to the PC via Ethernet and using the CNC interface to send the program file to the FTP server. These methods provide flexibility and reliability in ensuring that machining programs are effectively managed and transported. (Fig 2)

Fig 2



To ensure successful program import and export, certain best practices should be followed

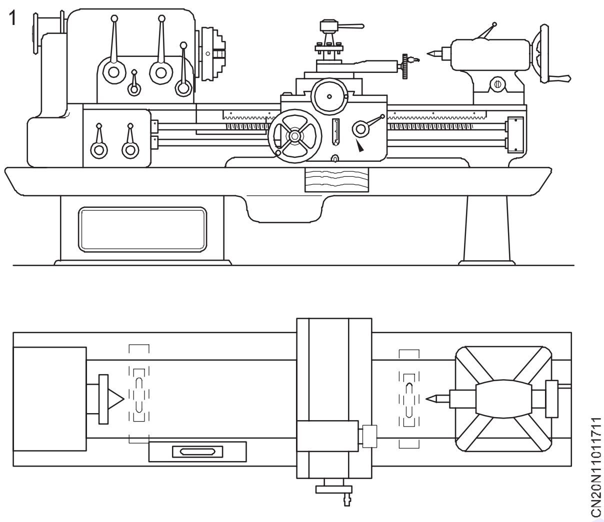
- **Verify compatibility**: Ensure that the CNC machine supports the file format and communication protocol being used.
- **Label programs clearly:** Assign clear, meaningful, and consistent names to programs. For instance, include details such as part number, revision, or operation type in the file name. This practice helps avoid confusion and ensures that the correct program is selected during the transfer process.
- **Secure data:** Protect sensitive programs with passwords or encryption when transferring over networks.
- **Test programs:** Always test the program on the CNC machine after importing to verify accuracy and functionality.
- **Organize files:** Keep a well-structured folder system for CNC programs on external devices or computers. Organizing programs based on projects, part numbers, or customers will make it easier to locate and manage files, reducing the chances of errors.

Basic maintenance of turning machine

Objectives: At the end of this lesson you shall be able to
- **explain the method of maintaining the turning machine**
- **suggest the remedial measure for the defects.**

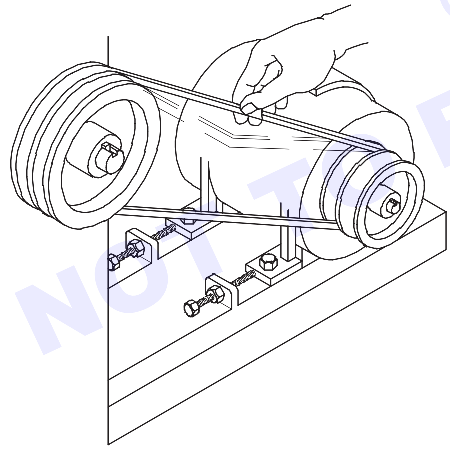
- Check the level of the lathe with a spirit level and adjust using wedges. (Fig 1)

Fig 1



- Check the tension of the belt and adjust. (Fig 2)

Fig 2



- Check the free movement of tailstock over the bed.
- Check the movement of the carriage of the lathe. (Fig 3)
- Run the machine on different spindle speeds and check the speed.
- Engage the power feed and check the longitudinal and transverse feed movements.
- Check the function of clutches by operating the clutch lever.
- Check the movement of the cross-slide and the compound slide. (Fig 4)

Fig 3

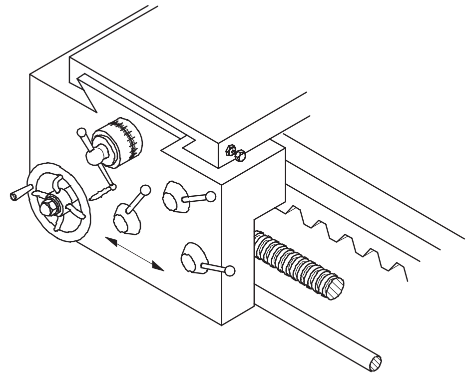
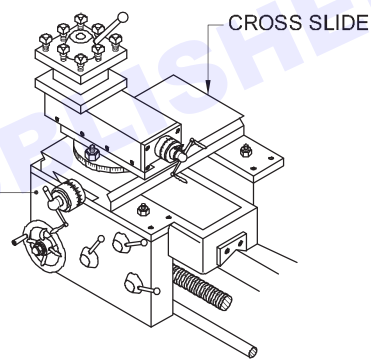
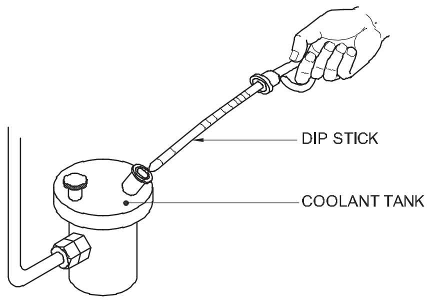


Fig 4



- Check the oil level and the functioning of the lubricating pump.
- Check the coolant level and the functioning of the coolant pump. (Fig 5)
- Check the safety guards and ensure they are in position.
- Inspect the machines as per check list and record it accordingly.

Fig 5



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

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

Basic troubleshooting of CNC machines

Objectives: At the end of this lesson you shall be able to
- **list the problem and its solution that accured in CNC machine.**

CNC machines are critical for precision manufacturing, and occasional issues can arise during operation. Below are some basic troubleshooting steps for common CNC machine problems

1 **Power issues**

- **Problem:** Machine won't power on.
- **Solution**
  - Check the main power supply and ensure it's switched ON. .
  - Inspect circuit breakers and fuses.
  - Verify the Emergency Stop (E-Stop) button is released.
  - Test the voltage levels to confirm they match machine requirements.

2 **Axis movement problems**

- **Problem:** Axis not moving or jerky motion.
- **Solution**
  - Inspect and clean the guide rails and ball screws.
  - Check for proper lubrication.
  - Verify servo motor or stepper motor connections.
  - Calibrate the axis alignment.

Tooling issues

- **Problem:** Tool breakage or improper cutting.
- **Solution**
  - Ensure the correct tool is mounted and properly tightened.
  - Verify the spindle speed and feed rate settings.
  - Check tool alignment and replace worn-out tools.

4 **Program errors**

- **Problem:** CNC machine stops mid-operation or shows an error.
- **Solution**
  - Check the G-code program for syntax errors or missing commands.
  - Ensure the correct co-ordinate system is selected.
  - Re-run the program in simulation mode before executing it on the machine.

5 **Spindle problems**

- **Problem:** Spindle not rotating or making unusual noises.

- **Solution**
  - Inspect spindle belts and tension.
  - Check for proper lubrication of the spindle.
  - Test spindle motor connections and drive.

6 **Overheating**

- **Problem:** Machine components overheating
- **Solution**
  - Check coolant levels and refill if necessary.
  - Ensure coolant nozzles are positioned correctly.
  - Clean fans, air filters and ensure adequate quotes airflow

7 **Sensor or limit switch issues**

- **Problem:** Machine stops unexpectedly or gives limit switch errors.
- **Solution**
  - Inspect limit switches for physical damage.
  - Ensure proper wiring and connections.
  - Clean sensors and confirm they are correctly aligned.

8 **Vibration or noise**

- **Problem:** Excessive vibration or abnormal noise during operation.
- **Solution**
  - Inspect machine mounts and level the machine.
  - Tighten any loose bolts or components.
  - Check for misalignment in the spindle or axis.

9 **Communication errors**

- **Problem:** Machine not responding to commands or software.
- **Solution**
  - Verify the connection between the CNC controller and the computer.
  - Check the compatibility of software versions.
  - Reset the controller or reboot the machine.

10 **Material-related issues**

- **Problem:** Workpiece not cutting accurately or shifting.
- **Solution**
  - Ensure the material is clamped securely.
  - Verify the fixture setup and adjust as needed.
  - Recheck the material dimensions and tolerance settings.

Preventive maintenance tips

- Regularly clean and lubricate all moving parts.
- Schedule routine inspections for mechanical and electrical components
- Keep the workspace free of dust and debris.
- Update CNC software and firmware to the latest versions.

By following these troubleshooting steps and maintaining the machine regularly, you can minimize downtime and improve the efficiency of your CNC operations.

Concept of TPM & OEE

Objectives: At the end of this lesson you shall be able to
- **explain the concept of TPM**
- **state advantage of TPM**
- **explain the concept of OEE**
- **describe the components of OEE and their effects.**

Total Productive Maintenance(TPM) concepts

TPM aims to maximize overall equipment effectiveness. Establishes a complete system of productive maintenance for the machines/equipments entire lifespan is implemented by various departments. [Engineering, Operations, Maintenance, Quality and Administration]

TPM can be considered as the medical science of machines.

TPM involves every single employee, from top management to all the operators on the shop floor. TPM raises and implements productive maintenance based on autonomous small group activities.

TPM is a maintenance program which involves a newly defined concept for maintaining plants and equipment.

The goal of TPM is to an extent increase production while, at the same time, increasing employee morale and job satisfaction.

TPM brings maintenance into focus as a necessary and vitally important part of the business. It is no longer regarded as a non-profit activity.

Downtime for maintenance is scheduled as a part of the manufacturing day. In some cases, as an integral part of the production process.

The goal of TPM is to stop the emergency and unscheduled maintenance.

Form different teams to reduce defects and self-maintenance.

Advantages of TPM

- Avoids wastage in quickly changing economic environment.
- Produces goods without reducing product quality.
- Reduces maintenance cost.
- Produces a low batch quantity at the earliest possible time.
- Ensures the non-defective goods to the customers.
- Reduce customer's complaints.
- Reduce accidents.
- Follow pollution control measures.
- Favourable change in the attitude of the operator.

Overall equipment effectiveness (OEE)

Overall equipment effectiveness (OEE) is a concept utilized in a lean manufacturing implementation. OEE is described as one such performance measurement tool that measures different types of production loses and indicate areas of process development. The OEE concept normally measures the effectiveness of a machine center or process line, but can be utilized in non-manufacturing operation also.

The high level formula for the lean manufacturing OEE is

OEE = Availability x Productivity x Quality

Availability

The availability is part of the above equation measures the percentage of time the machine/equipment of operation was running compared to the available time. For example if the machine was available to run 20 hours but was only run for 15, then the availability is 75 percent 15/20. The five hours when the machine didn't run would be set up time, breakdown or other downtime. The 4 hours the company did not plan to run the machine is rarely used in the calculation.

Performance

The performance part of the equation measures the running speed of the operation compared to its maximum capability often called the rated speed. For example, if a machine produced 80 pieces per hour while running, but the capability of the machine is 100, then the performance is 80% (80/100). The concept can be used multiple ways depending on the capability number. For example, the machine might be capable of producing 100 pieces per hour with the perfect part, but only 85 on that particular order. When the capability of 100 is used for the calculation, the result is more a measure of facility OEE.

Quality

The third portion of the equation measures the number of good parts produced compared to the total number of parts made. For example, if 100 parts are made and 95 of them are good, the quality is 95% (95/100).

Combining the above example into the OEE equation the OEE is

OEE = 75% x 80% x 95% = 57%

Components of OEE and their effects

Variable	Prevents	Methods to improve
Availability	Idle time of machines Adjustment time Breakdowns	Decrease the reactive maintenance improve PM scheduling Restrain personnel information transfer
Performance	Material variation inefficient work process poorly operating equipments outdated system poor lubrication	Replacements Equipment service Regular equipments PMs
Quality	Improper alignment of system Improper maintenance of records Inconsistent raw materials doubtful work	Improve the quality of product