TOOL AND DIE MAKER (Dies & Moulds)

NSQF LEVEL - 4

2nd Year

TRADE THEORY

SECTOR : CAPITAL GOODS & MANUFACTURING

(As per revised syllabus July 2022 - 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 : Capital Goods & Manufacturing

Duration : 2 Years

Trades : Tool and Die Maker (Dies & Moulds) - 2nd Year - Trade Theory - NSQF Level - 4 (Revised 2022)

Developed & Published by

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

First Edition: August 2023 Copies : 1000

Rs.345/-

FOREWORD

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

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

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

Jai Hind

Athul Kumar Tiwari, ***I.A.S***
Secretary
Ministry of Skill Development & Entrepreneurship,
Government of India.

August 2023
New Delhi - 110 001

PREFACE

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

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

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

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

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

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

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

Chennai - 600 032 **EXECUTIVE DIRECTOR**

ACKNOWLEDGEMENT

National Instructional Media Institute (NIMI) sincerely acknowledges with thanks for the co-operation and contribution extended by the following Media Developers and their sponsoring organisation to bring out this IMP **(Trade Theory)** for the trade of **Tool and Die Maker (Dies & Moulds)** under the **CG & M** Sector for ITIs.

MEDIA DEVELOPMENT COMMITTEE MEMBERS

Shri. A. Vijayaraghavan	-	Assistant Director of Training (Retd.), ATI, Chennai - 32.
Shri. M. Sampath	-	Training Officer (Retd.), CTI, Chennai - 32.
Shri. G.D. Rajkumar	-	Director GTTI, Coimbatore.
Shri. P.K. Vijayan	-	Senior Manager (Training), GTTI, Coimbatore.

NIMI - COORDINATORS

Shri.Nirmalya Nath	-	Deputy Director of Training NIMI- Chennai - 32.
V. Gopala Krishnan	-	Assistant Manager NIMI - Chennai -32.

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

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

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

INTRODUCTION

TRADE PRACTICAL

The trade practical manual is intented to be used in practical workshop. It consists of a series of practical exercises to be completed by the trainees during the course of the **Tool and Die Maker (Dies & Moulds)** trade supplemented and supported by instructions/ informations to assist in performing the exercises. These exercises are designed to ensure that all the skills in compliance with NSQF LEVEL - 4 (Revised 2022) syllabus are covered.

This manual is divided into eight modules. The eight modules are given below

Module 1	Wire Cut EDM
Module 2	CNC Lathe
Module 3	CNC Milling
Module 4	Hand injection mould
Module 5	Cavity injection mould
Module 6	Hydraulic & Pneumatics
Module 7	Machine Maintenance
Module 8	Two Cavity Injection Mould

The skill training in the shop floor is planned through a series of practical exercises centred 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 **Tool and Die Maker (Dies & Moulds)** Trade. The contents are sequenced according to the practical exercise contained in the manual on Trade practical. Attempt has been made to relate the theortical aspects with the skill covered in each exercise to the extent possible. This co-relation is maintained to help the trainees to develop the perceptional capabilities for performing the skills.

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

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

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

CONTENTS

LEARNING OUTCOME

On completion of this book you shall be able to

S.No.	Learning Outcome	Ref. Ex.No.
1	Produce components of high accuracy by different operations using Electric Discharge machine (EDM) and Wire EDM with accuracy of ± 0.02mm. NOS:CSC/N9493	83 - 84
2	Set (both job and tool) CNC turn centre and produce components as per drawing by preparing part programme. NOS:CSC/N0316	85 - 89
3	Set (both job and tool) CNC machining centre and produce components as per drawing by preparing part programme. NOS:CSC/N0316	90 - 94
4	Construct a Hand Injection Mould and try out/ test the mould assembly. NOS:CSC/N9494	95 - 96
5	Construct of two cavity injection mould and try out component. NOS:CSC/N 9495	97 - 98
6	Construct single cavity mould (Compression mould/ plunger type transformer mould). NOS:CSC/N9496	99 - 99A
7	Construct circuit of pneumatics and hydraulics observing standard operating procedure & safety aspect. NOS:CSC/N9497	100 - 101
8	Plan and perform simple repair, overhauling of different machines and check for functionality. [Different Machines – Drill Machine, milling machine and Lathe] NOS:CSC/N9498	102 - 106
9	Develop isometric drawing and construct two cavity moulds with side core. NOS:CSC/N9499	107 - 109

SYLLABUS FOR TOOL & DIE MAKER

Duration	Reference Learning Outcome	Professional Skill (Trade Practical) (With indicative hour)	Professional Knowledge (Trade Theory)
Professional Skill 50 Hrs.; Professional Knowledge 06 Hrs.	Produce components of high accuracy by different operations using Electric Discharge machine (EDM) and Wire EDM with accuracy of ± 0.02mm. NOS:CSC/N9493	83.EDM machining practice/ observation on EDM machine exercises. (25 hrs.) 84. Machining practice on Wire EDM machine. (25 hrs.)	Electrical discharge machine (EDM) introduction principle of operation, advantages and disadvantages and its applications. Introduction principle of operation advantaged and disadvantaged and applications. (06 hrs.)
Professional Skill 70 Hrs.; Professional Knowledge 10 Hrs.	Set (both job and tool) CNC lathe and produce components as per drawing by preparing part programme. NOS:CSC/N0316	85Study of CNC lathe, key board and specifications. (06 hrs.) 86 Machine starting & operating in Reference Point, JOG, and Incremental Modes. (04 hrs.) 87.Co-ordinate system points, assignments and simulations Absolute and incremental programming assignments and simulations. (20 hrs.) 88. Co-ordinate points, assignments and simulations. Identification of machine over travel limits and emergency stops. (20 hrs.) 89. Work and tool setting. Automatic Mode operation: facing, profile turning, drilling, tapping, reaming, thread cutting etc. (20 hrs.)	Safety Precautions: Safe handling of tools, equipment & CNC machines, CNC turning with FANUC CNC CONTROL- (Fanuc-OiT latest) CNC Machine and Control specifications. CNC system organization Fanuc-0i-T. Coordinate systems and Points. CNC lathe, Types, Machine axes. (10 hrs.)
Professional Skill 62 Hrs.; Professional Knowledge 10 Hrs.	Set (both job and tool) CNC machining centre and produce components as per drawing by preparing part programme. NOS:CSC/N0316	90. Study of CNC Machining centre, keyboard and specifications. (16 hrs.) 91. Machine starting & operating in Reference Point, JOG, and Incremental Modes. (06 hrs.) 92. Co-ordinate system points, assignments and simulations Absolute and incremental programming assignments and simulations. (10 hrs.) 93. Polar co-ordinate points, assignments and simulations. Identification of machine over travel limits and emergency stops. (12 hrs.) 94. Work and tool setting. Automatic Mode operation: Face Milling, profile milling, drilling, tapping, reaming etc. (18 hrs.)	Safety Precautions: Safe handling of tools, equipment & CNC machines, CNC Mill with FANUC CNC CONTROL- (Fanuc-0i-M latest) CNC Machine &Control specifications. CNC system organization Fanuc-0i-M. Coordinate systems and Points. CNC Machines Milling, Types, Machine axes. (10 hrs.)

Duration	Reference Learning Outcome	Professional Skill (Trade Practical) (With indicative hour)	Professional Knowledge (Trade Theory)
Professional Skill 75 Hrs.; Professional Knowledge 18 Hrs.	Construct a Hand Injection Mould and try out/ test the mould assembly. NOS:CSC/N9494	95. Manufacture hand injection mould. (May use the plates used in turning, milling and grinding exercise). (70 hrs.) 96. Try out and rectification. (05 hrs.)	Hand injection mould Introduction to plastic material: Types of plastics, differentiation of plastics, Properties, application, fillers and additives and reinforced plastics. Mould terminology: Core, cavity, impression, runner, gate, sprue bush, mould base etc. Parting line: Types of parting line, mould matching (Bedding down), vent and relief. Requirement for ejection: Types of ejector grids, ejector elements and ejector system. Feed System: Sprue, runner, gate, types, design and calculations, vent design, balancing, etc. (18 hrs.)
Professional Skill 150 Hrs.; Professional Knowledge 48 Hrs.	Construct two cavity injection mould and try out component. NOS:CSC/N 9495	97. Develop isometric drawing and manufacture 2 cavity injection moulds in a group of 5 trainees using various tool room machines (conventional and nonconventional machines). (130 hrs.) 98. Try out component and rectification. (20 hrs.)	Shrinkage: Introduction mould life, cavity/core dimensions, and various shrinkage values for different plastic materials. Temperature controlling of moulds: Introduction, factors effecting the cooling of moulds, layout and sizing of cooling channel, cooling integer type mould plate (core cavity, Bolster), cooling core and cavity inserts and sub inserts, mould cooling requirements and calculations. Injection moulding machines: Introduction, clamping system/ injection system terminologies and specifications, screw terminology construction of screw, types of moulding machines, and sequence in the moulding cycle. Selection of mould base, material and no. of cavities: Introduction, Selection of mould base and material, advantages and disadvantages of single/ multicavity mould, calculation of no. of cavities. Splits: External undercut components, methods of operation, split locking methods, splits safety arrangements. Side cores and side cavities: Introduction, moulding embedded side holes/ recess/slots, Design requirements for side core/ side cavities, internal side core/side cavities. Moulding internal under cuts/ threads: Definition, form pin/ split core/ side core, stripping internal under cuts purpose of threads in plastics, moulding internal threads, power and transmission system layout of impression, and moulding of external threads. (48 Hrs.)

Duration	Reference Learning Outcome	Professional Skill (Trade Practical) (With indicative hour)	Professional Knowledge (Trade Theory)
Professional Skill 100 Hrs.; Professional Knowledge 28 Hrs.	Construct single cavity mould (Compression mould/ plunger type transformer mould).	99. Manufacture single cavity plunger type transfer mould in a group of 5 trainees using various tools room machine (conventional and nonconventional) OR Manufacture multi cavity compression mould construct a single cavity compression mould in a group of 5 trainees using various tool room machine (conventional and nonconventional) (100 hrs.) 126. Identification and familiarisation of various types of hydraulic & pneumatic elements such as cylinder, valves, actuators and filters. (15 hrs.)	Moulding of thermoset materials: Introduction, processing method, compression moulding, definition, pellet, compression moulding types, advantages and disadvantages of semi positive and fully positive mould, automatic compression mould, mould heaters and thermo couples, etc., Transfer moulding, types of transfer moulding, advantages and disadvantages of transfer moulding, Injection moulding of thermo set material, Advantages and disadvantages of injection moulding of thermo set material, Compression/ transfer moulding defects. Surface finish: Mould polishing, different types and appearance required after finishing, overview of the process, standard specification of finish, mechanical equipment of mould polishing, finishing process, problems in mould polishing and solutions, surface treatment method. Multi day light mould: Introduction, under feet mould with reverse tapered sprue, floating runner plate, working system for floating cavity plate, other standard designs, some non-standard latch/ locks, some sample multi-day light design. Introduction of blow moulding, types of blow moulding advantage and disadvantage of blow moulding. Material used in blow moulding, blow moulding fault & remedy. (28 Hrs.)
Professional Skill 35 Hrs; Professional Knowledge 08 Hrs.	Construct circuit of pneumatics and hydraulics observing standard operating procedure& safety aspect. NOS:CSC/N9497	100. Identification and familiarisation of various types of hydraulic & pneumatic elements such as cylinder, valves, actuators and filters. (10 hrs.) 101. Study of simple hydraulic & pneumatic circuit. (25 hrs.)	Basic principles of hydraulics/ pneumatics system, advantages and disadvantages of hydraulics and pneumatics systems, theory of Pascal's law, Brahma's press, Pressure and flow, types of valves used in hydraulics and pneumatics system. (08 Hrs.)

Professional Skill 43 Hrs.; Professional Knowledge 08 Hrs.	Plan and perform simple repair, overhauling of different machines and check for functionality. [Different Machines – Drill Machine, milling machine and Lathe] NOS:CSC/N9498	102. Perform Periodic maintenance of lubrication system on Machines. (06 hrs.) 103. Perform simple repair work. (12 hrs.) 104. Perform the routine maintenance with check list. (05 hrs.) 105. Inspection of Machine tools such as alignment, levelling etc. (10 hrs.) 106. Accuracy testing of machine tools such as geometrical parameters. (10 hrs.)	Lubricating system-types and importance Maintenance: Definition, types and its necessity. System of symbol and colour coding. Possible causes for failure and remedies. (08 Hrs.)
Professional Skill 255 Hrs.; Professional Knowledge 90 Hrs.	Develop isometric drawing and construct two cavity moulds with side core. OR Construct an injection mould with side cavities (with cam pin) (two cavities rounded square bobbin) NOS:CSC/N9499	107. Develop isometric drawing and manufacture 2 cavity injection moulds with side cavities in a group of 5 trainees using various tool room machines (conventional and nonconventional) (220hrs.) 108. Assemble all the parts of mould and try-out and find out fault of component and rectification. (15 hrs.) 109. Prepare different types of documentation as per industrial need by different methods of recording information for the project. (20 hrs.)	Hot runner mould: Definition, runner less mould, advantages and disadvantages of hot runner moulding system, type of hot runner system, valve system, selecting a hot runner system, advantages and disadvantages of insulated runner mould and modified insulated runner mould, starting/ restarting nozzles in a manifold application. Injection moulding defects: Introduction, common faults, possible problems and remedies, analysis of moulding problems and solutions. Other moulding processes: Blow moulding, Extrusion moulding, rotational moulding, thermo forming, sheet and film forming. Multi-color moulding: Introduction, multi-color moulding, multi-material moulding and multi-process moulding. Maintenance of mould: Introduction, upkeep and maintenance, types of maintenance of idle moulds, maintenance control, and frequency of maintenance. Die cast mould: Introduction to Die casting, Die casting, gating system design, force calculation, defects and remedies. Die and mould economics: Estimation and casting of mould raw material, machining hour rate, business transactions, cost of components, activity-based costing, estimation of moulds and standard items. (90 Hrs.)

Electrical discharge machine

Objectives: **At the end of this lesson you shall be able to**
- **state what is EDM machine**
- **list elements of EDM machine**
- **brief the working principle and process of EDM machines**
- **list the advantages and disadvantages of EDM machines.**

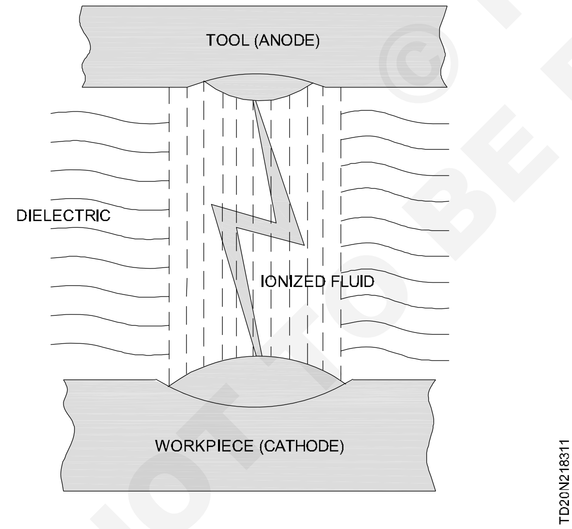
Electrical discharge machining (EDM), also known as spark machining, spark eroding, die sinking, wire burning or wire erosion, is a metal fabrication process whereby a desired shape is obtained by using electrical discharges (sparks)

Principal of EDM

Principles of EDM Electrical Discharge Machining (EDM) is a controlled metal-removal process that is used to remove metal by means of electric spark erosion In this process an electric spark is used as the cutting tool to cut (erode) the work piece to produce the finished part to the desired shape. The metal-removal process is performed by applying a pulsating (ON/OFF) electrical charge of high-frequency current through the electrode to the work piece. This removes (erodes) very tiny pieces of metal from the work piece at a controlled rate (Fig 1)

Elements of EDM are shown in Fig 2

Fig 1



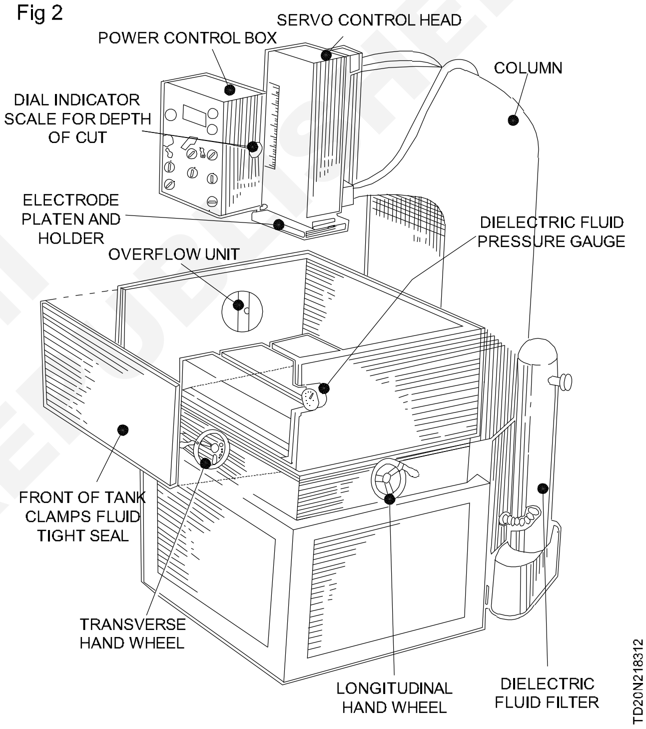
EDM Process EDM spark erosion is the same as having an electrical short that burns a small hole in a piece of metal it contacts. With the EDM process both the work piece material and the electrode material must be conductors of electricity

The EDM process can be used in two different ways:

1 A preshaped or formed electrode (tool), usually made from graphite or copper, is shaped to the form of the cavity it is to reproduce. The formed electrode is fed vertically down and the reverse shape of the electrode is eroded (burned) into the solid work piece.

2. A continuous-travelling vertical-wire electrode, the diameter of a small needle or less, is controlled by the computer to follow a programmed path to erode or cut a narrow slot through the work piece to produce the required shape.

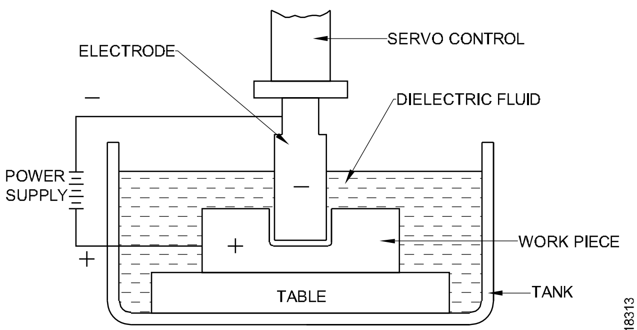
Fig 2



Conventional EDM

In the EDM process an electric spark is used to cut the work piece, which takes the shape opposite to that of the cutting tool or electrode. The electrode and the work piece are both 2 submerged in a dielectric fluid, which is generally light lubricating oil. A servomechanism maintains a space of about the thickness of a human hair between the electrode and the work, preventing them from contacting each other In EDM ram or sinker machining, a relatively soft graphite or metallic electrode can be used to cut hardened steel, or even carbide. The EDM process produces a cavity slightly larger than the electrode because of the overcut (Fig 3)

Fig 3



SCHEMATIC DIAGRAM OF EROSION MACHINE USING NEGATIVE POLARITY ELECTRODE

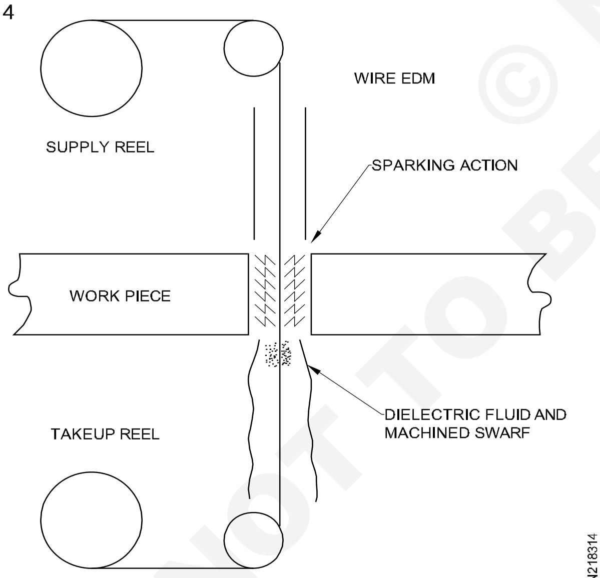
Application EDM machines are normally used to produce three dimensional shapes

These shapes utilize either cavity type machining or hole type machining Wire cut EDM machines are always used for through hole machining, since the electrode wire must pass through the work piece being machined.

Wire-Cut EDM

The wire-cut EDM is a discharge machine that uses CNC movement to produce the desired contour or shape. It does not require a special shaped electrode, instead it uses a continuous-traveling vertical wire under tension as the electrode. The electrode in wire-cut EDM is about as thick as a small diameter needle whose path is controlled by the machine computer to produce the shape required. (Fig 4)

Fig 4



TRAVELING WIRE EDM (TW EDM)

Dielectric Fluids

During the EDM process the work piece and the electrode are submerged in the dielectric oil, which is an electrical insulator that helps to control the arc discharge. The dielectric oil, that provides a means of flushing, is pumped through the arc gap. This removes suspended particles of work piece material and electrode from the work cavity

Flushing

One of the most important factors in a successful EDM operation is the removal of the metal particles (chips) from the working gap. Flushing these particles out of the gap between the work piece to prevent them from forming bridges that cause short circuits

Flushing Ram Type EDM

Flushing is the most important function in any electrical discharge machining operation. Flushing is the process of introducing clean filtered dielectric fluid into the spark gap Flushing applied incorrectly can result in erratic cutting and poor machining conditions.

There are a number of flushing methods used to remove the metal particles efficiently while assisting in the machining process. Too much fluid pressure will remove the chips before they can assist in the cutting action, resulting in slower metal removal Too little pressure will not remove the chips quickly enough and may result in short-circuiting the erosion process

Advantages of EDM

- Any material that is electrically conductive can be cut using the EDM process. Hardened work pieces can be machined eliminating the deformation caused by heat treatment.
- X, Y, and Z axes movements allow for the programming of complex profiles using simple electrodes. Complex dies sections and moulds can be produced accurately, faster, and at lower costs.
- The EDM process is burr-free.
- Thin fragile sections such as webs or fins can be easily machined without deforming the part

Disadvantages

- This process can only be employed in electrically conductive materials;
- Material removal rate is low and the process overall is slow compared to
- Conventional machining processes
- Unwanted erosion and over cutting of material can occur.
- Rough surface finish when at high rates of material removal.

Electrodes

Objectives: At the end of this lesson you shall be able to
- **state the electrodes**
- **identify the materials of electrode**
- **state the advantages and disadvantage of electrodes**
- **state the dielectric fluid**
- **state the circulation of di-electric fluid**
- **state the spark gap**
- **state the flushing**
- **state the filtering.**

Electrodes

The EDM machining process removes material from a workpiece, or piece of material being worked on, via electric discharge between two electordes, or conductors used to make contact with part of a circuit. The electrodes are separated by a dielectric liquid -- a liquid that acts as an insulator. One electrode is the tool electrode, and the other electrode is the workpiece electrode. When distance between these two electrodes decreases, the dielectric liquid breaks down, allowing curren tto flow between the electrodes. This vaporizes material from the electrodes in a controlled fashion, enabling the production of a desired workpiece shape.

Materials of Electrodes

Electrode materials must give good electrical and heat conductivity, low wear rate and be readily machineable, or economic to produce.

Table showing advantages and disadvantages of some electrode materials.

Electrode Material	Advantages	Disadvantages
Graphite (Positive Polarity can be used when roughing	Easily machined. Easily attached to tool holder by screws and/or adhesive Very good metal removal rate. Excellent wear ratio. Dense graphite grades give good Surface finish.	Needs good flow of dielectric fluid or vibration or periodic withdrawal by means of timer. Not generally used for machining carbides-tendency to arcing Dielectric fluid soaked graphite cannot be cemented for re-use. Brittle Not recommended for use with relaxation power supply when finishing.
Copper	Produces excellent surface finish. Can be used in spraying technique.	More expensive than graphite. Clogs grinding wheels.

Dielectric fluid : The dielectric fluid perfoms the following functions.

1 Helps to initiate the spark between the electorde and work.

2 Confines the spark to the narrow channel.

3 Serves as an insulator between the electrode and the work.

4 Flushes away the metal particles and prevents shorting.

5 Acts as a coolant for the elctrode and work.

The dielectric fluid must be able to ionise (Vapourise) and be ionised rapidly. It should have low viscosity so that it can be pumped through narrow machining gaps. The common dielectric fluids are

1 light lubricating oils

2 transdormer oils

3 silicon based oils

4 kerosene

5 deionised water

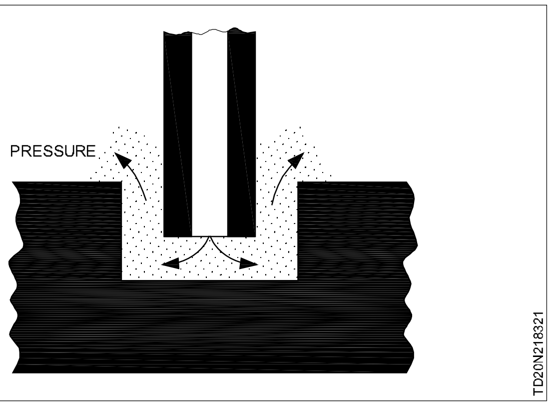
During the EDM process the workpiece and the electrode are submerged in the dielectric oil, which is an electrical insulator that helps to control the arc discharge. The dielectric oil, that provides a means of flushing, is pumped through the arc gap. This removes suspended particles of workpiece material and electrode from the work cavity.

Flushing One of the most important factors in a successful EDM operation is the removal of the metal particles (chips) from the working gap. Flushing these particles out of the gap between the workpiece to prevent them from forming bridges that cause short circuits.

Circulation of dielectric fluid

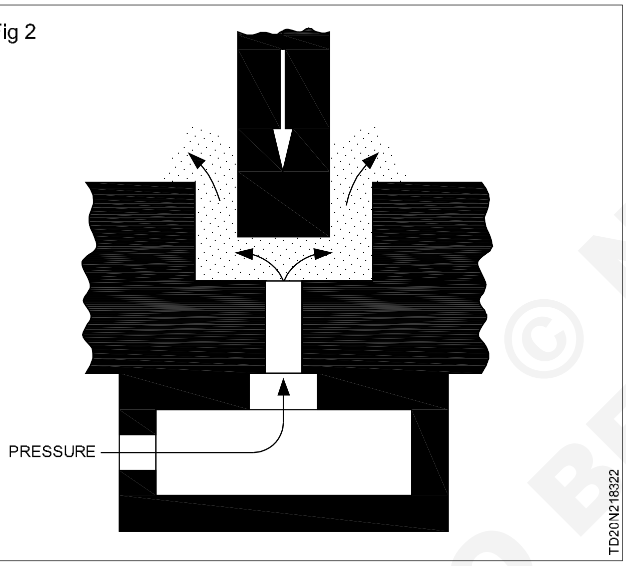
1 Down through the electrode (Fig 1)

Fig 1



2 Up through the workpiece (Fig 2)

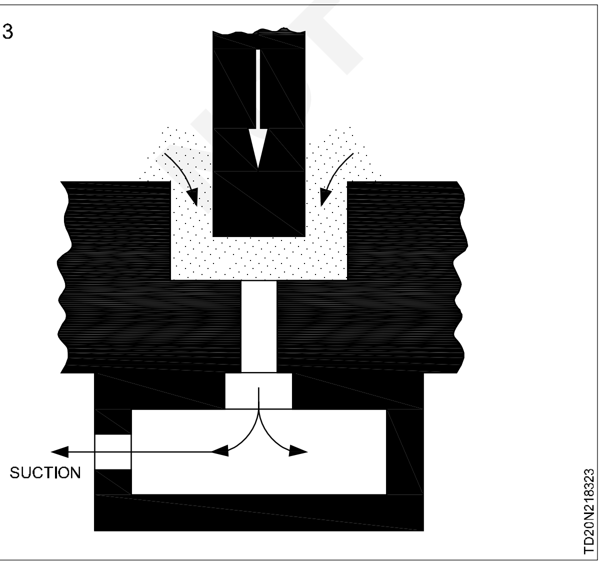
Fig 2



This method is used for flushing through holes.

3 Vacuum flow (Fig 3)

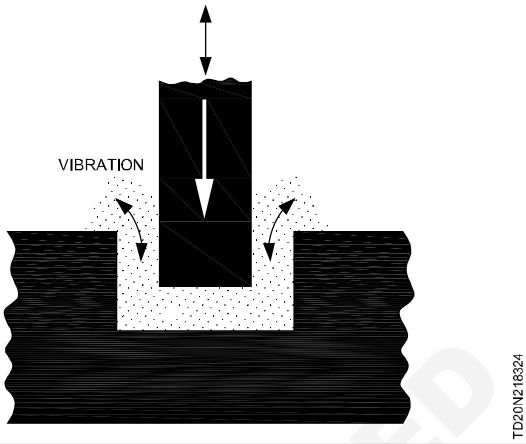
Fig 3



Used when the clearance between the electrode and work is smaller compared to above cases.

4 Vibration (Fig 4)

Fig 4



Pumping and sucking action causes the dielectric to disperse chips from the spark gap. This method is used for very small holes, deep holes and blind cavities.

Spark gap

The spark gap is the distance between the electrode and the workpiece during the machining operation. It is also referred to as overcut.

The spark gap facilitiates the flow of dielectric fluid between the electrode and workpiece. Inaddition to causing a cavity larger than the electrode to be cut (overcut). It must be remembered that the sharpest corner which can be produced is a radius equal to the spark gap.

Flushing

Once current flow in the EDM machining process stops, new dielectric liquid is added between the electrodes. This process, known as flushing, clears away debris from the workpiece.

Flushing is the most important function in any electrical discharge machining operation. Flushing is the process of introducing clean filtered dielectric fluid into the spark gap. Flushing applied incorrectly can result in erratic cutting and poor machining conditions.

There are various methods of flushing which may be chosen to suit a particular situation.

1 Passing dielectric fluid through a pipe directed at required position on workpiece.

2 (a) Passing dielectric fluid through flow holes in the electrode. The holes are designed to allow a constant passage of fluid over the whole face of the electrode.

3 Vibration can be used when it is not possible to use flush holes. This causes a pumping action to agitate elecroded particles from the spark gap.

A combination of some of the above methods can be used.

Filtering

The dielectric fluid should be effectively filtered as demanded by surface texture required. Thus, for roughing operations, no filtering may be required while for finishing a filter of 5 micro inch. may be used.

Introduction of wire EDM

Objectives: At the end of this lesson you shall be able to
- **introduction of wire EDM**
- **what is EDM wire machining**
- **list the major parts of wire EDM machine**
- **state the principle of wire EDM**
- **state the application of wire EDM**
- **differentiate between wire EDM and conventional EDM**
- **state the advantages and disadvantages of EDM.**

Introduction of Wire EDM

Wire EDM (Vertical EDM's kid brother). It was introduced in the late 1960s', and has revolutionized the tool and die, mould, and metal working industries. It is probably the most exciting and diversified machine tool developed for this industry in the last fifty years, and has numerous advantages to offer.

It can machine anything that is electrically conductive regardless of the hardness, from relatively common materials such as tool steel, aluminum, copper, and graphite, to exotic space-age alloys including hastily, wispily, inconel, titanium, carbide, polycrystalline diamond compacts and conductive ceramics. The wire does not touch the workpiece, so there is no physical pressure imparted on the workpiece compared to grinding wheels and milling cutters. The amount of clamping pressure required to hold small, thin and fragile parts is minimal, preventing damage or distortion to the workpiece.

The accuracy, surface finish and time required to complete a job is extremely predictable, making it much easier to quote, EDM leaves a totally random pattern on the surface as compared to tooling marks left by milling cutters and grinding wheels. The EDM process leaves no residual burrs on the workpiece, which reduces or eliminates the need for subsequent finishing operations.

Wire EDM also gives designers more freedom from rules in designing dies, and management more control of manufacturing, since the machining is completed automatically. Parts that have complex geometry and tolerances don't require you to rely on different skill levels or multiple equipment. Substantial increases in productivity is achieved since the machining is untended, allowing operators to do work in other areas. Most machines run overnight in a "lights-out" environment. Long jobs are cut overnight, or over the weekend, while shorter Jobs are scheduled during the day. Most work pieces come off the machine as a finished part, without the need for secondary operations. It's a one-step process.

What is EDM Wire Machining

Wire EDM, or electrical discharge machining, is a high-precision method for cutting nearly any electrically conductive material. A thin, electrically-charged EDM wire held between upper and lower mechanical guides forms one electrode, while the material being cut forms a second electrode. Electrical discharge between the wire and the workpiece creates sparks that rapidly cut away material. Both the wire and the workpiece are submerged in deionized water, which flushes away cutting debris.

As the charged wire never makes physical contact with the workpiece in EDM machining, there are no cutting forces involved, making it possible to manufacture extremely small and/or delicate parts. Parts that require levels of accuracy and intricacy that traditional machining cannot achieve can easily be produced via wire EDM.

Wire cut EDM machine is consist of machine bed, column, working table, wire running system, taper unit, working solution recycle system, Z axis, water-proof cover. (Fig1)

CNC high speed feed wire cut EDM machine is composed of machine bed, column, working table, wire running system, taper unit, working solution recycle system, Z axis, water-proof cover. (Fig 1)

Major parts of wire EDM

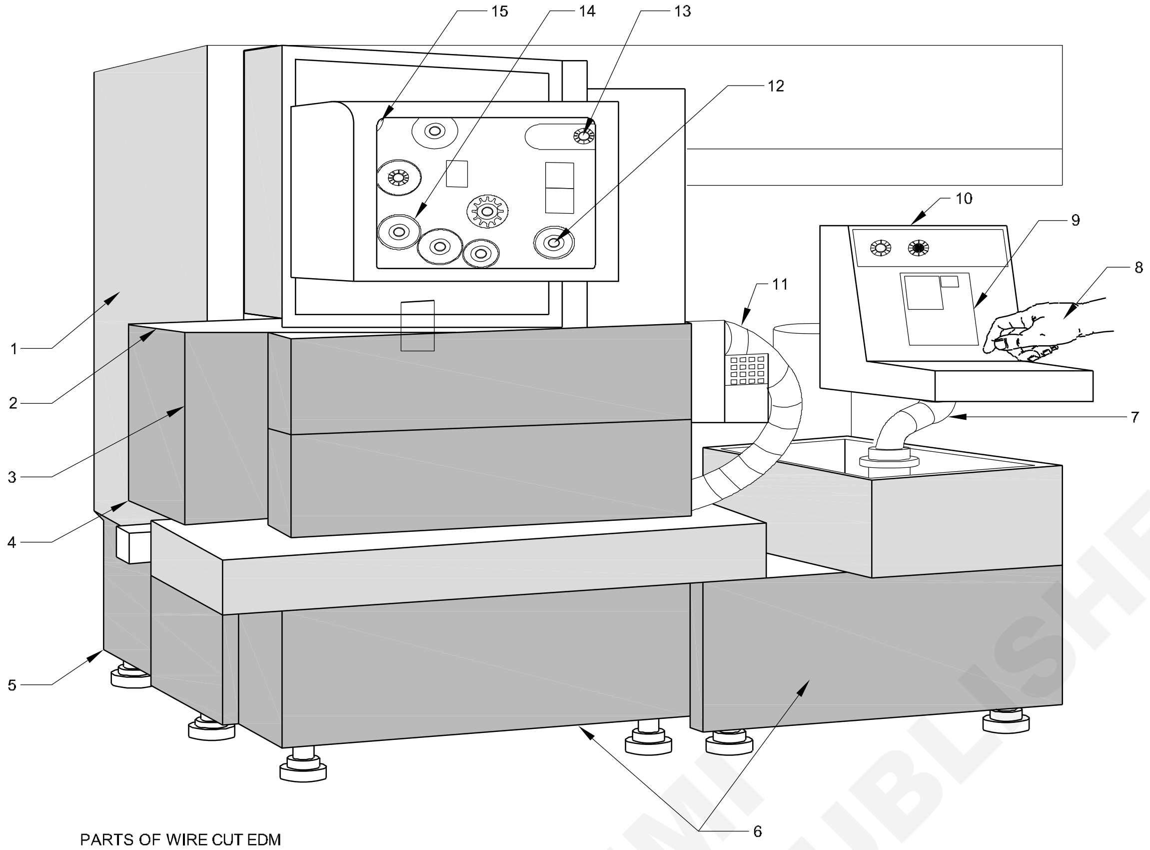
Machine bed and column (Fig 2)

Machine bed and column are two basic structure wire cut EDM machine it adopt large "T" shape machine bed, "C" structure column, HT250 casting with resin sand casting technology, compact structure, strong rigidity of whole machine. Column is fixed on machine bed as constructional element, machine bed hold whole machine as a base. Machine bed and column had been age treated to remove inner stress, to reduce deformation. Machine bed is strong enough take whole machine, good ability of anti-vibration, less thermal deformation. Manufacturing and assembly of machine bed and column must meet strict requirement of geometric and mechanical accuracy.

Working table

Working table supports and holds workpiece. Its movement is controlled by two stepper motors servo motors of closed-loop control wire cut EDM). The while wire cut EDM machining is finished by relative movement between working table and electrode wire. The working table of high speed feed wire cut EDM machine includes X and Y axis slides, adopts precious linear guide way and precious ball screw as moving components. XY cross structure had been used for decades, the mechanical rigidity and

Fig 1



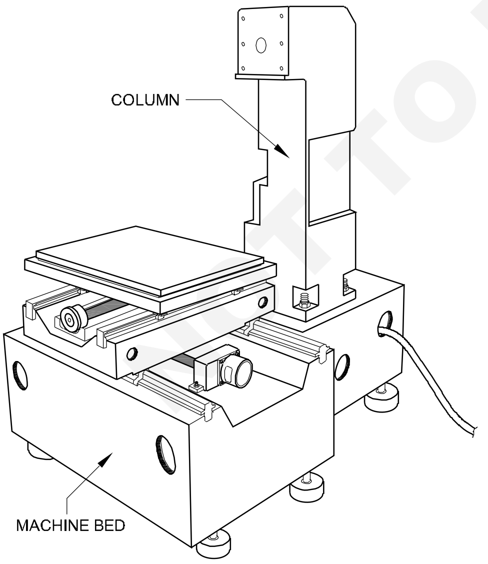
PARTS OF WIRE CUT EDM

1.COLUMN	6.DIELETRIC FLUID TANK	11.HAND HELD REMOTE CONTROL
2.WIRE GUIDE ARM	7.AC SERVO MOTOR	12.COMPACT WIRE CHOPPER
3.WORK TABLE	8.FEED INPUT / PROGRAM FEED	13.WIRE SPOOL UNIT
4.TRANSPARENT SAFETY GUARD	9.LED TOUCH PANEL	14.AUTOMATIC WIRE THREADING (AWT)
5.BASE	10.CNC CONTROLLER	15.DE-COILER UNIT

EDM MECHINE

TD20N218411

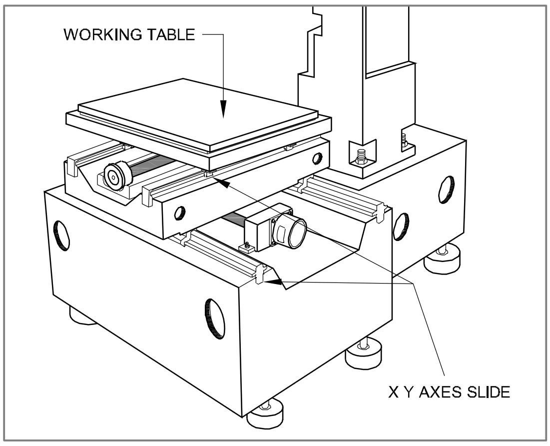
Fig 2



TD20N218412

controllability of Cross structure had been proved and widely accepted, its design and manufacturing technology is pretty mature today, widely used for many machine tools. (Fig 3) of cross structure XY axis slides.

Fig 3

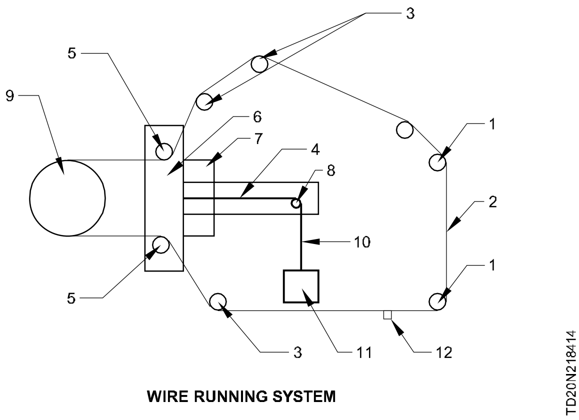


TD20N218413

Wire running system (Fig 4)

Wire running system controls electrode wire feed at a certain speed and tension, electrode wire feed reciprocally, and coil electrode wire on the wire drum without overlap, in Fig 4 part no.1-5 is diagram of wire running system of high speed feed wire cut EDM machine, this system mainly includes guide, wire drum, power contact, tension control units.

Fig 4

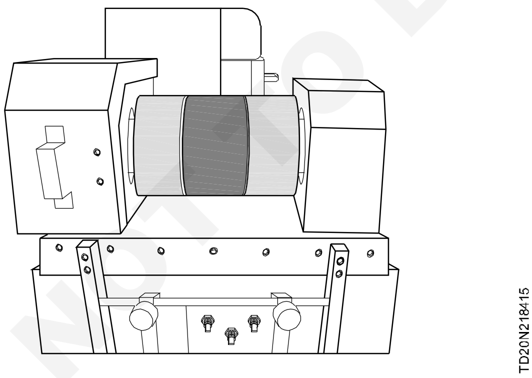


1- Main guide; 2-electrode wire; 3- auxiliary guide; 4-linear guide; 5-guides of tightening; 6-slide block; 7- slide of linear guide way; 8-fixed pulley; 9-wire drum; 10-rope; 11-weight drop; 12-electical block(power contact)

Wire cut EDM machine is working, guides need to bear fast feeding speed of electrode wire, normally use rolling bearing as guide. There is strict accuracy requirement of "V" groove of guide the radius of arc at the bottom of groove must be smaller than radius of electrode wire to avoid transverse movement of electrode wire. Basically, reduce the weight of guide at the condition of that guide has met required strength. Beside, the working surface should have enough hardness and low surface roughness.

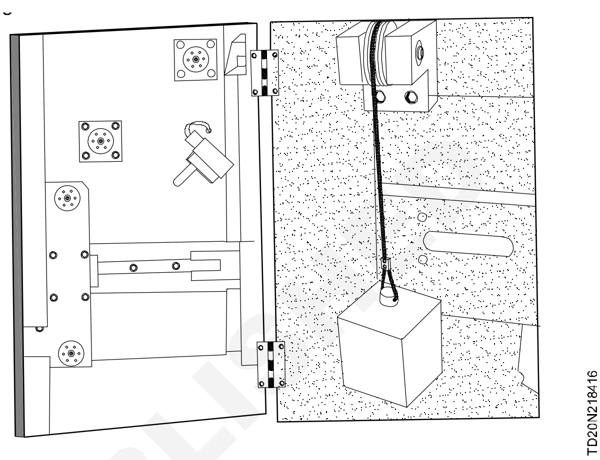
Wire drum is one of key units to ensure electrode feeding quickly and where to store electrode wire. (Fig 5) Wire drum does axial moving while turning at high speed, this way, it can coil electrode wire on the wire drum evenly and tidily without overlap, wire drum is able to turn clockwise and counterclockwise to realize wire feed reciprocally. In order to ensure wire drum turn smoothly, inverter was brought into wire cut EDM machine to control the speed of wire drum turning.

Fig 5



Wire cut EDM machine, electrode wire obtain cathode electricity from power feed contact (electrical block), the contact resistance of power feed contact must be very small, otherwise, power feed contact touch the fast feeding for long time, so it must be wear resistant, normally, hard alloy, with good wear resistance and conductivity, is chosen as material of power feed contact.

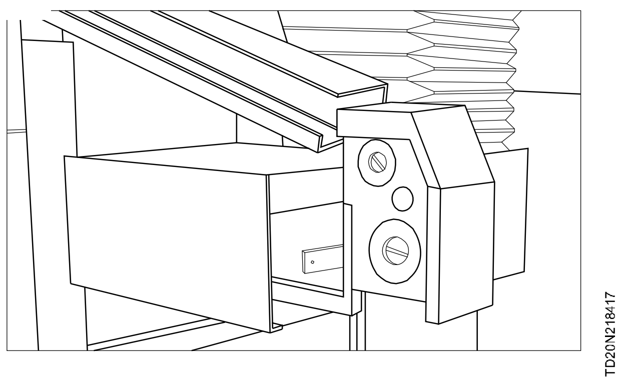
During process of wire cut EDM machining (Fig 6) electrode is getting loose under impact of direction change, discharging, this situation will affect machining accuracy and surface finish. If there is no tension control unit, it is necessary to tighten electrode manually, when cut large thickness workpiece, cutting quality will getting worse. At present, some high rank CNC high speed feed wire cut EDM machine has weight drop tension control system; it can keep constant tension, save a lot work of operator and increase cutting quality obviously.



Taper unit

Taper cutting is realized through taper unit (Fig 7).the move axis of taper unit is U and V axis. Usual taper cutting principle is central axis of down guide doesn't move, fixed, up guide is driven by stepper motor U and V axis and move in four directions. Working table and taper unit are moving simultaneously to realize some angle of electrode.

Fig 7



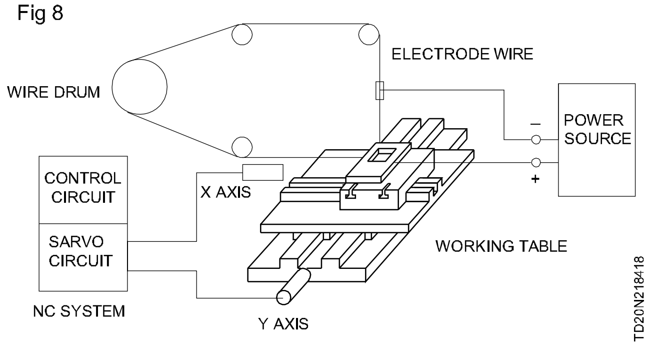
Working solution recycle system

Pump sucks working solution out of tank, then working solution goes through filter to get rid of unclean stuff; and filtered working solution goes into up pipe and then up and down nozzle separately, then go back to tank through filter net. It is necessary to change working solution and filter if they affect cutting quality

Principle of wire cut EDM

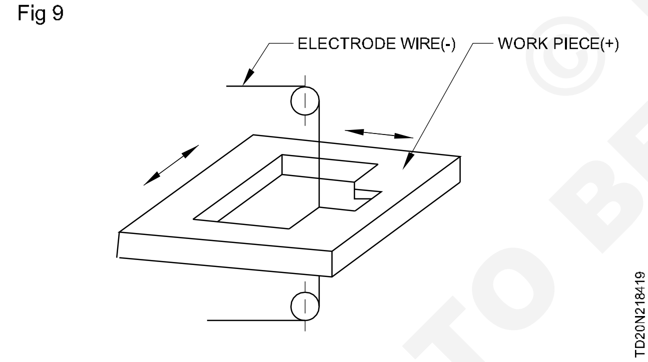
Wire cut EDM machine puts impulse voltage between electrode wire and workpiece through impulse source, controlled by servo system, to get a certain gap, and realize impulse discharging in the working liquid between electrode wire and workpiece. Numerous tiny holes appear due to erosion of impulse discharging, and therefore get the needed shape of workpiece. (Fig 8)

Fig 8



Electrode wire is connecting to cathode of impulse power source, and workpiece is connecting to anode of impulse power source. When workpiece is approaching electrode wire in the insulating liquid and gap between them getting small to a certain value, insulating liquid was broken through; very shortly, discharging channel forms, and electrical discharging happens. And release huge high temperature instantaneously, up to more than 10000 degree centigrade, the eroded workpiece is cooling down swiftly in working liquid and flushed away. (Fig 9)

Fig 9



Three basic conditions that wire cut EDM work correctly

1 The gap between electrode wire and workpiece should be certainly maintained in a required range. Within this range, not only impulse power can break through insulating liquid to create spark discharging, but also the eroded workpiece can be flushed away after discharging process. If gap is too big, insulating liquid can't be break through,, and there will be no spark discharging; if gap is too small, short circuit is easy to happen, no spark discharging neither.

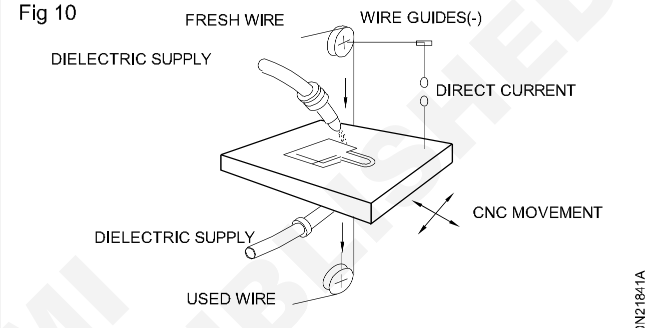
2 The procedure should happen in the liquid with insulate capacity, for example saponification and deionized water, the liquid could act as medium of discharging channel and provide cooling and flushing.

3 Electrical discharging should be short time impulse discharging, as with short discharging time, the released heat won't affect inside material of workpiece, and limits energy to a tiny field and keep characteristics of cool machining of wire cut EDM machine.

Application of wire cut EDM machining (Fig 10)

Wire cut EDM machining is mainly used to process various punch die, plastic mould, Powder metallurgy mould and etc, which have 2D and 3D faces combined, or components. It can also cut various sample plate, magnetic steel, Silicon Steel Sheet, semi-conductive material or precious metal. Further more, it is able to do tiny machining, abnormal shape groove or machining of standard defect of sample parts, widely used in electrics, precious machine tools, light industry, army industry and so on.

Fig 10



APPLICATIONS-WIRE EDM

1 Mold machining

Wire cut EDM is widely used to machine various moulds, such as punch die, squeezing die, powder metallurgy mould, bend mould, plastic mould. Among these different kind moulds, cutting punch die take a great share, to precious punch die machining, wire cut EDM machining is a indispensable technology. By adjusting different compensation value while programming, wire cut EDM can cut terrace die, punch plate, stripper plate and etc, it is easy to meet the requirement of mould fitting clearance and machining accuracy.

Advanced punch dies have characteristics of complicated structure, big producing difficult, high accuracy, long life, high efficiency and low consumption ratio of material. These new advanced technology of mold also brings continuous development of wire cut EDM technology,

Mechanical parts machining

To mechanical parts machining, wire cut EDM is suitable to machine parts with large varieties and small quantity, special material which is not easy to processed by conventional machines, special gear, forming cutting tools, various shape holes.

When try to make new product, use wire cut EDM machine to cut parts out of workpiece, there is no need to make moulds, producing time is shortened obviously, at the same time, cost is reduced. Besides, as it is easy to adjust electrical parameters of wire cut EDM, so it is possible to overlap thin plate and cut together to increase machining efficiency. And wire cut EDM machine can also machine parts with tiny gap, narrow slit and complex shape.

Differences between Wire Cut and Conventional EDM

There are two main types of EDM: conventional and wire. Conventional EDM, as described above, uses a tool to disperse the electric current. This tool, the cathode, runs along the metal piece, the anode, and the electrical current reacts to melt or vaporize the metal. As a result of the dielectric fluid, what little debris produced washes away from the piece. Wire EDM or WCEDM discharges the electrified current by means of a taut thin wire, which acts as the cathode and is guided alongside the desired cutting path, or kerfs. A dielectric fluid submerges the wire and workpiece, filtering and directing the sparks. The thin wire allows precision cuts, with kerfs as wide as three inches and a positioning accuracy of +/- 0.0002". This heightened precision allows for complex, three dimensional cuts, and produces highly accurate punches, dies and stripper plates.

Wire cut EDM equipment is run by computer numerically controlled (CNC) instruments, which can control the wire on a three-dimensional axis to provide greater flexibility. Whereas conventional EDM cannot always produce tight corners or very intricate patterns, wire EDM's increased precision allows for intricate patterns and cuts. Additionally, wire EDM is able to cut metals as thin as 0.004". At a certain thickness, wire EDM will simply cause the metal to evaporate, thereby eliminating potential debris. The wire of a WCEDM unit emits sparks on all sides, which means the cut must be thicker than the wire itself. In other words, because the wire is surrounded by a ring of current, the smallest and most precise cutting path possible is the added diameter of the ring and wire; technicians easily account for this added dimension. Manufacturers continue to produce thinner and thinner wires to allow for smaller kerfs and even finer precision.

Advantages of EDM Machining

EDM machining offers a broad range of advantages that other manufacturing methods cannot match. Some reasons to choose Wire EDM include:

Precision: When you need highly accurate machining, EDM manufacturing is your best option. It is capable of making cuts that match your designs within ± 0.0001".

Intricacy: Wire EDM is an excellent option for manufacturing delicate and complex parts. With Wire EDM, it is possible to manufacture the same intricate part over and over again with the same accuracy.

Shorter lead times: Because the EDM process is so accurate, it makes it possible to get it right the first time, significantly shortening the time it takes to get a product from the prototype phase to the marketplace.

Less set up time: EDM manufacturing reduces the need for tooling, which in turn lessens the amount of time that it takes to set up.

Tapers: EDM machining is useful for cutting long tapers that other machining methods cannot produce.

Accurate internal cuts: Square-edged internal cuts are difficult to produce with other machining methods, but Wire EDM makes it possible.

Better results: EDM machining is capable of producing parts that do not have burrs or require tooling. Wired EDM also produces a better surface finish than other forms of manufacturing.

Fine hole drilling: EDM drilling is capable of producing very small holes that are difficult to produce with other methods.

Disadvantage of wire cut EDM machining

As the electrode needs to go through workpiece, so it can't machine blame hole kind or step plane kind workpiece, besides, machining efficiency of wire cut EDM machine is compare to other conventional metal processing equipments.

Wire Electrode Materials

Materials like steel are difficult to cut or mould without adding heat first. Wire EDM makes it possible to cut steel through and other materials without applying heat. EDM machining works best with materials that conduct electricity, such as

- Brass
- Copper
- molybdenum
- Bronze
- Tungsten
- Carbon steel
- High alloy
- Stainless steel
- Titanium

The wire electrode for WEDM is usually made of copper, brass, or molybdenum in a diameter ranging from 0.01 to 0.5 mm. Stratified copper wire coated with zinc brass with diameter of 0.25 mm is often used.

Dielectric fluid

Deionized water is always used as a dielectric fluid in WEDM to provide a larger gap size and lower wire temperature in order to reduce the wire rupture risk. This fluid also serves to flush debris from the gap and thus helps maintain surface quality.

Safety precautions

Objectives: At the end of this lesson you shall be able to
- **list the safety precaution to be observed while handling tools**
- **brief the safety precautions to be observed while handling equipments**
- **state the safety point while working on CNC machines.**

Precaution for using cutting tools

Item	Caution	Counter plan
General Cutting Tools	Direct touch to the sharp cutting edge may cause injury.	• When placing them in the machine or taking them out of the case, please wear protective gloves.
	Excess impact or heavy wear will increase cutting resistance may cause tool breakage or dispersion of broken pieces.	• Use safety items, such as safety glasses and protective gloves. • Use them according to our recommended cutting condition.
	During cutting operation, cutting tools get very hot. Touching the tools immediately after operation may cause burn.	• Use safety items, such as safety glasses and protective gloves.
	Sparks, generation of heat or chips in high temperature during operation may cause fire.	• Don't operate in an area where there is a potential of fire or explosion. • When oil-coolant is used, please be sure to have a fire prevention system installed.
	Lack of dynamic balance in high-speed revolution can cause vibration and broken tool.	• Please use safety items, such as safety glasses and protective gloves. • Please operate test-run before cutting, and confirm that there is no vibration or unusual sound.
Indexable Cutting Tools	Loose inserts or parts may cause injury.	• Please clean the insert pocket and fixture before operation. • Please tighten the inserts with supplied wrench only, and confirm that the inserts and parts are clamped completely.
	When clamped too tight by supplementary tools like pipe etc, inserts or body may be broken.	• Please tighten them with supplied wrench only.
	When index able tools are used in high-speed revolution, they may burst out of the body due to centrifugal force.	• Please use them according to our recommended cutting condition. (See our catalogue for instruction.)
	When drilling through hole with milling work, a kind of disk (reminder parts sometimes flies out from the end of the table.	• Please use safety items, such as safety glasses, protective gloves and covers on the machine.

Precaution for using measuring instrument

Lubricate measuring instruments

Proper lubrication of measuring instruments will prevent damage due to corrosion and oxidation. Precision measuring instruments should be lightly oiled after each use, and any excess oil should be removed from metal surfaces with a clean, dry cloth. Avoid spraying your instruments with WD-40, as it may leave a film that can alter the calibration of the instrument

Store instruments in clean environment

Whenever you place an instrument in its storage location, be sure that it is protected from dust and moisture. Do not store precision instruments against one another in a drawer. Avoid stacking instruments on top of each other, unless they are well-packed. keep measuring instruments stored in an area as far away from vibration.

Properly handle during use

Once you have a precise measuring instrument in your hand, it becomes that much more important to protect that tool from harm or providing false readings. Do not drop, throw, or bang measuring instruments against hard surfaces.

If you use precision instruments often at a workbench, then use rubber mat for resting . This will help protect your instruments from damage. In addition, never use a measuring instrument to pry or hammer another object.

If you are handling a measuring instrument, it is important to avoid permitting it to either heat up or cool down. Expansion and contraction can alter the calibration of the instrument.

General safety on CNC machine

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

Safety check on the machine - before start

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

Safety check on the machine

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

Safety while machining

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

Safety - while working on machining centre

- First check the operations of the machine then try to machine a workpiece, preferably a trail with out job / tools is advisable to avoid unexpected accident.
- Check first thoroghly the work zero point , tool offsets etc. These two checks are very important.
- Check speed and feed in the program otherwise it may cause vibrations, loosening of the fixture etc. and end up in an accident.
- Avoid changing the factory set parameters; setting wrong and unknown parameter may damage the machine.
- Immediately after switching on the power do not touch any key on the MDI panel until the position display appears on the screen. Some of the keys are dedicated to maintenance (or) other special operations.
- Unexpected pressing of these keys may end up abnormal condition.
- Operations and programming manuals are general which describes several machines function. Therefore some functions may not be available to your machines. Hence check the specifications.

- Some functions may be implemented at the request of the consignees. These functions may be referred with the machine manual.

Safety - related to programming

- **Coordinate system setting**

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

- **Position by nonlinear interpolation**

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

- **Function involving 4th axis**

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

- **Inch/ metric conversion**

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

- **Stroke check**

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

- **Tool offset check**

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

- **Plane selection**

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

- **Programmable mirror image**

  Note that programmed operation vary considerably when a programmable image is established.

- **Compensation function**

  Compensation is cancelled during reference position return command (or) a command based on the machine coordinate system. Before issuing such commands always cancel compensation function mode.

- Every time when the workpiece zero is set kindly avoid programming with G10 use the G 10 command when standard fixture is used for machining.

Safety - while replacing the memory backup battery

When the battery alarm is on the CRT, only maintenance people who know this work may be allowed.

Follow the given instructions

- First power on the CNC.
- Apply the emergency on.
- Do not touch any high Voltage part of this mark.
- Replace only the recommended type batteries after checking the voltage & current.
- Secure the batteries correctly otherwise loss of data may end with a heavy loss when the alarm is displayed on the CRT within a week's time.

Note

- **Change the battery within a week's time, which will prevent the contents of the memory,in programs, offsets, and parameters.**
- **Refer the maintenance manual for further details.**

CNC machine specification and control specifications

Objectives: At the end of this lesson you shall be able to
- **interpret the CNC turning centre machine specification**
- **explain the features of Fanuc controler for the CNC turning centre.**

Specification

A specification often refers to a set of documented requirements to be satisfied by a material, design, product, or service. A specification is often a type of technical standard.

There are different types of technical or engineering specifications (specs), and the term is used diffeently in different technical contexts. They often refer to particular documents, and/or particular information within them. The word specification is broadly defined as to state explicitly or in detail" or "to be specific".

The purpose of a specification is to provide a description and statement of the requirements of a product, components of a product, the capability or performace of a product, and/ or the service or work to be performed to create a product. The term product is a very braod term that could inculde commodities, supplies, goods, equipments, materials, construction and services.

Example of CNC lathe specification as shown in Table 1.

SPECIFICATIONS OF CNC LATHE

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

CNC Control (Fanuc 18iTB)

Standard Specifications

- Max. controlled axes (NT, 2 axes / VTC, 3 axes)
- Simultaneously controlled axes (2 axes)
- Least input increment 0.001 mm:0.0001”
- Inch/Metric conversion
- Mirror image
- Chamfering on/off
- Blacklash compensation
- Pitch error compensation
- MDI operation
- Program number search
- Sequence number search
- Dry run
- Single block
- JOG feed
- Incremental feed (x1, x10, x100, x1000)
- Jog and handle simultaneous mode
- Manual handle feed (1 unit/each path)
- Positioning - G00
- Linear interpolation
- Circular interpolation
- Dwell
- Threading, synchronous cutting
- Skip function-G31
- Reference positon return--G28
- Reference position return check--G27
- 2nd referene position return
- Rapid traverse rate (Max. 10m/min)
- Rapid traverse override (F0,25,50,100%)
- Feed per minute/Feed per revolution
- Tangential speed constant control
- Cutting feedrate clamp
- Automatic acceleration/deceleration
- Override cancel
- Manual per revolution feed
- Optional block skip (1)
- Max. programmable dimension (8-digit)
- Program number (04-digit)
- Sequence number (N5-digit)
- Input unit 10 time multiply
- Rotary axis roll-over function
- Coordinate system shift
- Direct input of coordinate system shift
- Manual absolute on and off
- G code system (A)
- Sub program call (4 folds nested)
- Canned cycles (G90, G92, G94)
- Custom macro B
- Chamfering/corner R
- Auxillary Function (M8-digit)
- Spindle speed function (S5-digit)
- Spindle serial output (S5-digit)
- Constant surface speed control
- 1st spindle orientation
- Tool function (T4 digits)
- Tool offset pairs (16 pairs)
- Tool nose radius compensation (G40-G42)
- Tool offset value counter input
- Part program storage length 80m
- Number of registerable programs (63)
- Part program editing
- Background editing
- Status display
- Clock function
- Self-diagnosis function
- Alarm display
- Alarm history display
- Operation history display
- Help function
- Servo setting screen
- Configuration
- Language display English
- Data protection key
- Erase CRT screen display
- External workpiece number search 9999
- Memory card interface
- Reader/puncher interface RS 232C
- External message
- External workpiece number search 15
- Status output signal
- Setting and display unit 10.4” color LCD
- Manual pulse generator

Optional Accessories

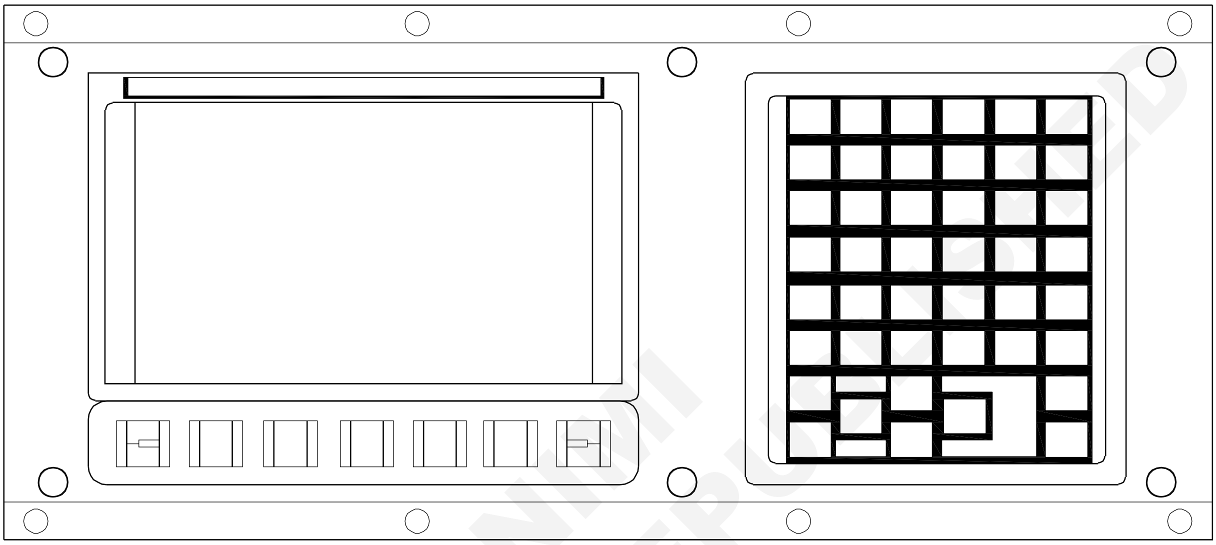
- Increment system 1/10 (0.0001mm, 0.00001”)
- Tool retract and recover
- Manual linear/circular interpolation
- Polar coordinate interpolation
- Continuous threading
- Variable lead threading
- Circular threading
- Polygon turning
- Rapid traverse bell shaped acceleration/deceleration
- Linear acceleration/deceleration after cutting feed interpolation
- Optional block skip (9)
- Program restart
- Cylindrical interpolation
- Helical interpolation
- Programmable data input
- Interruption type custom macro
- Multiple repetitive cycle
- Workpiece coordinate system
- Addition of custom macro common variables (#100-#199-#500-#999
- Rigid tapping
- Conversional programming with graphic display
- Tool offset pairs (64/99 pairs)
- Tool life management fucntion
- Automatic tool offset
- Direct input of tool offset value measured B
- Part program storage length (160/ 320/640/1280)
- Number of registerable programme (125/200/400/1000)
- Graphic function
- Fanuc handy file
- Ethernet Function

Machine control and its functions

Objectives: At the end of this lesson you shall be able to
- **describe the mahcine control specification**
- **state the functions of CNC machine controls.**

Machine control functions (Fig 1)

Fig 1



TD20N228711

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.

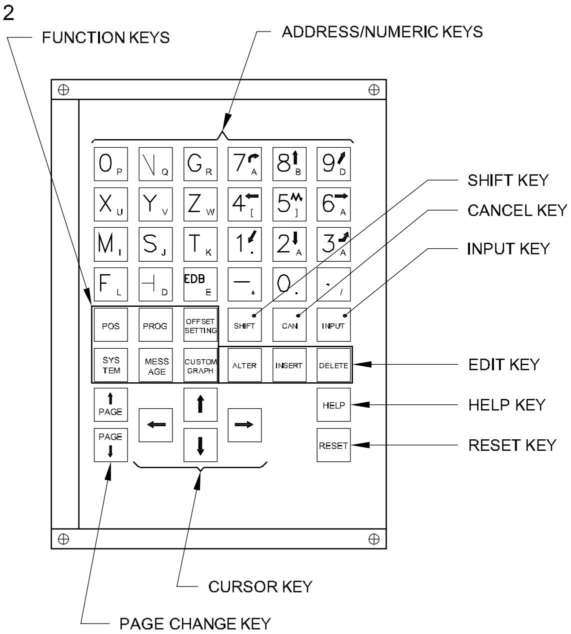
Monochrome

LCD/MDI unit

The layout of the various control keys in the MDI panel are as follows.

MDI key location (Fig 2)

Fig 2



TD20N228712

MDI KEY LOCATION

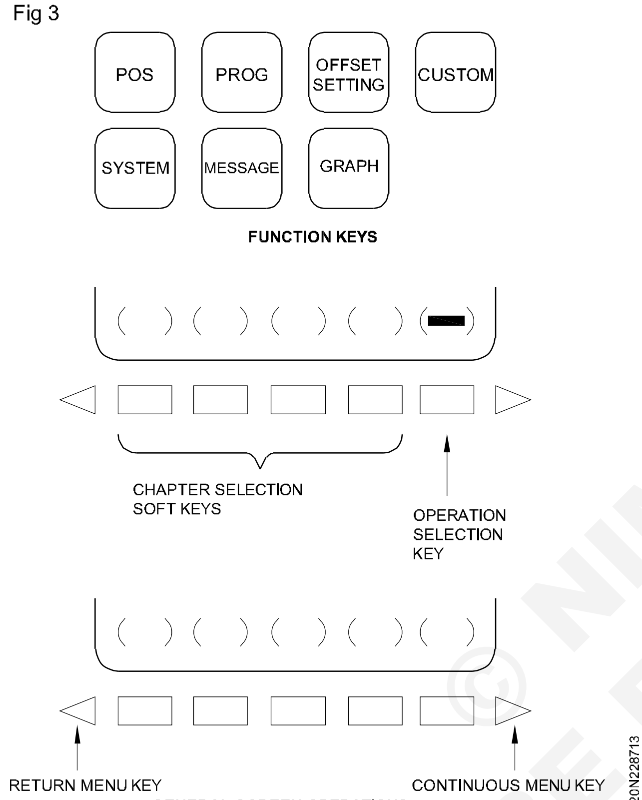
Explanation of the MDI keyboard

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	The soft keys have various functions, according to the applications. The soft key functions 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] : ALTER [RESET] : INSERT [DELETE] : 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.

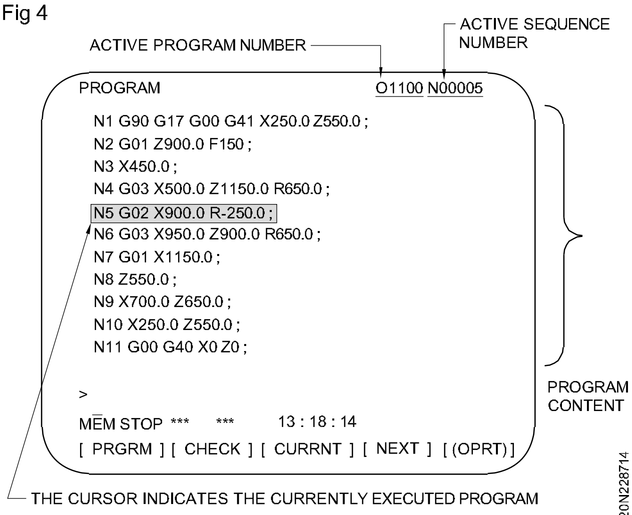
[GRAPH] Press this key to display the graphics screen.

[CUSTOM] Press this key to display the custom screen (conversational acro screen).

Program display

The contents of the currently active program are displayed. In addition the programs sheduled next and the program list are displayed. (Fig 4 & 5)

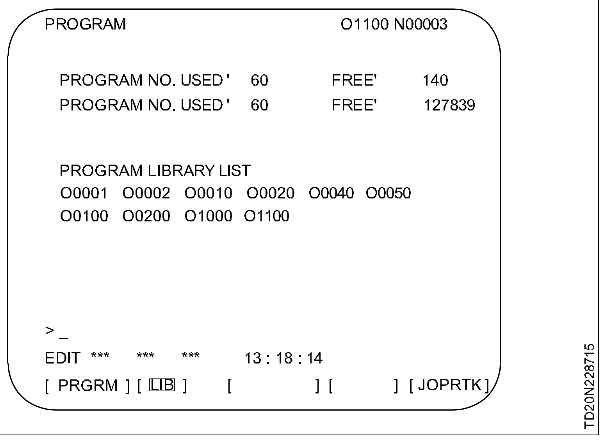
Fig 4



PROGRAM CONTENTS DISPLAY

The cursor indicates the currently executed location.

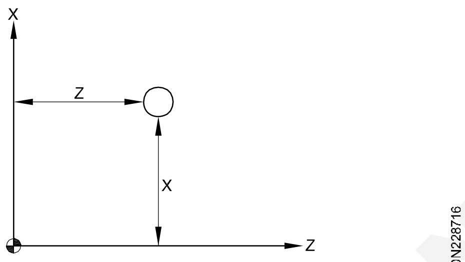
Fig 5



Current position display

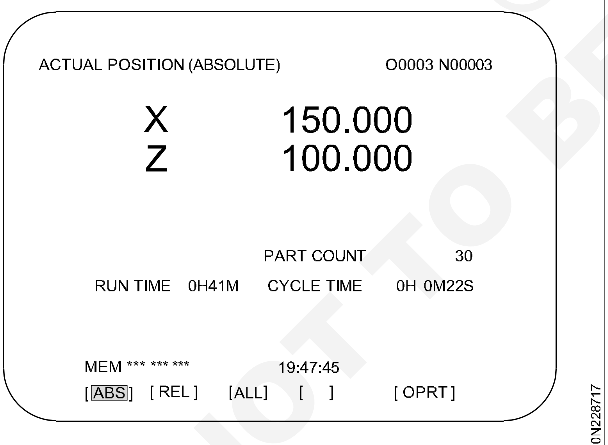
The current position of the tool is displayed with the coordinate values. The distance from the current position to the target position can also be displayed. (Fig 6 & 7)

Fig 6



WORKPIECE COORDINATE SYSTEM

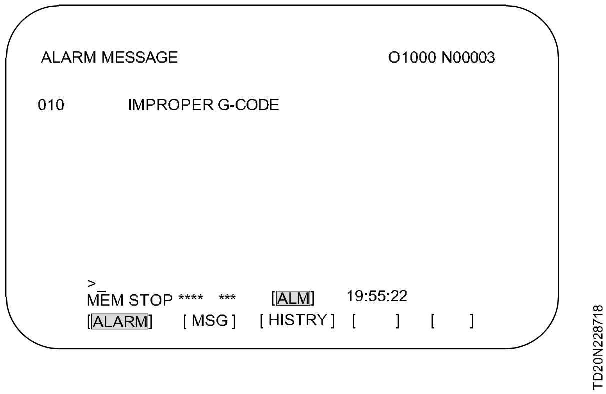
Fig 7



POSITION DISPLAY IN THE WORK (ABSOLUTE) COORDINATE SYSTEM

When a trouble occurs during operation, alaram numbers and alaram message are displayed on CRT screen. See APPENDIX G for the list of alarms and their meansings. (Fig 8)

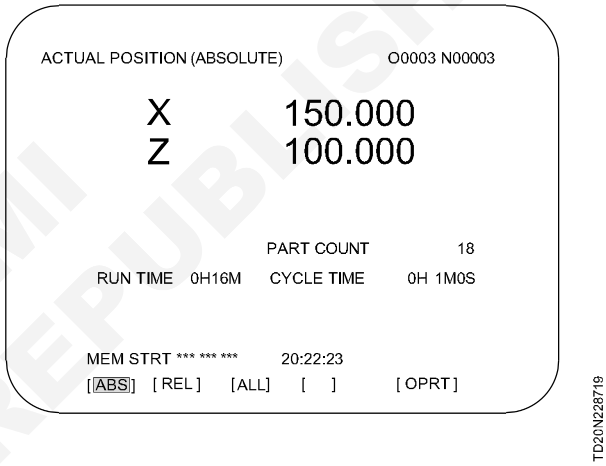
Fig 8



Parts count display run time display

Two types of run time and number of parts are displayed on the screen. (Fig 9)

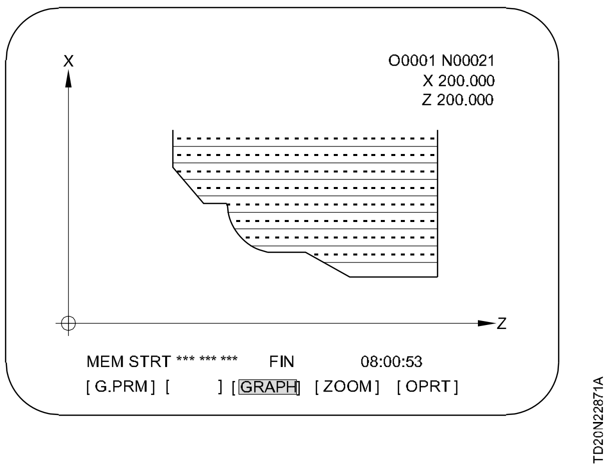
Fig 9



Graphic display

The graphic can be used to draw a tool pathe for automatic operation and manual operation, thereby indicating the progress of cutting and the position of the tool. (Fig 10)

Fig 10



For specification: Refer Related Theory for Exercise 2.2.103

Co-ordinate systems and points

Objectives: At the end of this lesson you shall be able to
- **state what is co ordinate system**
- **list the types of co ordinate system related to CNC machine**
- **explain the each coordinate system and their points**
- **state what is machine zero and work zero**
- **explain the CNC system organisation.**

A coordinate system is a method for identifying the location of a point on the earth. Most coordinate systems use two numbers, a coordinate, to identify the location of a point. Each of these numbers indicates the distance between the point and some fixed reference point, called the origin. The first number, known as the X value, indicates how far left or right the point is from the origin. The second number, known as the Y value, indicates how far above or below the point is from the origin. The origin has a coordinate of 0, 0.

Types of co ordinate system used in CNC machines

- Catesian co-ordinate system
- Absolute co-ordinate system
- Incremental or Relative co-ordinate system
- Polar co-ordinate system
- Machine co-ordinate system/Machine zero point
- Work co-ordinate system/Work zero point

A **Cartesian coordinate system** is coordinate system that specifies each point uniquely in a plane by a pair of numerical **coordinates,** which are the signed distance to the point from two fixed perpendicular directed lines, measured in the same unit of length. Each reference line is called a coordinate axis of the system, and the point where they meet is its orgin, at ordered pair (0,0). The coordinates can also be defined as the positions of the perpendicular projections of the point on to the two axes, expressed as signed distances from the origin.

One can use the same principle to specify the position of any point in three-dimensional space by three cartesian coordinates, its signed distances to three mutally perpendicular planes (or, equivalently, by its perpendicular projection onto three mutually perpendicular lines). In general, n-cartesian coordinates (an element of real n-space) specify the point in an n-dimensioanl Euclidean space for any dimension n. These coordinates are equal, up to sign, to distances from the point to n mutually perpendicular hyper planes.

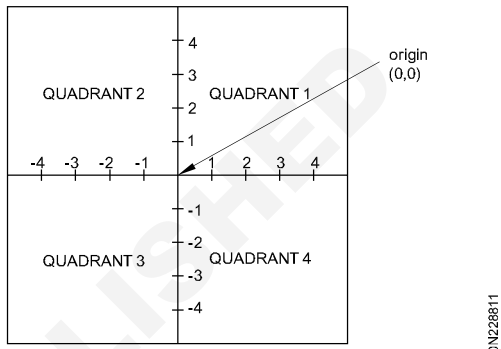
Coordinate system and ordered pairs

A coordinate system is a two-dimensional number line, for example, two perpendicular number lines or axes.

This is a typical coordinate system:

The horizontal axis is called the x-axis and the vertical axis is called the y-axis.

Fig 1



The centre of the coordinate system (where the lines intersect) is called the origin. The axes intersect where both x and y are zero and is taken as origin The coordinates of the origin are (0,0).

An ordered pair contains the coordinates of one point in the coordinate system. A point is named by its ordered pair in the form of (x,y). The first number corresponds to the X-coordinate and the second to the Y-coordinate.

To graph a point, you draw a dot at the coordinates that corresponds to the ordered pair. It's always a good idea to start at the orgin. The x-coordinate tells you how many steps you have a take to the right (positive) or left (negative) on the x-axis. And the y-coordinate tells you how many steps to move up (positive) or down (negative) on the y-axis.

The ordered pair (3,4) if found in the coordinate system when you move 3 steps to the right on the x-axis and 4 upwards on the y-axis.(Fig 2)

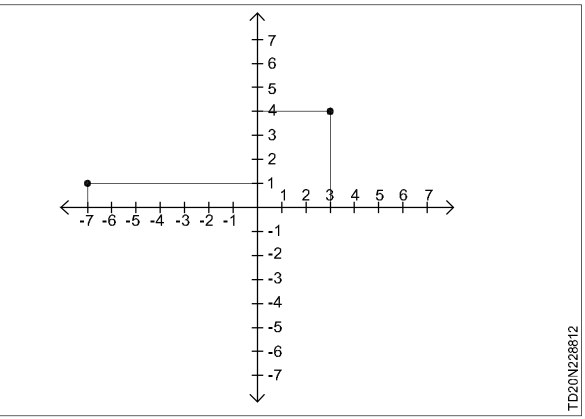
The ordered pair (-7, 1) is found in the coordinate system when you mave 7 steps to the left on the x-axis and 1 step upwards on the y-axis.(Fig 2)

To find out the coordinates of a point in the coordinate sysem you do the opposite. Begin at the point and follow a vertical line either up or down to the x-axis. There is your x-coordinate. And then do the same but following a horizontal line to find the y-coordinate.

Absolute coordinates

Absolute coordinates are based on the orgin (0,0), which is the intersection of the X and Y axes. Use absolute coordinates when you know the precise X and Y values of the point.

Fig 2



With dynamic input, you specify absolute coordinates with the # prefix is not used. For example, entering #3,4 specifies a point 3 units along the X axis and 4 units along the Y axis from the orgin.

The following example draws a line beginning at an X value of -2, Y value of 1, and an endpoint at 3,4. Enter the following in the tooltip.

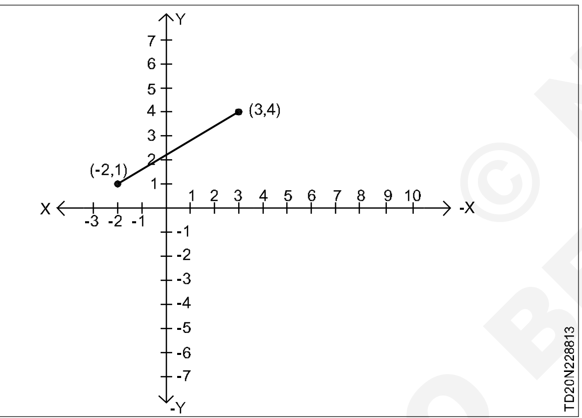
Command: **line**

From point: **#-2,1**

To point: **#3,4**

The line is located as follows: (Fig 3)

Fig 3



Relative coordinates or incremental coordinate

Relative coordinates are based on the last point entered. Use relative coordinates when you know the location of a point in relation to the previous point.

To specify relative coordinates, precede the coordinate values with an @ sign. For example, entering @3,4 specifies a point 3 units along the X axis and 4 units along the Y axis from the last point specified.

The following example draws the sides of a triangle. The first side is a line starting at the absolute coordinates -2,1 and ending at a point 5 units in the X direction and 0 units in the Y direction. The second side is a line starting at the endpoint of the first line and ending at a point 0 units in the X direction and 3 units in the Y direction. The final line segment uses relative coordinates to return to the starting point. (Fig 4)

Command: **line**

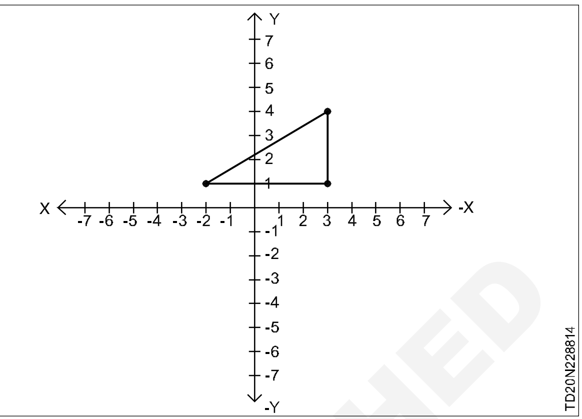
From point: **#-2,1**

To point: **#5,0**

To point: **#0,3**

To point: **#-5,-3**

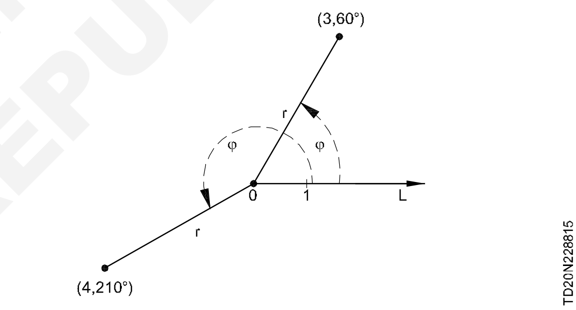
Fig 4



Polar coordinate system (Fig 5)

Points in the polar coordinate system with pole 0 and polar axis L. The point with radial coordinate 3 and angular coordinate 60 degrees or (3, 60°). and the point (4, 210°) represents polar coordinates.

Fig 5



In mathematics, the **polar coordinate system** is a two-dimensional coordinate 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 orgin of a Cartesian coordinate system) is called the pole, and the ray from the pole in the reference direction is the polar axis. The distance from the pole is called the radial coordinate or radius, and the angle is called the angular coordinate, polar angle.

Uniqueness of polar coordinate & conversion to cartesian coordinate.

Adding any number full turns (360°) to the angular coordinate does not change the corresponding direction. Similarly any polar coordinate is identical to the coordinate with the negative radial component and in the opposite direction.

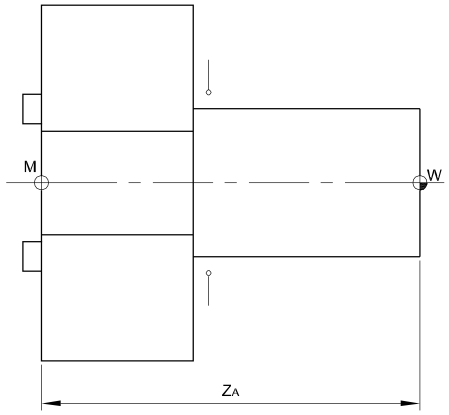
CNC Lathe (turning) Machines

Coordinate system (Fig 6)

To ensure that the control system of the machine will read the specified coordinates correctly to indicate the position of the workpiece; the machine tool has its own "coordinate system".

The following points are part of this system.

Fig 6



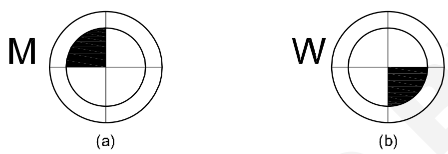
MACHINE ZERO POINT AND WORKPIECE ZERO POINT

Machine Zero Point (M) (Fig 7)

The origin of the coordinate system. It is defined by the manufacturer and cannot be changed. In general, the machine zero point M is located in the center of the work spindle nose for CNC lathes.

Fig 7a shows (M) machine zero point and 7b shows (W) work zero point

Fig 7



Workpiece Zero Point (W) (Fig 8)

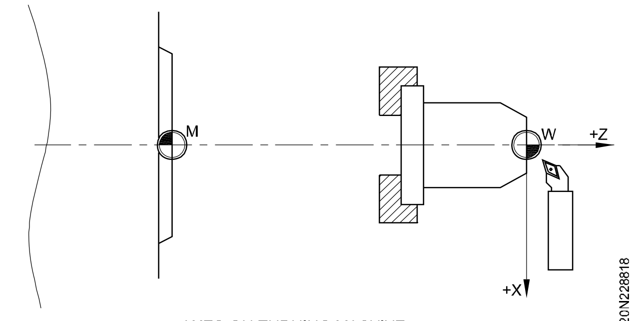
The workpiece zero point (W) is the origin of the work part-based coordinate system. Its location is specified by the programmer. The ideal location of the work part zero point allows the dimensions to be directly taken from the drawing. In case of turning the workpiece zero point is generally in the center of the left or right side of the completed part.

Axes on turning machine

Turning Machine axes

CNC Turning machine has at least controllable feed axes, marked as X and Z; (Fig 8)

Fig 8



AXES ON TURNING MACHINE

- When the cutting tool moves toward and backward the machine spindle, this is called movement along Z axis.
- When the cutting tool moves in cross direction to the longitudinal axis of the workpiece, this is called movement along X axis.
- Positive Z direction is when the tool moves away from the workpiece in Z axis.
- Positive X direction is when the tool moves away from the work part in X axis.

Dimensioning

To machine a workpiece we need a technical drawing on which we should illustrate the required dimensions to make the required shape. To dimension the workpiece we need to specify a certain point on it, from which we should take the measurement. This point is the origin point. The origin point on the workpiece is called Workpiece zero point (W).

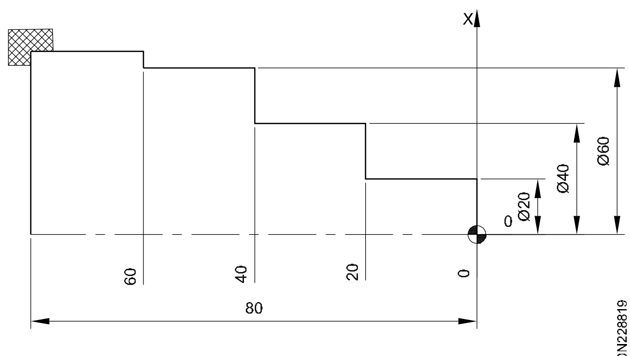
There are two types of dimensioning

Absolute Dimensioning (Fig 9)

All measurements are taken from the workpiece zero point.

In the drawings for CNC turning absolute system considers the value of X as the diameter value not the radius.

Fig 9

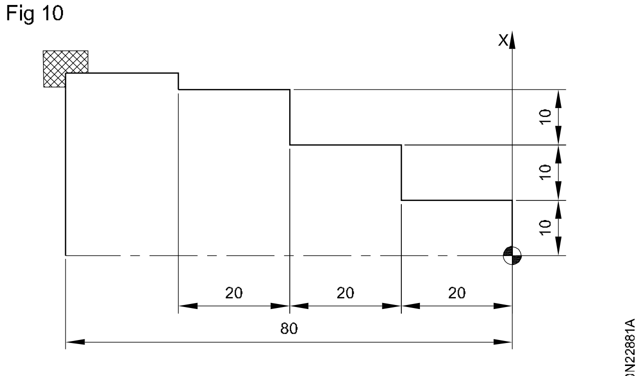


ABSOLUTE DIMENSIONING

Incremental Dimensioning (Fig 10)

Uses incremental values that are always measured from the current point to the next point

Fig 10

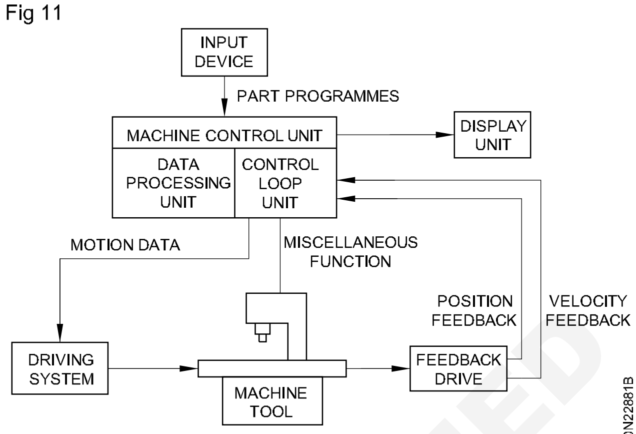


INCREMENTAL DIMENSIONING

CNC System organisation (Fig 11)

The given flow charts clearly shows how the CNC system is smoothly looped with different control like input device, machine control unit, driving system, feed back system.

Fig 11



CNC SYSTEM ORGANISATION

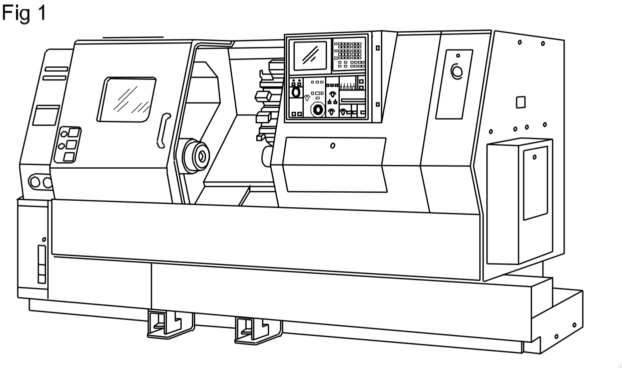
CNC lathe types and points

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).

Fig 1



HORIZONTAL TURNING CENTER

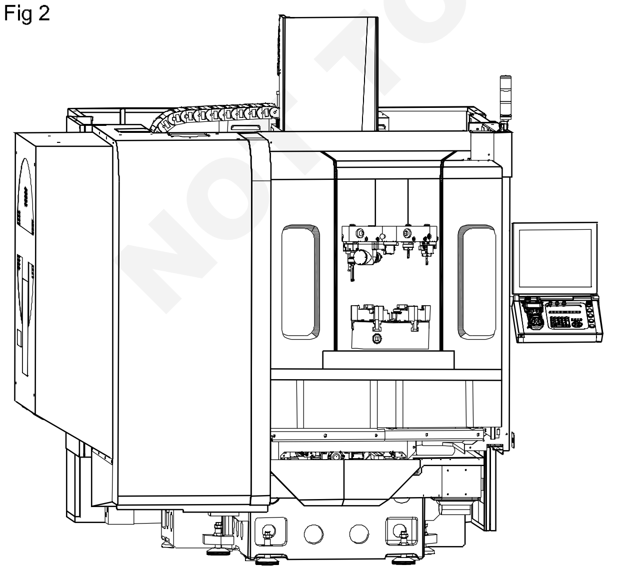
There are two types of CNC turning centers: Horizontal (Fig 1) and Vertical (Fig 2). Horizontal turning centers are the most common. Vertical turning centers are typically called a vertical turret lathe or VTL.

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



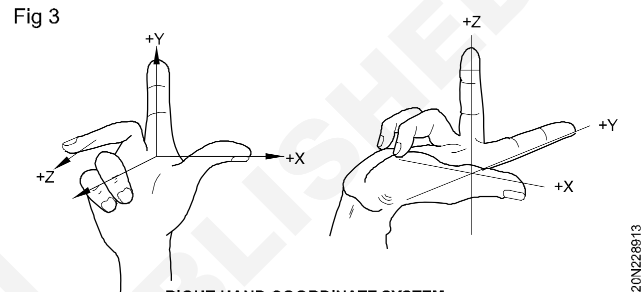
VERTICAL TURNING CENTER

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)**

Fig 3



RIGHT HAND COORDINATE SYSTEM

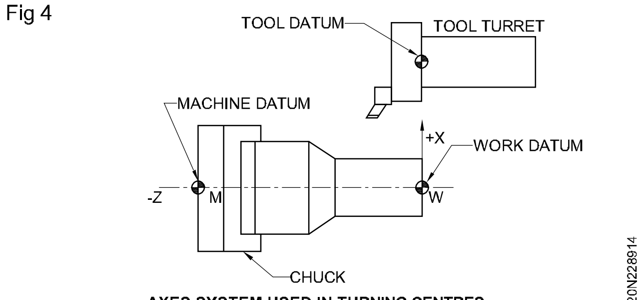
Designation of axes

First axes to be identified is the z axis .This is then followed by x and y axes respectively.

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 toward 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

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.

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.

Fig 5

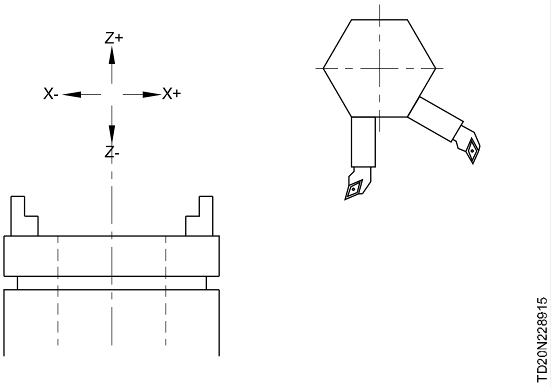
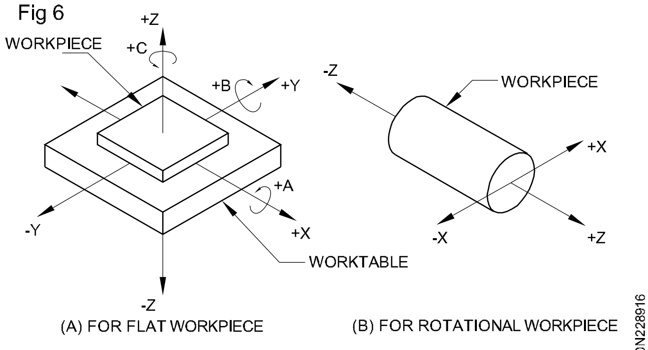


Fig 6



DESIGNATION OF AXES

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)

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 matching. 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.

Preparation of part programming

Objectives : At the end of this exercise you shall be able to
- **explain part programming preparation**
- **state the purpose of 'G' codes and 'M' codes**
- **explain the canned cycles and it formats.**

Introduction

Part programming comprises of collection of data, arrangement of information in a standard format and calculation of tool path, the data relates to dimensions of feature, direction of cutting , tool required, sequencing, and familiarity with NC system codes.

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, 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 specifies 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 programme, 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 turning ON/OFF spindle ,coolant ON/OFF or rewinding the tape .These functions do not relate two dimensional movement of the machine. This is more explained in next topic.

8 End of Block (EOB)

It identifies the end of instruction block.

G and M codes (G-codes)

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

List of G codes

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:

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

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 manufactured defines only G codes which are same on all the machines with the same control.

M00	Program stop
M01	Optional stop
M02	End of program execution
M03	Spindle forward(CW , as viewed towards the tail-stock)
M04	Spindle reverse (C CW , as viewed towards the tail-stock)
M05	Spindle stop
M06	Auto tool change V (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 orientation
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 input -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

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;        │
_______________;           │  Part program
_______________;           │
_______________;           │
M01;                       ┘

N2;                        ┐
G28  U 0.0 W 0.0;          │
G50  S 1200 T 0200;        │
_______________;           │  Part program
_______________;           │
_______________;           │
M01;                       │
M30;                       ┘
```

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

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 input for the following addresses.

X, Z, U, W, A, B, C, I, J, K, P, R, Q, F.

Note:

1 micron=0.001mm

1 mm=1000 microns

1 inch=25.4mm

1 sec=1000 millisec

Structure or format of a part program

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

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

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

Program 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

```
            %              (Program start)
            O3642          (Program number)
Blocks  {   N010 ---------
            --------------
            --------------
            N100 M02;      (Program end)
```

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

Comments

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

Example

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

Table common word addresses used in word address format

Address	Function
N	Sequence number to identify a block.
G	Preparatory word that prepares the controller for instruction given in the block.
X, Y, Z	Coordinate data for three linear axes.
U, V, W	Coordinate data for incremental moves in turning in the X,Y and Z directions respectively.
A, B, C	Coordinate data for three rotational axes X, Y and Z.
R	Radius of arc, used in circular interpolation.
I, J, K.	Coordinate values of arc centre, corresponding to X, Y and Z-axes respectively.
F	Feed rate per minute or revolution in either inches or millimeters.
S	Spindle rotation speed.
T	Tool selection, used for machine tools with automatic tool changer or turrets.
D	Tool diameter word used for offsetting the tool.
P	It is used to store cutter radius data in offset register. It defines first contour block number in canned cycles.
Q	It defines last contour block number in canned cycles.
M	Miscellaneous function.

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 - Programme end

The code is inserted at the end of the program. The machine stops permanently. Spindle rotation, Feed of axis and coolant discharge all 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 modal.

G codes which remains active only in the block in which it is programmed, is said to be Blockwise 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 conduct with the work piece.

3 Machine 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 feedrate specified in mm/Rev.

Function F

The feed rate is used to move the tool from one point to another point with constant feedrate. 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

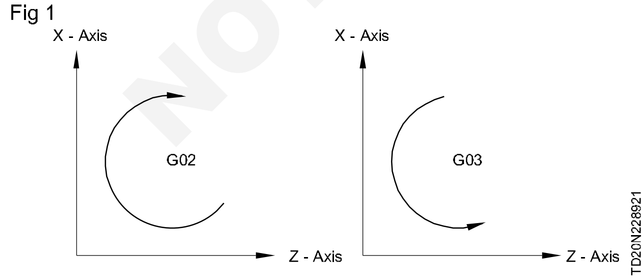
G01 X 50.0 Z -50.0 F0.1; X -axis & Z - axis move with feed 0.1mm/Rev.

Circular interpolation (Fig 1)

G02-Circular interpolation clockwise direction

G03-Circular interpolation Anti clockwise direction

Fig 1



Format

G02 } X --Z --R --F --;
G03 }

OR

G02 } X --Z --I --K --F --;
G03 }

Where

XZ - End point of Arc

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

R - Radius of the arc

F - Feed

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

Example

G02 X 40.0 Z-5.0 R 5.0 F 0.1

G03 X 40.0 Z-5.0 R5.0 F 0.1

Where, R=Radius

G04-Dwell

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

Format

G04 (U, X, P) time

Example

G04 U 1.0 (Dwell of 1.0 second)

Note

Decimal point is not available in 'P'

Ex. Dwell of 2.5 seconds.

G04 U 2.5

G04 X 2.5

G04 P 2500

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

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

1 G28 U0

2 G28 W0

3 G28 U0 W0

G30 - Second reference return

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

1 G30 U0

2 G30 W0

3 G30 U0 W0

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

1 G50 X—Z—;

Ex. G50 X 300.0 Z 150.0;

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

Ex. G50 X300.0 Z 150.0 S 3000

G96-Constant Surface Speed Control (Cutting Speed Specification)

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

The G96 is used when the cutting speed is specified.

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

Calculation for cutting speed

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

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

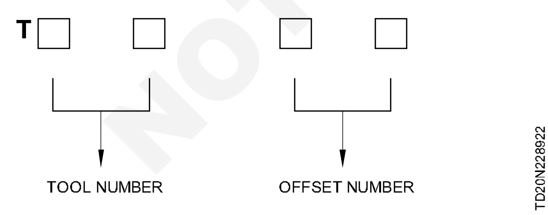
a. Threading

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).

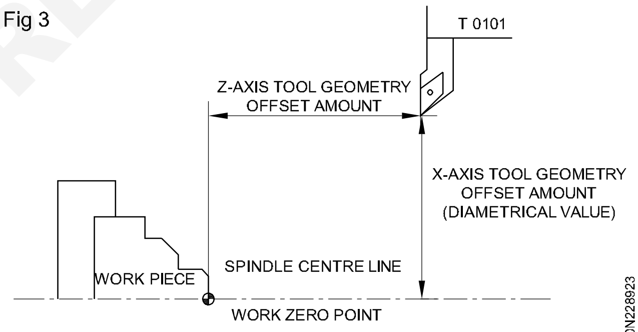
Tool geometry offset (Fig 3)

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 and the amount to be offset should be decided by the operator.

Procedure for setting work coordinate system (Fig.4)

Step 1 Make sure that the component is securely clamped.

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

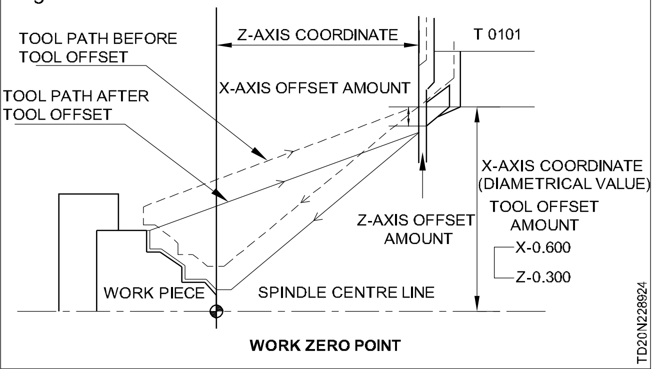
Step 3 I. Select MDI mode.
II. Press PROGRAM button.

Step 4 Enter S500

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.

Fig 4



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

Step 8 After the finish cut, move the tool back in x only. do not 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).

Step 13 Now rotate the spindle in appropriate direction and machine on OD

Step 14 Do not move X.

Step 15 Take Z away from the job.

Step 16 Stop the spindle.

Step 17 Press MENU OFFSET PB.

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- 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 issue S0.

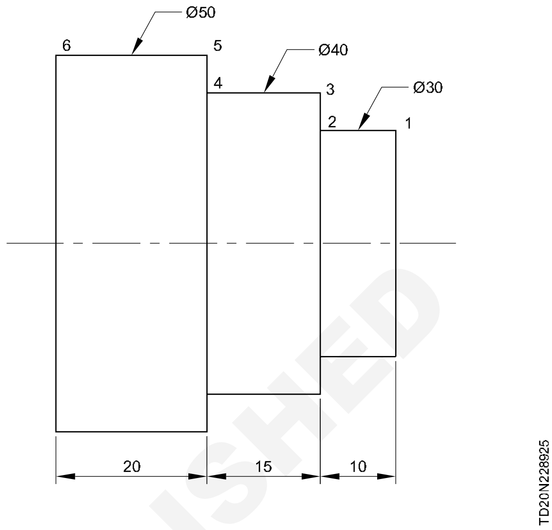
Programming method

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



All the travel commands for tool are mean their co-ordinate 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

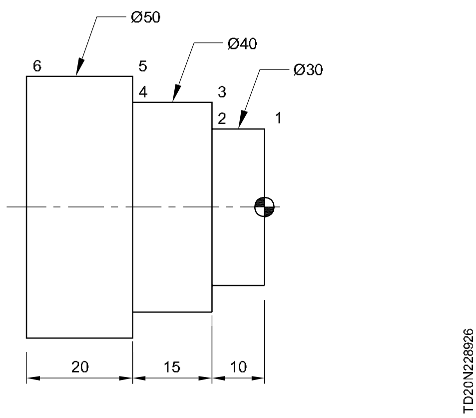
In Fig 5 points 1 to 6 can be specified as follows in absolute dimension programming.

Incremental method (Fig 6)

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

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

Fig 6



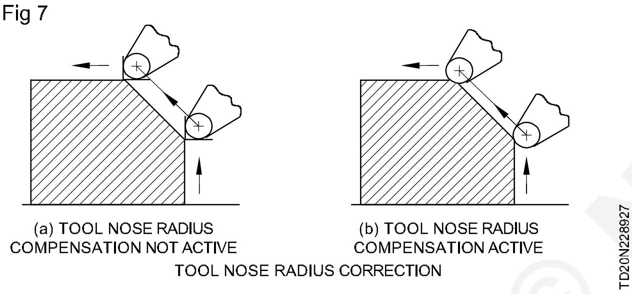
Example

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

Position	U	W
1	30.0	0.0
2	0.0	-10.0
3	10.0	0.0
4	0.0	-15.0
5	10.0	0.0
6	0.0	-20.0

The tool path for finish cutting of a profile can be easily derived by offsetting the nose radius. However, at the beginning and end of inclined path, it is necessary to make calculation based on simple trigonometry for the offset point from the original contour. By using the cutter compensation, the need for all complex calculations will be eliminated. The programming for the finishing cutting will be the direct path of the actual contour to be machined. (Fig 7)

Fig 7

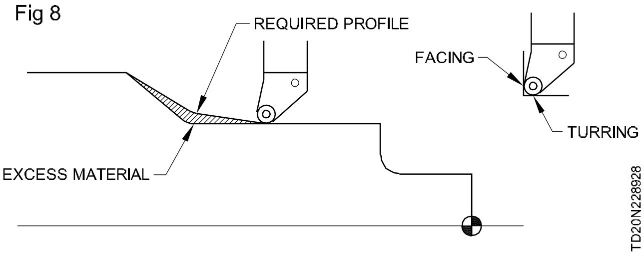


However, even after compensating the nose radius, the point of contact between the tool nose and the work piece will still be along the nose radius periphery which will be changing depending upon the orientation of the tool with respect to the cut surface. For example, the tool will leave a small amount of metal along the inclined surface, when the tool nose radius compensation is not active. For this purpose, the turning centre controllers will provide the necessary correction.

If the correction is active, then the controller automatically compensates and removes the unwanted material. However, in order for the correction to be active, the controller will have to know the correct orientation of the nose radius with respect to the machining surface. For this purpose, nose radius direction is included in the tool-offset registers.

Tool nose radius compensation (Fig 8)

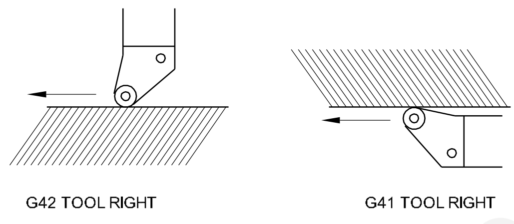
Fig 8



The following data's must be specified to carry out automatic tool nose radius compensation, to obtain required profile exactly.

– Give in the program (finish turning and boring). G41 (Tool Left) or G42 (Tool Right), The position of tool viewing along the traveling direction and G40 (Tool Nose Radius Compensation Cancel). (Fig 9)

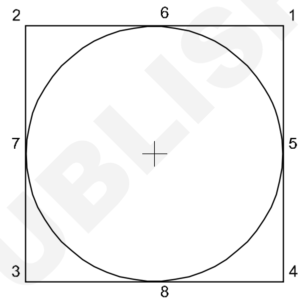
Fig 9



– Input nose radius of tool to R in geometry offset.

– Input the imaginal nose position to T in geometry offset. (Figs 10&11)

Fig 10



Example programmer

Facing operation (Fig 12)

In facing operation the tool moves down wards with parallel to X axis. The depth of cut in Z axis.and material removal in X axis. If work piece diameter is 100.0mm then the program as follows

O01234

G28 U0 W0

T0101

G96 S150

G50 S1500 M03

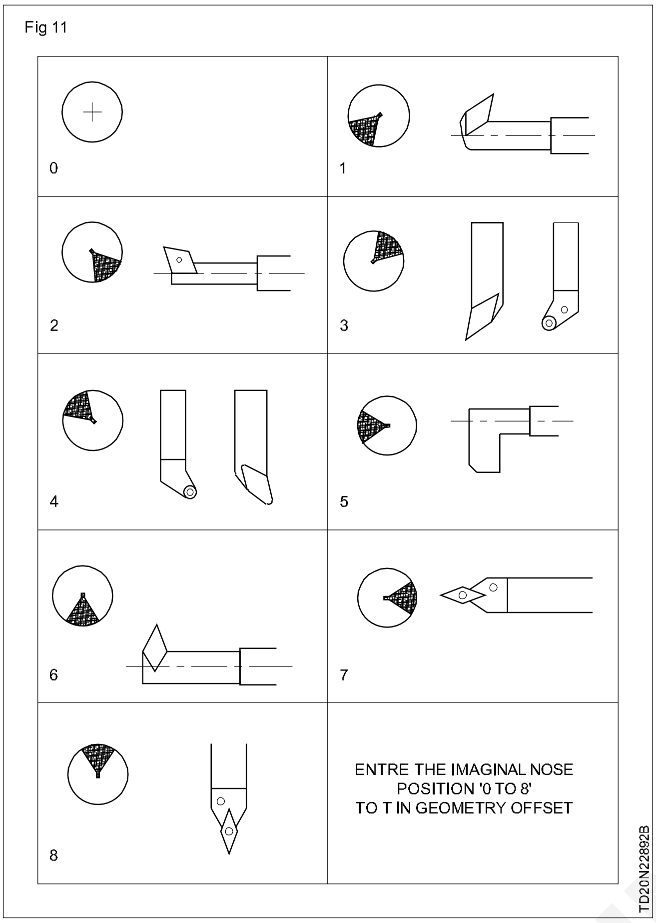
G0 X102.0 Z0.0 M08

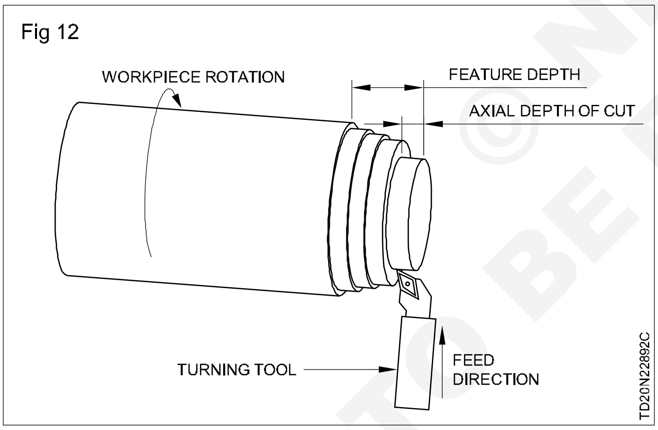
G1 X-1.6 F0.1

G0 X105.0 Z2.0 M09

G28 U0 W0

M30





G71-Profile Turning Cycle (Fig 13)

Format: G71 U__ R__;

G71 P__ Q__ U__ W__ F__;

U - Depth of cut per pass in 'x' axis. (Radial value).

R - Tool relief.

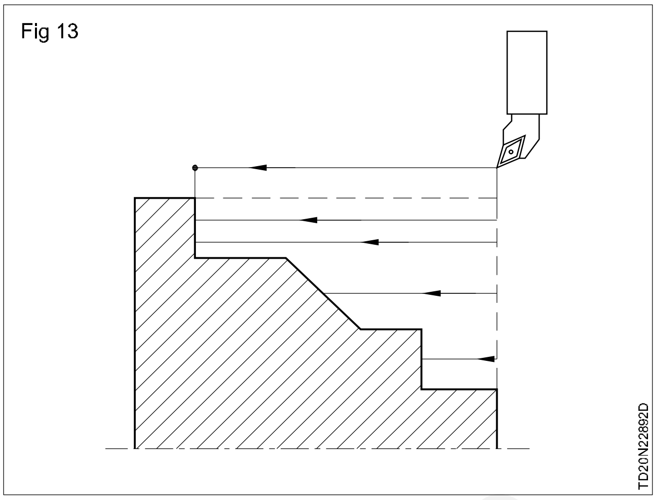
P - Starting block number.

Q - Ending block number.

U - Finishing allowance in 'x' axis.

W - Finishing allowance in 'z' axis.

F - Feed.



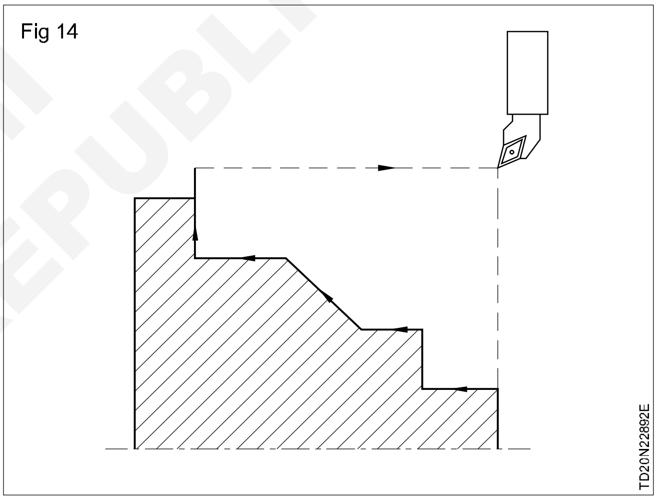
G70 - Finishing Cycle (Fig 14)

Format: G70 P__ Q__ F__;

P - Starting block number.

Q - Ending block number.

F - Feed.



Example: 1 (Fig 15)

O0010;

N10 G28 U0 W0;

N20 T0101;

N25 G96 S200

N30 G50 S1500 M03;

N60 G00 X42.0 Z2.0;

N70 G71 U2.0 R1.0;

N80 G71 P90 Q165 U1.0 W1.0 F0.25;

N90 G01 X16.0;

N100 Z0.0

N110 X20.0 Z-2.0;

N120 Z-10.0;

N130 X28.0;

N140 Z-25.0;

N150 X40.0 Z-31.0;

N160 Z-51.0;

N165 X45.0

N170 G00 X42.0 Z2.0;

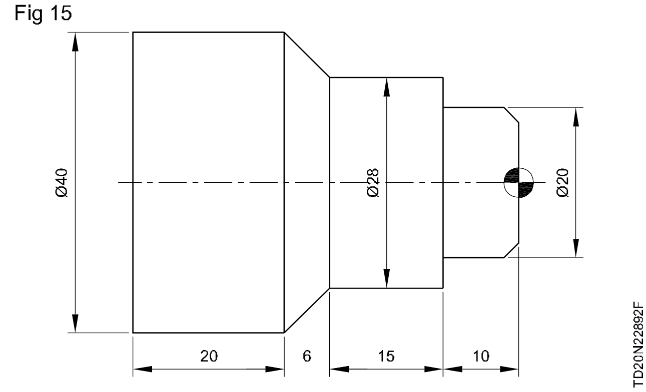
N180 G70 P90 Q165 F0.10;

N190 G00 X100.0 Z50.0 M09;

N200 G28 U0 W0 M05;

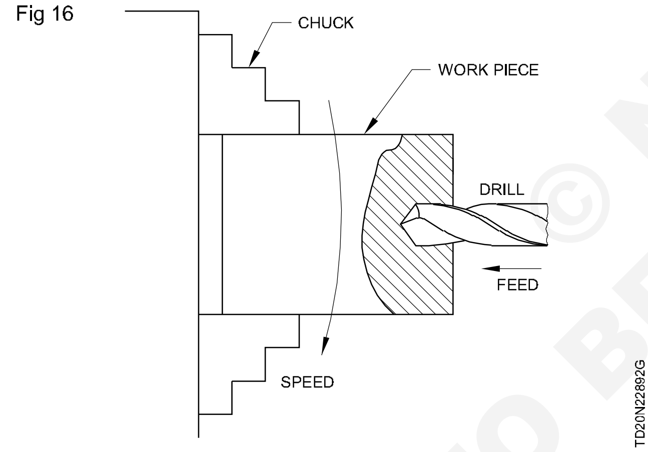
N210 M30;

Fig 15



G74- Peck Drilling Cycle (Fig 16)

Fig 16



Format: G74 R__;

G74 Z__ Q__ F__;

R - Tool relief.

Z - Total depth.

Q - Depth of cut per pass (in microns).

F - Feed.

Example: 1

O0003;

N5 G28 U0 W0;

N10 T0101; (10 mm drill)

N15 G97 S500 M03;

N20 G00 X0.0 Z2.0 M08;

N25 G74 R5.0;

N30 G74 Z-60.0 Q10000 F0.10;

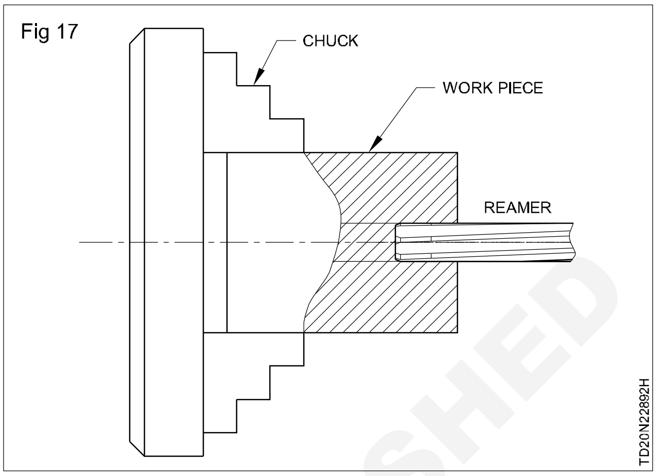
N35 G00 Z3.0 M09;

N40 G28 U0 W0 M05;

N45 M30;

G85-Reaming Cycle (Fig 17)

Fig 17



Format: G85 X__ Z__ R__ F__

O0004;

N5 G28 U0 W0;

N10 T0101; (10 mm drill)

N15 G97 S250 M3;

N20 G00 X0.0 Z2.0 M08;

N25 G85 X0.0 Z-20.0 F0.05;

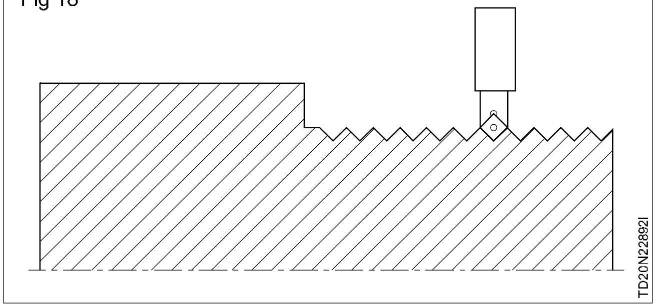
N30 G00 Z10.0 M09;

N35 G28 U0 W0 M05;

N40 M30;

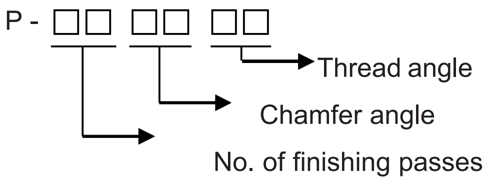
G76 -Thread Cutting Cycle (Fig 18)

Fig 18



Format: G76 P__ Q__ R__;

G76 X__ Z__ P__ Q__ F__;



Q - Minimum depth of cut in microns (Radial value)

R - Finishing depth of cut in microns

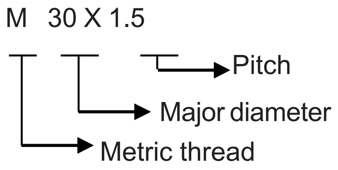
X - Thread diameter Minor dia

Z - Thread length

P - Height of thread in microns

Q - First depth of cut in microns (Radial value)

F - Feed (Pitch value)



Formula to calculate diameter

d = D - 2h

h = 0.649 x P (to find the height of the thread)

P- Pitch value

Example: 1 (Fig 19)

Using formula,

h = 0.649 x P

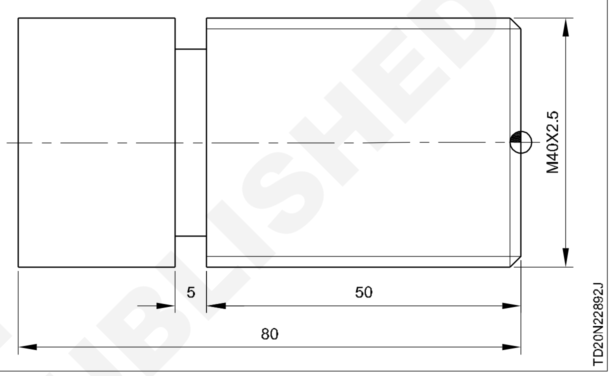
h = 0.649 x 2.5 = 1.6225

d = D - 2h = 40 - (2 x 1.6224)

d = 40 - 3.245 = **36.755**

```
O0005;
N5 G28 U0 W0;
N10 T0606;
N15 G97 S800 M03;
N20 G00 X42.0 Z5.0 M08;
N25 G76 P030060 Q150 R25;
N30 G76 X36.755 Z-50.0 P1623 Q500 F2.5;
N35 G00 X100.0 Z50.0 M09;
N40 G28 U0 W0 M05;
N45 M30;
```

Fig 19



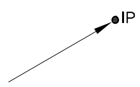
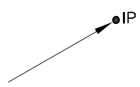
List of function and tape format for fanuc oi-tb

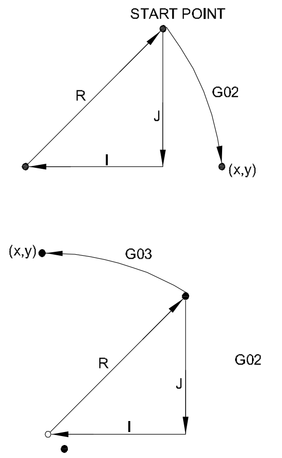
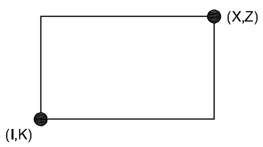
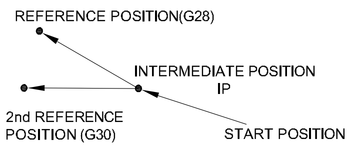
Objectives : At the end of this exercise you shall be able to
- **list the 'G' codes for fanuc oi-tb**
- **describe the formats for fanuc 'G' codes.**

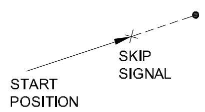
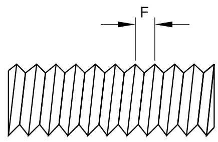
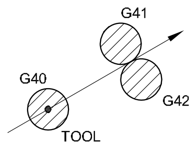
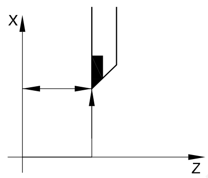
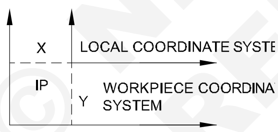
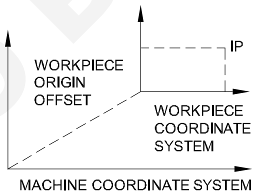
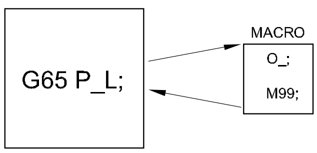
Some functions cannot be added as options depending on the model. In the tables below, IP_:presents a combination of arbitrary axis addresses using X and Z.

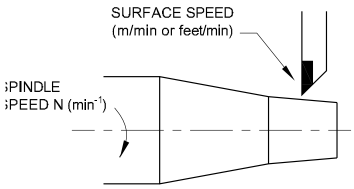
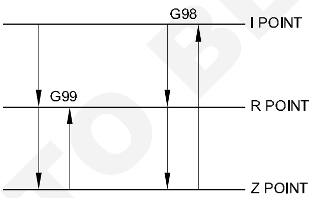
x = 1st basic axis (X usually)

z = 2nd basic axis (Z usually)

Functions	Illustration	Tape format
Positioning (G00)	Start point 	G00 IP_;
Linear interpolation (G01)	Start point 	G01 IP_F_;

Functions	Illustration	Tape format
Circular interpolation (G02, G03)		
Dwell (G04)		$G04\begin{Bmatrix}X\_\\P\_\end{Bmatrix};$
Cylindrical interpolation		G07 IP _r_; Cylindrical interpolation mode G07 IP 0; Cylindrical interpolation mode cancel r: Radius of cylinder
Change of offsetvalue by program (G10)		Tool geometry offset value G10 P_X_Z_R_Q_; P = 1000+Geometry offset number Tool wear offset value G10 P_X_Z_R_Q_; P = Wear offset number
Polar coordinate interpolation (G12,1, G13, 1) (G112, G113)		G12,1 ; Polar coordinate interpolation mode G13,1 ; Polar coordinate interpolation mode Cancel
Plane selection (G18)		G18 ; ZpXp plane selection
Inch/metric conversion (G20, G21)		Inch input : G20 Metric input : G21
Stored stroke check 2,3 (G22, G23)		G22X_Z_I_K_; G23 ;
Reference position return check (G27)	Start position	G27 IP_;
Reference position return (G28) 2nd reference position return (G30)		G28 IP_; G30 IP_;

Functions	Illustration	Tape format
Skip fubction (G31)		G31 IP_F_;
Thread cutting (G32) Cutter compensation		Equal lead thread cutting G32 IP_F_;
(G40, G41, G42)		{G41 / G42} IP_; G40; Cancel
Coordinate system setting spindle speed setting (G50)		G50 IP_; Coordinate system setting G50 S_ ; Spindle speed setting
Workpiece coordinate system preset (G50.3)		G50.3 IP 0 ;
Local coordinate system (G52) Selecting machine		G52 IP _; G53 IP_;
coordinate system (G53) Selecting a workpiece		
coordinate system (G54 to G59)		{G54 : G59} IP_;
Custom macre (G65, G66, G67)		One-shot call G65 P_L_ <argument> ; P : Program number L : Repetition count G66 P _ L_ <argument> ; G67 ; Cancel
Canned cycle (G71 to G76) (G90, G92, G94)	Refer to II.13. Functions to simplify programming	N_G70 P_Q_; G71 U_R_; G71 P_Q_U_W_F_S_T_; G72 W_R_; G72 P_Q_U_W_F_S_T_; G73 U_W_R_; G73 P_Q_U_W_F_S_T_;

Functions	Illustration	Tape format
		G74 R_; G74 X(u)_Z(w)_P_Q_R_F_; G75 R_; G75 X(u)_Z(w)_P_Q_R_F_; G76 P_Q_R_; G76 X(u)_Z(w)_P_Q_R_F_; {G90 / G92} X_Z_I_F_; G94 X_Z_K_F_;
Constant surface speed control (G96/G97)		G96 S_ ; Starts constant surface speed control (Surface speed command) G97 S_ ; Constant surface speed is cancelled (Maximum spindle speed command)
Feed per minute Feed per revolution	mm/min inch/min mm/rev inch/rev	G94 F_; Feed per minute G95 F_; Feed per revolution
Absolute/incremental programming (when G code system A)		X_Z_C_; Absolute programming U_W_H_; Incremental programming (Identified by an address word specified with a G function such as G00 or G01)
Absolute/incremental programming (G90/G91) (during G code system B, C)		G90_; Absolute programming G91_; Incremental programming G90_ G91_; Used together
(G98/G99) (during G code system B,C)		G98_; G99_;

Safety precautions

Objectives: At the end of this lesson you shall be able to
- **list the safety points related to the CNC machining centre**
- **brief the safety points related to the systems.**

Never attempt to machine a workpiece without first checking the operation of the machine. Before starting a production run, ensure that the machine is operating correctly by performing a trial run using, for example, the single block, feedrate override, or machine lock function or by operating the machine with neither a tool nor workpiece mounted. Failure to confirm the correct operation of the machine may result in the machine behaving unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.

Before operating the machine, thoroughly check the entered data. Operating the machine with incorrectly specified data may result in the machine behaving unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.

Ensure that the specified feedrate is appropriate for the intended operation. Generally, for each machine, there is a maximum allowable feedrate. The appropriate feedrate varies with the intended operation. Refer to the manual provided with the machine to determine the maximum allowable feedrate. If a machine is run at other than the correct speed, it may behave unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.

When using a tool compensation function, thoroughly check the direction and amount of compensation. Operating the machine with incorrectly specified data may result in the machine behaving unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.

The parameters for the CNC and PMC are factory–set. Usually, there is not need to change them. When, however, there is not alternative other than to change a parameter, ensure that you fully understand the function of the parameter before making any change. Failure to set a parameter correctly may result in the machine behaving unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.

Immediately after switching on the power, do not touch any of the keys on the MDI panel until the position display or alarm screen appears on the CNC unit. Some of the keys on the MDI panel are dedicated to maintenance or other special operations. Pressing any of these keys may place the CNC unit in other than its normal state. Starting the machine in this state may cause it to behave unexpectedly.

The operator's manual and programming manual supplied with a CNC unit provide an overall description of the machine's functions, including any optional functions. Note that the optional functions will vary from one machine model to another. Therefore, some functions described in the manuals may not actually be available for a particular model. Check the specification of the machine if in doubt.

Some functions may have been implemented at the request of the machine–tool builder. When using such functions, refer to the manual supplied by the machine–tool builder for details of their use and any related cautions.

> **Note: Programs, parameters, and macro variables are stored in nonvolatile memory in the CNC unit. Usually, they are retained even if the power is turned off. Such data may be deleted inadvertently, however, or it may prove necessary to delete all data from nonvolatile memory as part of error recovery.**
>
> **To guard against the occurrence of the above, and assure quick restoration of deleted data, backup all vital data, and keep the backup copy in a safe place.**

Machine and control specification

Objectives: At the end of this lesson you shall be able to
- **read the specification and compare with the machine**
- **interpret the fanuc control system specification.**

Machine specifications

Machine control specification is so important because it provides clear instructions about the Axis working range, machine precision, no. Of tool station, spindle speeds, machine dimensions and the construction feature. A typical specification of a CNC vertical machining centre is given in Table_1.

Table 1

Description	Vertical machining centre	Horizontal machining centre
Number of Axis	3 axes	4 axes
Number of tools	20	36
Table dimensions	780x400 mm	500x500 mm
Maximum travel - x axis	575mm	725mm
Maximum travel - y axis	380 mm	560mm
Maximum travel - z axis	470mm	560mm
Spindle speed	60-8000rpm	40-4000rpm
Spindle taper	BT 40	BT 50
Power	7.0 kw	15.0 kw
Feed rate range	2-5000mm/min	1-5000mm/min
Rapid traverse rate	30 m/min(X,Y), 24 m/min(Z)	30 m/min(X,Y), 24 m/min(Z)
Maximum tool diameter	80mm	105mm
Maximum tool length	300mm	350mm
Maximum tool weight	6kg	10kg

Example Specifications of a CNC System

1	Number of controlled axes	:	Two/Four/Eight, etc.
2	Interpolation	:	Linear/circular/parabolic or cubic/cylindrical
3	Resolution	:	Input resolution (feedback)
		:	Programming resolution
4	Feed rate	:	Feed/min
		:	Feed/revolution

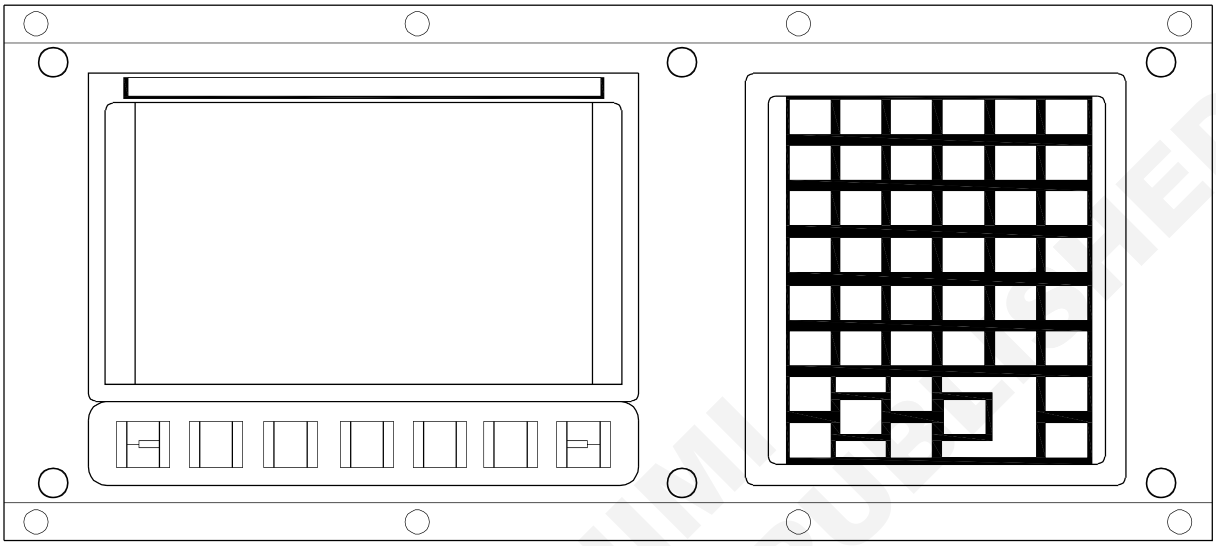
5	Rapid traverse rate	: Feed rate override
		: Feed/min
6	Operating modes	: Manual/Automatic/MDI(editing)/Input/Output/ Machine data set-up/Incremental, etc.
7	Type of feedback	: Digital (rotary encoders with train of pulses)
		: Analog (transducers, etc.)
		: Both
8	Part program handling	: Number of characters which can be stored
		: Part program input devices
		: Output devices
		: Editing of part program
9	Part programming	: Through MDI
		: Graphic simulation
		: Blue print programming
		: Background editing
		: Menu driven programming
		: Conversational programming
10	Compensations	: Backlash
		: Lead screw pitch error
		: Temperature
		: Cutter radius compensation
		: Tool length compensation
11	Programmable logic controller	: Built-in (integrated)/External
		: Type of communication with NC
		: Number of inputs, outputs, timers, counters and flags
		: User memory
		: Program organization
		: Programming Languages
12	Thread cutting/Tapping	: Types of threads that can be cut
13	Spindle control	: Analog/Digital control
		: Spindle orientation
		: Spindle speed overrides
		: RPM/min; constant surface speed
14	Other features	: Inch/metric switchover
		: Polar coordinate inputs
		: Mirror imagin
		: Scaling
		: Coordinate rotation system
		: Custom macros
		: Built-in fixed cycles
		: Background communication
		: Safe zone programming
		: Built-in diagnostics, safety function, etc.
		: Number of universal interfaces
		: Number of active serial interfaces
		: Direct numerical control interface
		: Network interface capability

Machine control function

Objective: At the end of this lesson you shall be able to
- **state the functions of CNC machine controls.**

Machine control functions (Fig 1)

Fig 1



TD20N239211

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.

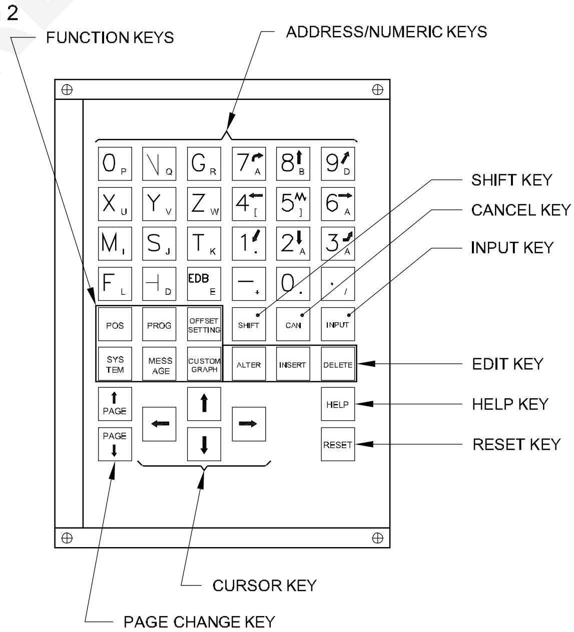
Monochrome

LCD/MDI unit

The layout of the various control keys in the MDI panel are as follows.

MDI key location (Fig 2)

Fig 2



TD20N239212

MDI KEY LOCATION

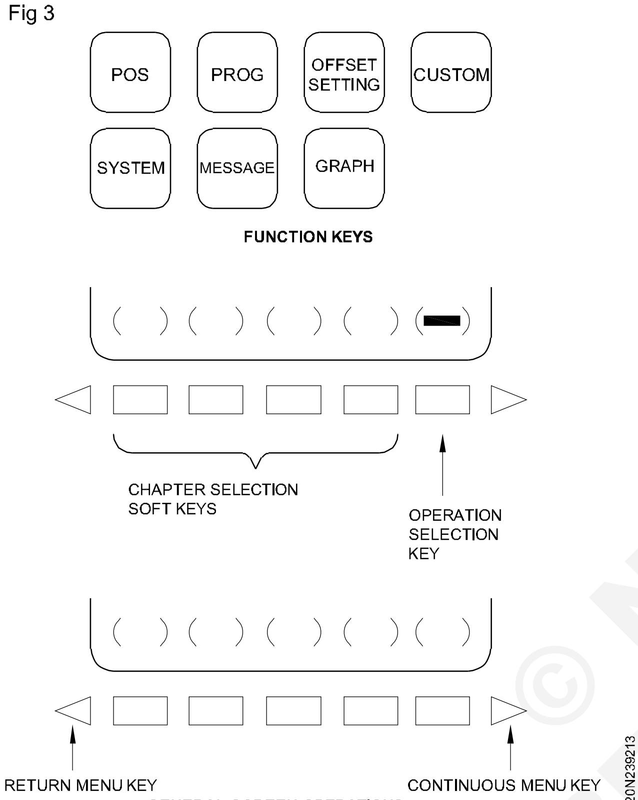
Explanation of the MDI keyboard

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	The soft keys have various functions, according to the applications. The soft key functions 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.

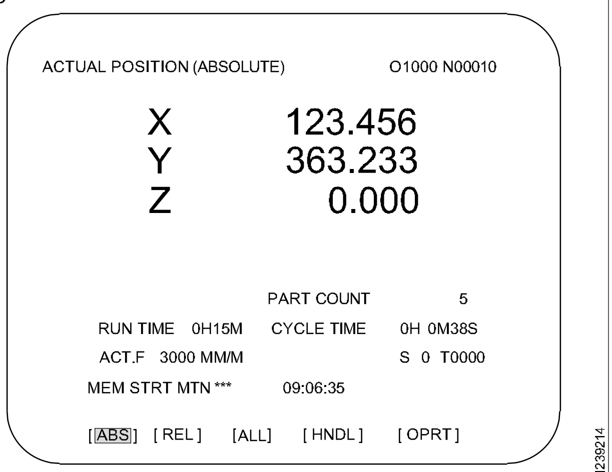
[GRAPH] Press this key to display the graphics screen.

[CUSTOM] Press this key to display the custom screen (conversational acro screen).

Position key

Position display in the work (Absolute) coordinate system (Fig 4)

Fig 4

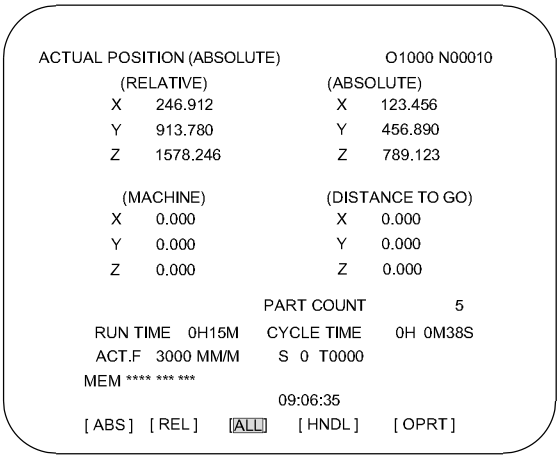


POSITION DISPLAY IN THE WORK (ABSOLUTE) COORDINATE SYSTEM

Position display in the relative coordinate system (Fig 5)

Overall position display

Fig 5



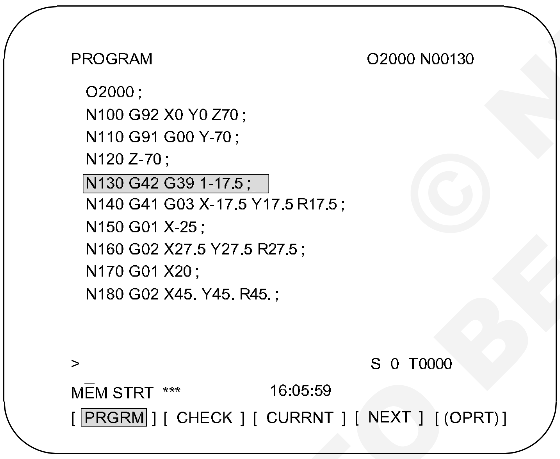
TD20N239215

POSITION DISPLAY IN THE RELATIVE COORDINATE SYSTEM

Program key

Program content display (Fig 6)

Fig 6

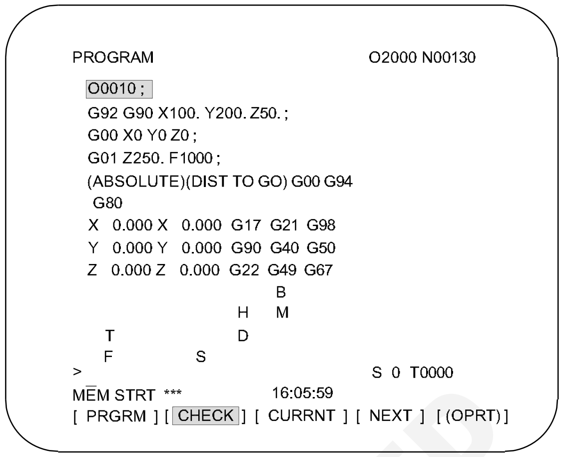


TD20N239216

PROGRAM CONTENTS DISPLAY

Program check screen (Fig 7)

Fig 7



PROGRAM CHECK SCREEN

TD20N239217

Co-ordinate system and point

Objective: At the end of this lesson you shall be able to
- **define different co-ordinates system used in CNC machining centres.**

Co-ordinate system and point

In geometry, a coordinate system is a system that uses one or more numbers, or coordinates, to uniquely determine the position of the points or other geometric elements with respect to a common base point.

Different co-ordinate system

Co-ordinates are specified in one of following two co-ordinate systems.

1 Machine co-ordinate system

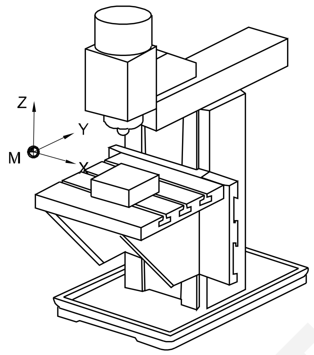
2 Work piece co-ordinate system

1 Machine co-ordinate system (M)

A coordinate system with a machine zero point set as its origin is referred to as a machine coordinate system

When a command is specified the position on a machine coordinate system, the tool moves to the position by rapid traverse. "G53" which is used to select a machine co-ordinate system. It is indicated as - M.

Fig 1



M MACHINE ZERO POINT

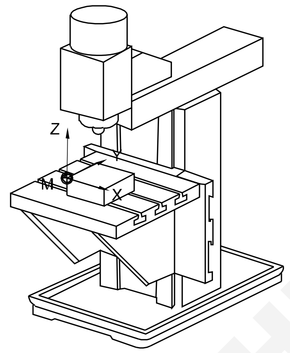
2 Work piece co-ordinate system (W)

A coordinate system used for machining a work piece is referred to as a work piece co-ordinate system. A machining program sets a work piece co-ordinate system. A set work piece co-ordinate system can be changed by shifting its origin. Program commands G54 to G59 can be used to select the work piece co-ordinate system.

Absolute and incremental co-ordinate

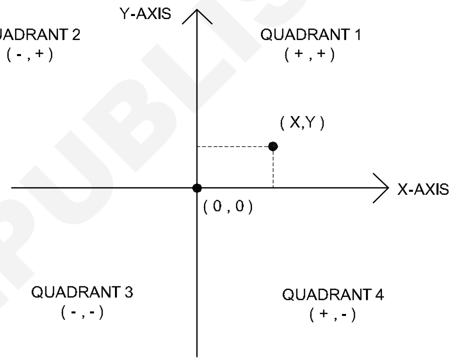
This given sketch explain the co-ordinate location in different quadrant with respect to common zero point (0,0). The value sign changes in different quadrant. It is shown in sketch_1.

Fig 2



W WORK PART ZERO POINT

Fig 3



There are two ways to command travels of the tool; the absolute programming, and the incremental programming.

In the absolute programming (G90), all coordinate value are from common base point is called zero point. In incremental programming (G91) the values are refer with previous position. There is no standard base point.

Absolute programming (G90)

In absolute programming, all measurements are made from the parts origin. Any programmed co-ordinate has the absolute value in respect to the fixed zero point.

Example 1 (G90)

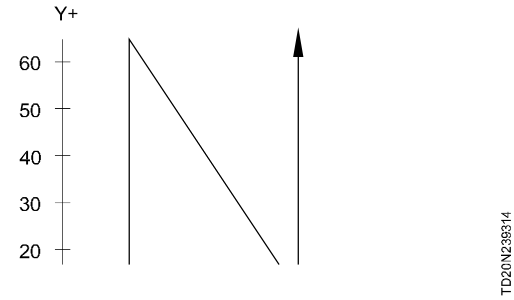
O0001 (ABS);

G90 G54

X20.0 Y10.0;

X20.0 Y60.0;

Fig 4



Incremental programming (G91)

Incremental programming every measurement refers to a previously dimensioned position (point-to-point). Incremental dimensions are the distances between two adjacent points.

Example 2 (G91)

O0002 (INC);

G91 G54

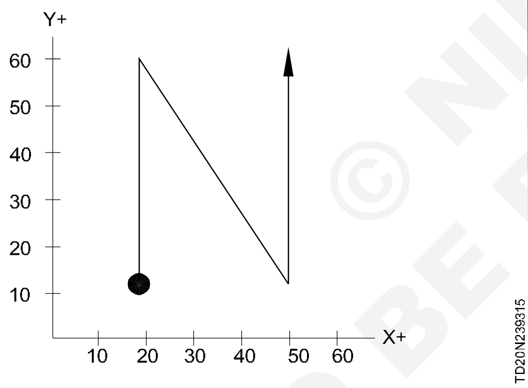
X20.0 Y10.0

X0.0 Y50.0;

X40.0 Y-50.0;

X0.0 Y50.0

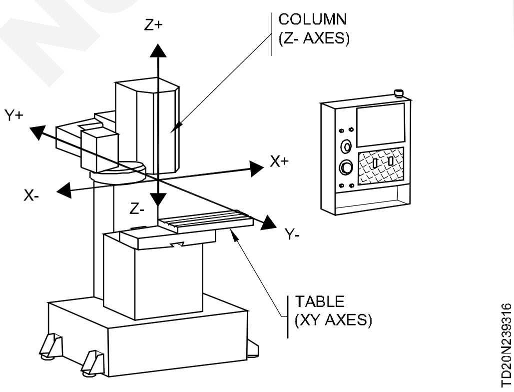
Fig 5



Vertical machining centre

The diagrams of vertical milling machine is shown (sketch-2). It has 3 axis namely X, Y Z. Axis Z is called tooling axis. It moves up (+) and down (-) the sign indicates the direction. Axis X & Y are positioning axis for the specific task. All the axis movements are controlled by servo motor.

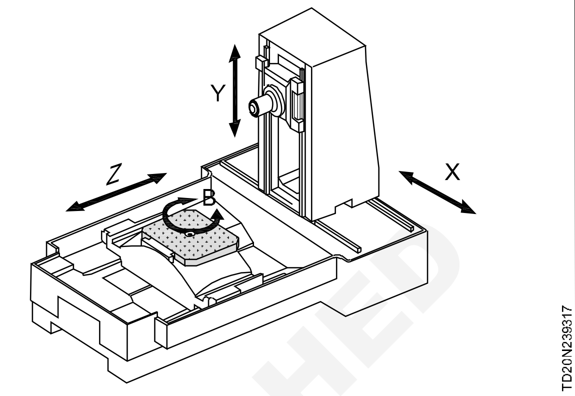
Fig 6



Horizontal machining centre

The horizontal milling machine diagram is shown (sketch-3). It has 3 axis namely X,Y Z. Axis is called tooling axis. It moves from (-) and back (+) the sign indicates the direction. Axis X & Y are positioning axis for the specific task. Axis B is rotational axis. All the axis movements are controlled by servo motor.

Fig 7



All the machine worktable traverses are designed in such a way that they are able to traverse to the target points in coordinate system only. The following are the movements.

Worktable to the right or left (X axis)

Work table upward or down ward (Y axis)

Spindle head forward or backward (Z axis)

Similarly, the rotary movements about the axis of rotation are defined as follows.

Rotation about X axis is A axis.

Rotation about Y axis is B axis.

Rotation about Z axis is C axis.

When a machine slide moves, we always have two alternative i.e. either the tool moves with the spindle head or the work piece moves with the machine table. Which of the two performs the movement, depends on the design of the machine. In CNC concept of relative tool motion makes the programming much simpler, because the programmer does not always have to reconsider which of the two work pieces or tool is actually moving. We can now give each direction of traverse of the machine a name.

A relative tool motion to the right X+

A relative tool motion to the left X-

A relative tool motion to the rear Z+

A relative tool motion to the front Z-

A relative tool motion to the downward Y+

A relative tool motion to the upward Y-

Types of machining centre and its axes

Objectives: At the end of this lesson you shall be able to
- **state the different types of CNC milling machines**
- **explain the axis movement of CNC machines.**

Types of CNC milling machines

Milling machines can divided into three categories

1 by the no of axes

(two, three or more)

2 by the orientation of axes

(vertical or horizontal)

3 by the presence or absence of a tool changer.

The spindle motion is up and down in vertical milling/ machining centre.

The spindle motion is in and out in horizontal milling/ machining centre.

These machines are capable to perform the following operations:

Drilling, Reaming, boring, tapping, profiling, thread cutting and many other operations.

ATC: Automatic Tool changer.

APC: Automatic pallet changer

CNC: Computer Numerical Control

With the above advanced featured built in milling machines become the new breed of machine tools called machining centres.

Machine axes

The machining centres are provided with minimum threee axes of 'X','Y'&'Z' axis of they have fourth axis machines become more flexible i.e the fourthy axis 'A' for vertical model and 'B' of horizontal model. The machine with fine or more axes is of higher level of capacity.

In aircraft industry 5 axes profile milling machine is used for complex shapes and reach cavaties and various angles.

Meaning of half / full axis in NC language (what is half / fall axis machine)

A four axis vertical machine has X,Y,Z as primary axes and indexing table designated as 'A' axis which can position but cannot rotate simultnesously is called 3 1/2 axes machine. If the maching equipped with full rotating table, simultaniously then it is called four axis machine tool.

In the milling systems, three most common machine tools are

CNC vertical machining centre - VMC

CNC horizontal machining centre - HMC

CNC horizontal boring mill

Vertical Machining Centres (Fig 1)

VMC is for flat type of work where the machining is done on only one face of the part in single set up.

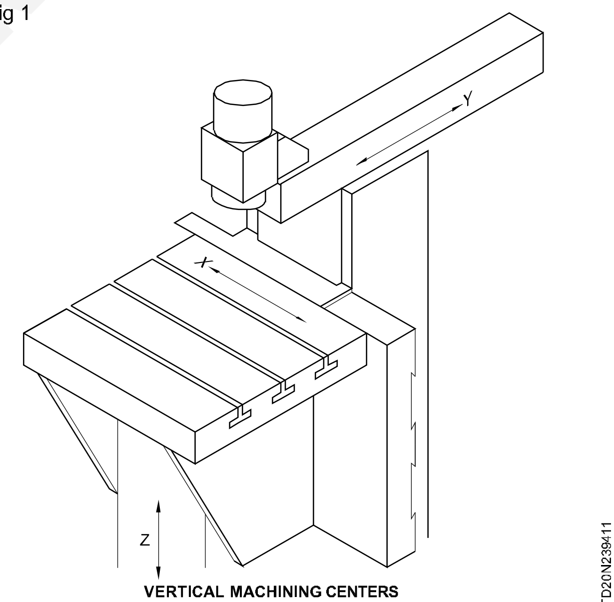
An optional fourth axis can be provided by mounting rotary can the main table either vertically or horizontally depending on the desired results and the model type.

In the combination with a tailstock (usually supplied) the fourth axis in the vertical configuration can be used for machining long parts. That need support at both ends.

For programming two systems followed. One programming always takes place form the view point of the spindle, not the operators eye. View is as if looking straight down at 90^{o} towards the machine take for development of the tool motion.

Two various markers located some where in the machine show the positive and negative motion of the machine axes. For programming these markers should be ingnored. The programming directions are exactly the opposite of the markers on the machine tool.

Fig 1



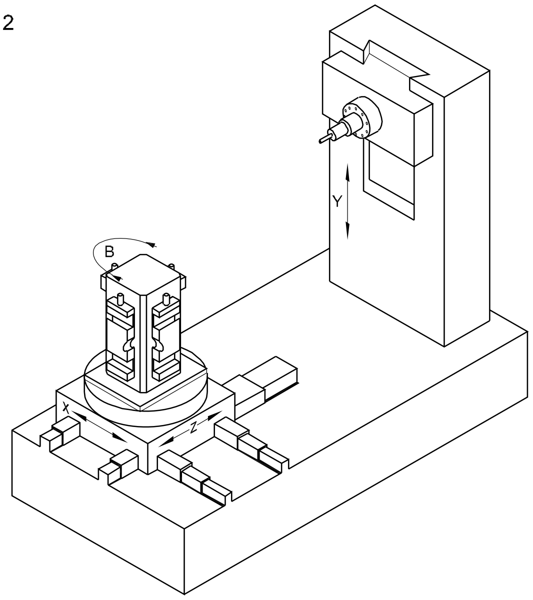
VERTICAL MACHINING CENTERS

Horizontal machining centers (Fig 2)

Horizontal CNC machining centres are also categorized as multi - tool and versatile machines, and are used for cubical parts, where the majority of machining has to be done on more than one face in a single setup.

There are many applications in this area. Common examples are large parts, such as pump housings, gear cases, manifolds, engine blocks and so on. Horizontal machining centres always include a special indexing table and are typically equipped with a pallet changer and other features.

Fig 2



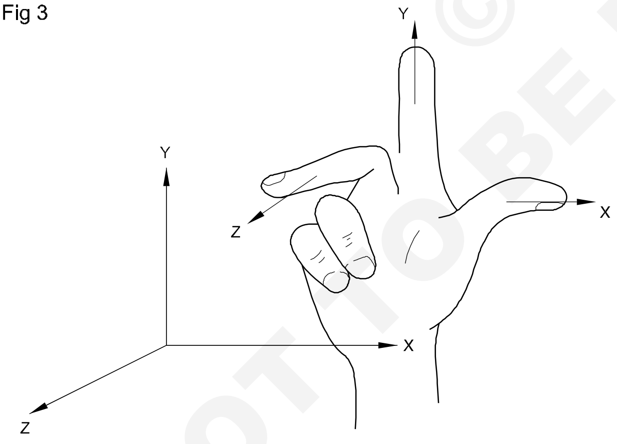
HMC SHOWING THE AXIS DIRECTIONS

TD20N239412

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 3



RIGHT HAND THUMB RULE

TD20N239413

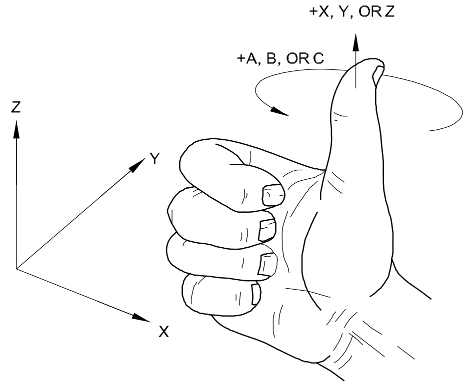
Auxillary axes on NC machine

Apart from each side movement axes on the machine, some other auxillary 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 definite the positive direction of the coordinate axes as per the following Fig 4.

Fig 4



RIGHT HAND RULE

TD20N239414

Z - axis (Fig 5)

The axis of the main (i.e. principle) machine spindle, whether it be the axis of the tool spindle or the axis about which the work piece rotates, is denoted as the Z axis. On machine tools, which do not possess principle spindle (e.g. planning machines) the Z - axis is perpendicular to the work holding surface.

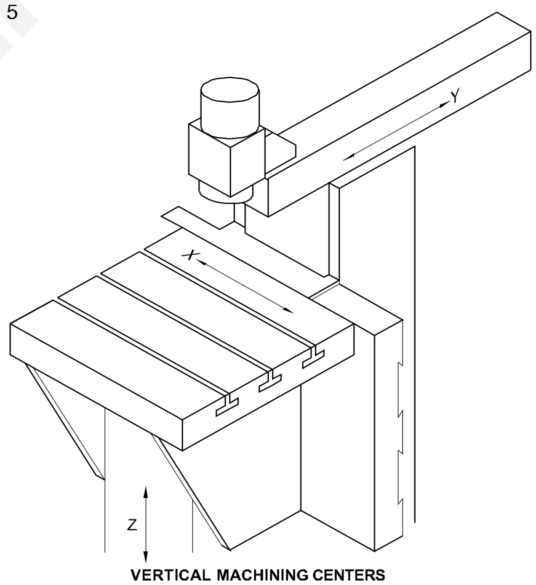
X - axis

The X - axis is always horizontal, parallel to the work holding surface and perpendicular to the z - axis.

Y - axis

The Y - axis is perpendicular to both Z and X axis.

Fig 5



VERTICAL MACHINING CENTERS

TD20N239415

Milling tool coordinate system (Fig 6)

Classification of machines.

CNC machines can be classified into various ways,

a) According to no.of axis

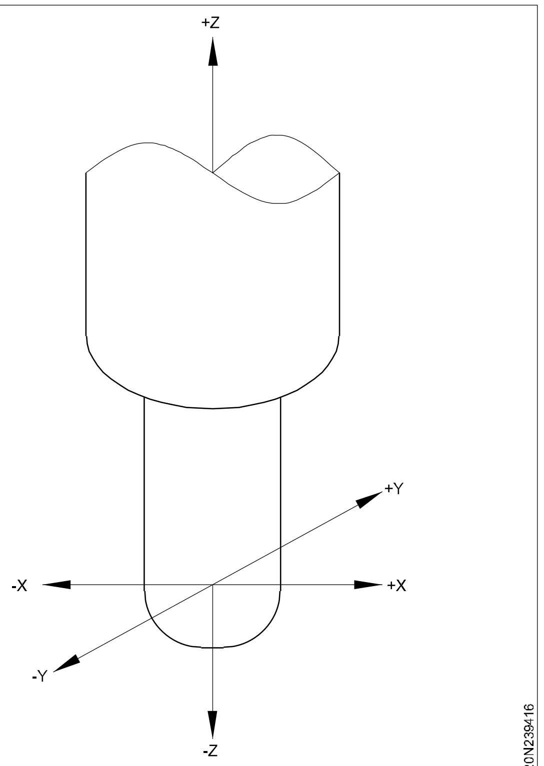
CNC machine can be classified as

2 axis machine

3 axis machine

4 axis machine

Fig 6



MILLING TOOL COORDINATE SYSTEM

It should be noted that each axis has it own drive motor.

b) According to CNC system

There are three types of CNC systems based on their capability in providing feed in different axes.

1 Point -to- point control

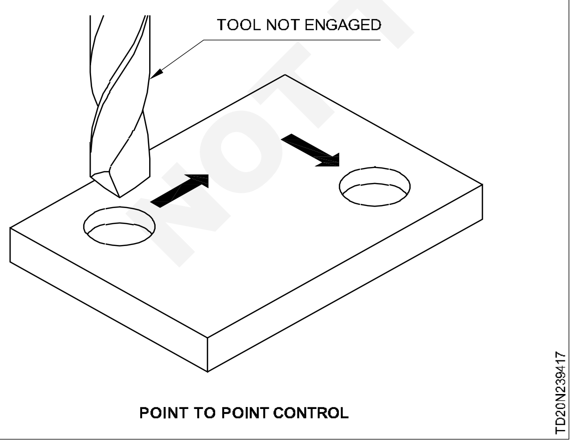
2 Straight cut control

3 Contouring control

Point-to-point control

Machines with point-to-point control provide only one feed axis while the other two axes can perform only rapid motion. Machines with point-to-points control are suited only for drilling operations. (Fig 7)

Fig 7

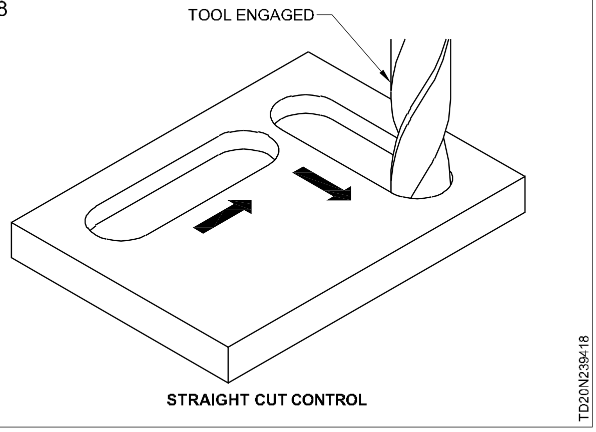


POINT TO POINT CONTROL

Straight cut control

The system provides feed motion in two axes (but not simultaneously) and hence their capability is limited to performing milling either along X axis or along Y axis. (Fig 8)

Fig 8



STRAIGHT CUT CONTROL

Contouring control

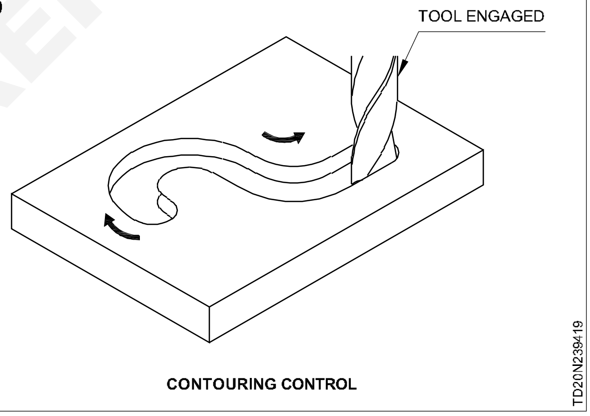
This can provide feed control in 3 axes. They are also capable of providing simultaneous feed in 2 or 3 axis.

Milling machine with contouring control can mill contours made up of straight lines and arc/circular elements.. Depending on the number of axes that can be simultaneously fed, contouring controls are further classified as 2D control, 2 1/2D control and 3D control.

i) 2D control

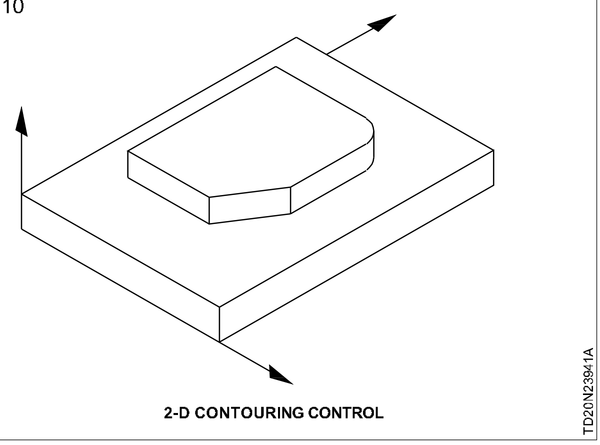
Machines with 2D control can be simultaneous feed only in two of the 3 axes. They can mill only contours with constant depth that too in just one plane (X-Y) (Fig 9 & 10)

Fig 9



CONTOURING CONTROL

Fig 10

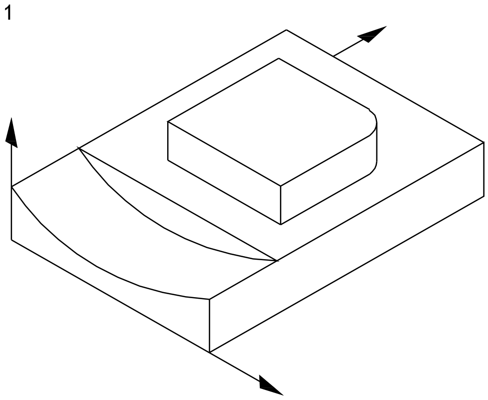


2-D CONTOURING CONTROL

ii) 2 1/2D control

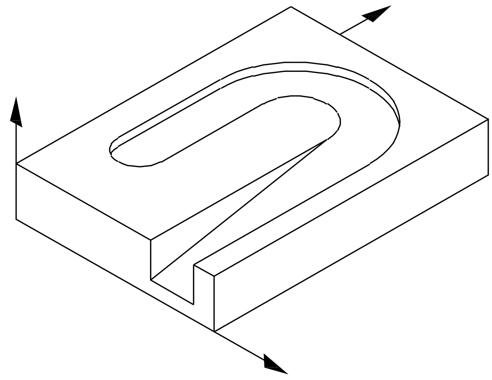
Machines with 2 1/2D control can have simultaneous feed of any of the two axis X-Y, X-Z,Y,Z and, hence they can mill contours (of constant depth) in any one of the 3 plances. (Figs 11 & 12)

Fig 11



$2\frac{1}{2}$-D CONTOURING CONTROL

Fig 12



3-D CONTOURING CONTROL

Programming

Objectives: At the end of this lesson you shall be able to
- **state the method of entering data to a CNC machine**
- **state various alpha numerical commands**
- **explain a programme block**
- **state tool path checking through simulation.**

A CNC program is a step-by-step instruction that tell the machine as to what action it has to take for machining a part.

Programme are initially written on paper using codes consisting of alphabets and numbers.

Before we can machine the part, the completed programme has to be fed into the memory of the machine computer and this can be done in different ways.

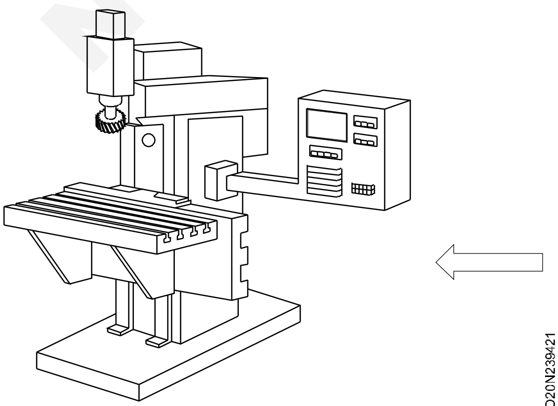
But the most easily available method is to enter the codes direct into the machine using the keys/buttons provided on the control panel.

Data entering

Data entering can be done in 2 ways:

By key board typing on the CNC milling machine (Fig 1)

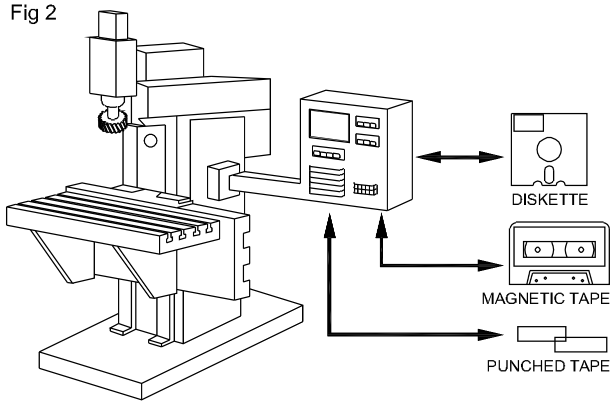
Fig 1



Entered data will be stored temporally, and be destroyed if there is any interruption of power supply

By re-entering of stored data (Fig 2)

Fig 2



Entered data is stored permanently and saved even there is an interruption of the power supply.

Programme codes

The alpha numeric codes used for programme writing are. in a way, machine-dependent because the codes may vary from machine to machine. The codes pertaining to a particular make of machine are available in the machine manual and it should be thoroughly studied before writing/ reading a programme.

However, ISO standards are available for CNC programming unless otherwise stated, all programming codes discussed in this book will follow ISO codes/Sinumeric system.

Few codes are,

G00

G01

M1

M2

T2

F90 etc.

Programme block and programme word

A line in the programme is called a block and the individual codes in the blocks are called programme words. The alphapet in these words are called address letters.

Example

N1 G00 X100 Y10 Z20

Block structure

The term block structure/format refers to the rules for writing a programme block. In otherwords it refers to the essential programme words/codes that should be written along with certain commands.

The first word of any block is the block identification number and they are formed by the letter N.

Thus N1, N2, N10, N20 etc are block numbers and they do not carry any specific instructions to the machine.

Block numbers can be written serially (N1, N2, N3, N4, etc) or in suitable increments like N10, N20, N30, etc)

Layout of a CNC-Program

A CNC-Program, also known as part program, consists of a logical sequence of commands, which are executed step-by-step the control unit after the program has been started.

Each program is compiled and stored under a program name in the control unit. The name can contain letters as well as numbers.

A block starts with a block number followed by the commands.

Each command consists of command words, which in turn consist of an address letter (A-Z) and an associated numerical value. Both upper or lower case characters are permissible)

Program layout

The block number is a program-technical assignment which is not evaluated by the control unit as a command. It is usually programmed to go up in steps of 10 and serves only the user for better oversight. It has no effect on the program execution.

The geometrical data include all instructions that clearly define mathematically the motion of the tool or the axes.

The technological data are used for instance to activate the required tool and to re-select the necessary cutting parameters feed rate and spindle speed. Miscellaneous functions can control for example such things as direction of rotation and auxiliary appliances. (Table 1)

Programming example

....

N80 T1; Roughing tool

N90 M6

N100 G54 F0.2 S180 M4

N110 G00 X20 Y0Z2 D1

N120 ....

In order to improve the oversight within a program commentaries can be optionally added at the end of a block. These must be preceded by a semicolon(;). Any characters that follow thereafter will not be taken account of by the control unit..

Simulation

Mechanical parts with free form surfaces used in various industries (moulds, automotive, aerospace, etc...) are machined on multi-axis CNC milling machines because of their highly complex geometric shapes. Toolpaths for obtaining these parts are generated by taking into account several parameters (cutting conditions, tools shapes, surface models, etc...)

The final shape of the part is obtained in three operations:roughing, semi-finishing and finishing. Before real machining. it is essential to simulate virtually the machining to verify the toolpaths geometry of the finished part and to predict physical factors that are necessary to optimize the cutting parameters.

During simulation it is necessary to lock all the machine axis and auxiliary functions.

Table 1

	Departure information(Geometrical data)							Switching information(Technological data)			
Block Nr...	Auxillary command	Co-ordinate axes			Interpolation parameter			Feed	Speed	Tool	Misc. function
N	G	X	Y	Z	I	J	K	F	S	T	M

Address codes

Objectives: At the end of this lesson you shall be able to
- **describe the list of address characters applied in CNC machine tools**
- **explain the features of the address characters.**

The lists of address characters as per DIN 66025 are as follows

Word address	Description
A	Rotation about X axis
B	Rotation about Y axis
C	Rotation about Z axis
D & E	Rotation about addition axes
F	Feed rate
G	Preparatory function
H	Tool height offset
I	Circular interpolation parameter or thread pitch parallel to X axis
J	Circular interpolation parameter or thread pitch parallel to Y axis
K	Circular interpolation parameter or thread pitch parallel to Z axis
L	Number of repetitions
M	Miscellaneous function
N	Block number
O	Program number
P,Q &R	Third movement parallel to X,Y and Z axes respectively
S	Spindle speed
T	Tool function
U,V &W	Second movement parallel to X, Y and Z axes
X	Movement in X axis
Y	Movement in Y axis
Z	Movement in Z axis

The word address format is based on a combination of one letter or more digits.

Character,

A smallest unit in CNC programming. It can be a letter, digit or symbol. Example : X,Y,Z, 1-9,+,-, etc.

X	-	3	.	0

Word,

Word is a combination of character. It consists of a capital letter, followed by a number, and sometime symbol, depending on the code. Example : N5, G01, F100, G91

N	5	G	0	1	Z	-	3	2	9	.	5

Block,

Block is a multiple of words seperated by end of block, otherwise each line in the programme is a block.

N	1	0	G	0	2	X	3	9.7	Z	-	8	7.	R	.	2	F	.	5

Alphabetical address codes

The following is a list of the address codes used in programming the HAAS milling machines.

A → Fourth axis rotary motion

The A address character is used to specify motion for the optional fourth, A axis.

B → Fifth axis rotary motion

The B address character is used to specify motion for the optional fifth, B axis.

C → Auxiliary external rotary axis

The C address character is used to specify motion for the optional external sixth, C axis.

D → Tool diameter offset selection

The D address character is used to select the tool diameter or radius used for cutter compensation. The number following must be between 0 and 200. The Dnn selects that number offset register that is in the offset display, which contains the tool diameter/radius offset amount when using cutter compensation (G41 or G42). D00 will cancel cutter compensation so that the tool size is zero and it will cancel any previously defined Dnn.

E → Engraving feed rate or continuous accuracy

The E address character is used, with G187, to select the accuracy required when cutting a corner during high speed machining operations. The range of values possible is 0.0001 to 0.25 for the E code.

F → Feed rate

The F address character is used to select the feed rate applied to any interpolation functions, including pocket milling and canned cycles. It is either in inches per minute with four fractional positions or mm per minute with three fractional positions. (It is recommended that the programmer always use a decimal point)

G → Preparatory functions (G codes)

The G address character is used to specify the type of operation to occur in the block containing the G code. The G is followed by a two or three digit number between 0 and 187. Each G code defined in this control is part of a group of G codes. The group 0 codes are non model that is they specify a function applicable to this block only and do not affect other blocks. The other groups are model and the specification of one code in the group cancels the previous code applicable from that group. A model G code applies to all subsequent blocks so those blocks do not need to re - specify the same G code. More than one G code can be placed in a block in order to specify all of the setup conditions for an operation.

H → Tool length offset selection

The H address character is used to select the tool length offset entry from the offsets memory. The H is followed by a two digit number between 0 and 200. H0 will clear any tool length offset and Hnn will use the tool length entered in on 'n' from the offset display. You must select either G43 or G44 to activate a tool length (H) offsets. The G49 command is the default condition and this command will clear any tool length offsets. A G28, M30 or pressing reset will also cancel tool length offsets.

I → Circular interpolation or canned cycle data

The I address character is used to specify data for either canned cycles or circular motions. It is defined in inches with four fractional positions or mm with three fractional positions.

J → Circular interpolation or canned cycle data

The J address character is used to specify data for either canned cycles or circular motions. It is defined in inches with four fractional positions or mm with three fractional positions.

K → Circular interpolation or canned cycle data

The K address character is used to specify data for either canned cycles or circular motions. It is defined in inches with four fractional positions or mm with three fractional positions.

L → Loop count to repeat a command line

The L address character is used to specify a repeat count for some canned cycles and auxiliary functions. It is followed by a number between 0 and 99999.

M → Code miscellaneous functions

The M address character is used to specify an M code. These codes are used to control miscellaneous machine functions. Note that only one M code is allowed per block in a CNC program and all M codes are performed secondary in a block.

N → Number of block

The N address character is entirelyoptional. It can be used to identify or number each block of a program. It is followed by a number between 0 and 99999. The M97 functions needs to reference an N line number.

O → Program number (Program name in parenthesis)

The O address character is used to identify a program. It is followed by a number between 0 and 99999. A program saved in memory always has a 0nnnnn identification in the first block. Altering the 0nnnnn in the first block causes the program to be renumbered. If you enter a program name (program text name) between parentheses in the first three lines of a program, that program name will also be seen in your list of programs. You can have up to 500 program numbers (200 programs on an older machine) in your list of programs. You can delete a program number from the list prog display, by cursor selecting the program, and pressing the erase prog key. You can also delete a program in the advanced editor using the menu item delete program from list.

P → Delay of time or M98 program number call or M97sequence number call

The P address character is used for either a dwell time in seconds with some canned cycles or in a G04, or P is used as a program number reference for an M98, or a program sequence number reference for an M97. The P value is a positive number without decimal point up to 99999 when used with an M98 or M97. When used as a dwell time, it may be a positive decimal number between 0.001 and 1000.0 in seconds.

Q → Canned cycle optional data

The Q address character is used in canned cycles and is always a positive number in mm between 0.001 mm to 1.0 mm.

R → Circular interpolation or canned cycle data

The R address character is used in canned cycles or circular interpolation. It's either in inches with four fractional positions or mm with three fractional positions. It is followed by number in inches or metric. It's usually used to define the reference plane for canned cycles.

S → Spindle speed command

The S address character is used to specify the spindle speed in conjunction with M41 and M42. The S is followed by an unsigned number between 1 - 99999. The S command doesn't turn the spindle ON or OFF; it only sets the desired speed. If a gear change is required in order to set the commanded speed, this command will cause a gear change to occur. Even if the spindle is stopped. If spindle is running, a gear change operation will occur and the spindle will start running at the new speed.

T → Tool selection code

The T address character is used to select the tool for the next tool change. The number following must be a positive number between 1 and (20) the number in parameter 65. It does not cause the tool change operation to occur. The Tnn may be placed in the same block that starts tool change (M06 or M16) or in any previous block.

U → Auxiliary external linear axis

The U address character is used to specify motion for the optional external linear, U - axis.

V → Auxiliary external linear axis

The V address character is used to specify motion for the optional external linear V - axis.

W → Auxiliary external linear axis

The W address character is used to specify motion for the optional external linear, W - axis.

X → Linear X - axis motion

The X address character is used to specify motion for the X - axis. It specifies a position or distance along the X - axis. It is either in inches with four fractional positions or mm with three fractional positions. It is followed by a singed number in inches or metric. If no decimal point is entered the last digit is assumed to be 1/10000 inches or 1/1000 mm.

Y → Linear Y - axis motion

The Y address character is used to specify motion for the Y- axis. It specifies a position or distance along the Y - axis. It is either in inches with four fractional positions or mm with three fractional positions. It is followed by a signed number in inches or metric. If no decimal point is entered, the last digit is assumed to be 1/10000 inches or 1/ 1000mm.

Z → Linear Z - axis motion

The Z address character is used to specify motion for the Z - axis. It specifies a position or distance along the Z - axis. It is either in inches with four fractional positions or mm with three fractional positions. It is followed by a signed number in inches or metric. If no decimal point is entered, the last digit is assumed to be 1/10000 inches or 1/1000 mm.

Canned cycle

Objectives : At the end of this lesson you shall be able to
- **state what is canned cycle**
- **list the different types of canned cycles, and their function.**

In canned cycle frequently used machining operations are specified with a G function in a single block. Canned cycles made easier for the programmer to create short program.

The different types of canned cycles are listed in table 1.

Table 1

Canned cycle used in OMC

G-code	Application	Z-direction	Operation at the bottom of the hole	Retraction in Z direction
G73	High speed peck drilling Cycle	Intermittent feed	Dwell-spindle rotates CW spindle orientation Tool tip moved opposite to cutting & retracted	Feed
G76	Fine boring	Feed	----	----
G80	Canceling of G81-G89 Cycles	----	----	----
G81	Center/spot drilling	Feed	----	----
G82	Counter bore/counter sink	Feed	Dwell	Rapid traverse
G83	Peck/deep hole drilling	Intermittent feed	----	Rapid traverse
G84	Tapping cycle	Feed	Dwell spindle CW	Feed

G85	Boring Cycle	Feed	------	Feed
G86	Boring cycle	Feed	Spindle stop	Rapid traverse
G87	Back boring cycle	Feed	Spindle CW	Rapid traverse
G88	Boring cycle	Feed	Dwell-spindle stop	Manual retract
G89	Boring	Feed	Dwell	Feed

A canned cycle consists of six operations only, shown in Fig 1. They are

Operation 1 - Positioning of x, y, x.

Operation 2 - Rapid position up-to R level.

Operation 3 - Hole machining.

Operation 4 - Operation at the bottom of a hole.

Operation 5 - Return to R point level.

Operation 6 - Rapid traverse to the initial point.

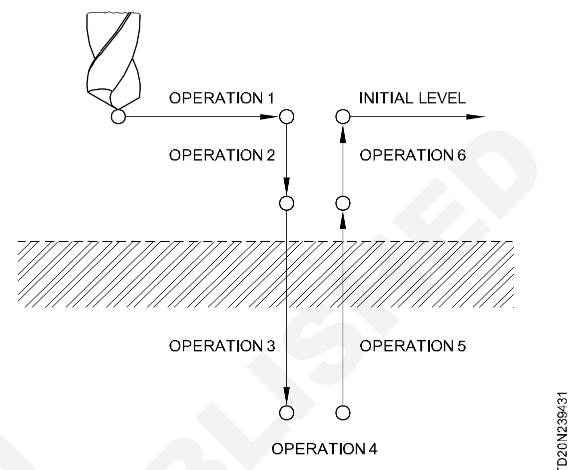
Important restrictions

1 Drilling is not performed when cycle call block does not contain X,Y,Z.R or any other axis.

2 Specify Q and/or R in blocks that perform drilling. If they are specified in a block that does not contain drilling they cannot be stored as modal data,.

3 In a cycle call block do not specify 01 Group G codes(G00 to G03). This will cancel the canned cycle G-codes.

Fig 1



4 In the canned cycle mode tool offsets are IGNORED.

5 The canned cycle call should be cancelled after the last drill point preferably with reference point return.

Cutter radius compensation (CRC)

Objectives: At the end of this lesson you shall be able to
- **state the significance of the term CRC**
- **explain the use of codes G42, G41 and G40.**

Usually the tool reference point being at the centre of the cutter, it will be the centre of the cutter that moves along the programmed path. But in a profiling operation, it is the periphery of the cutter that should travel along the programmed path and it is possible only if the centre of the cutter offsets itself from the programmed path by a distance equal to the cutter radius.

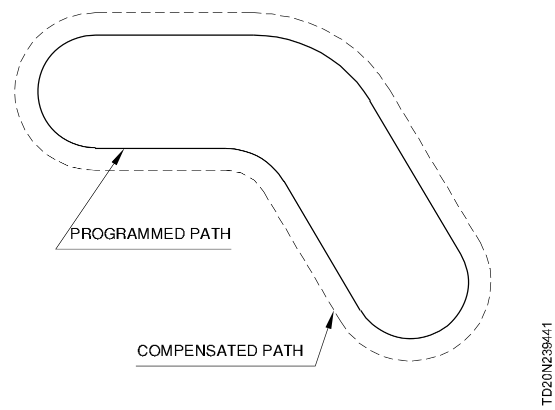
The term CRC signifies that the centre of the cutter should take a path away from the programmed path by a distance equal to the radius of the cutter. Such deviated path taken by the cutter centre is called compensated path. (Fig 1)

CRC commands (G41 and G42)

These commands instruct the control that the cutter centre should take a compensated path. These commands can be effective only if the control knows the radius of the cutter. For this purpose the radius of the cutter is stored in the tool data memory along with the length of the cutter. In the programme tool data memory is referred to by the D command.

In motion programme the command G41 or G42 is put in the block that takes the tool to the starting point of the profile. (Fig 2)

Fig 1



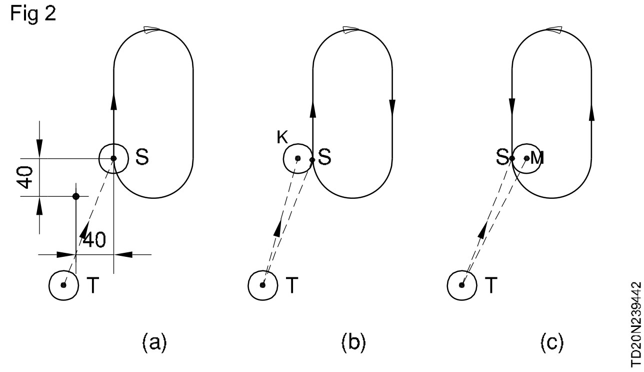
In Fig 2, the starting point of profile is marked 'S' and its X, Y coordinates are 40 and 40.

Difference between G41 and G42

If we write a block as 'G00 x 40 Y40' without either G41 and G42, the tool takes the path TS (Fig 2a) and the cutter will reach the point S, so that its centre coincides with S.

But if we add G41 in the block and write the block as G00 G41 x 40 Y40, the tool will take the path TK (Fig 2b) in such a way that the tool is on the left side of the profile with its periphery touching the point 'S'. Here TK is the compensated path.

Fig 2



If we write the block with G42 in place of G41, (G00 G42 X60 Y40) the compensated path will be TM such that the tool will be on the right side of the profile. (Fig 2C)

It may then be concluded that code G41 makes the compensated path to the left side while G42 makes the compensated path to the right side of the programmed path. The left/right has its reference to the direction of the cutter path. (See arrows in Fig 2a,b & c).

Cancellation of CRC G40

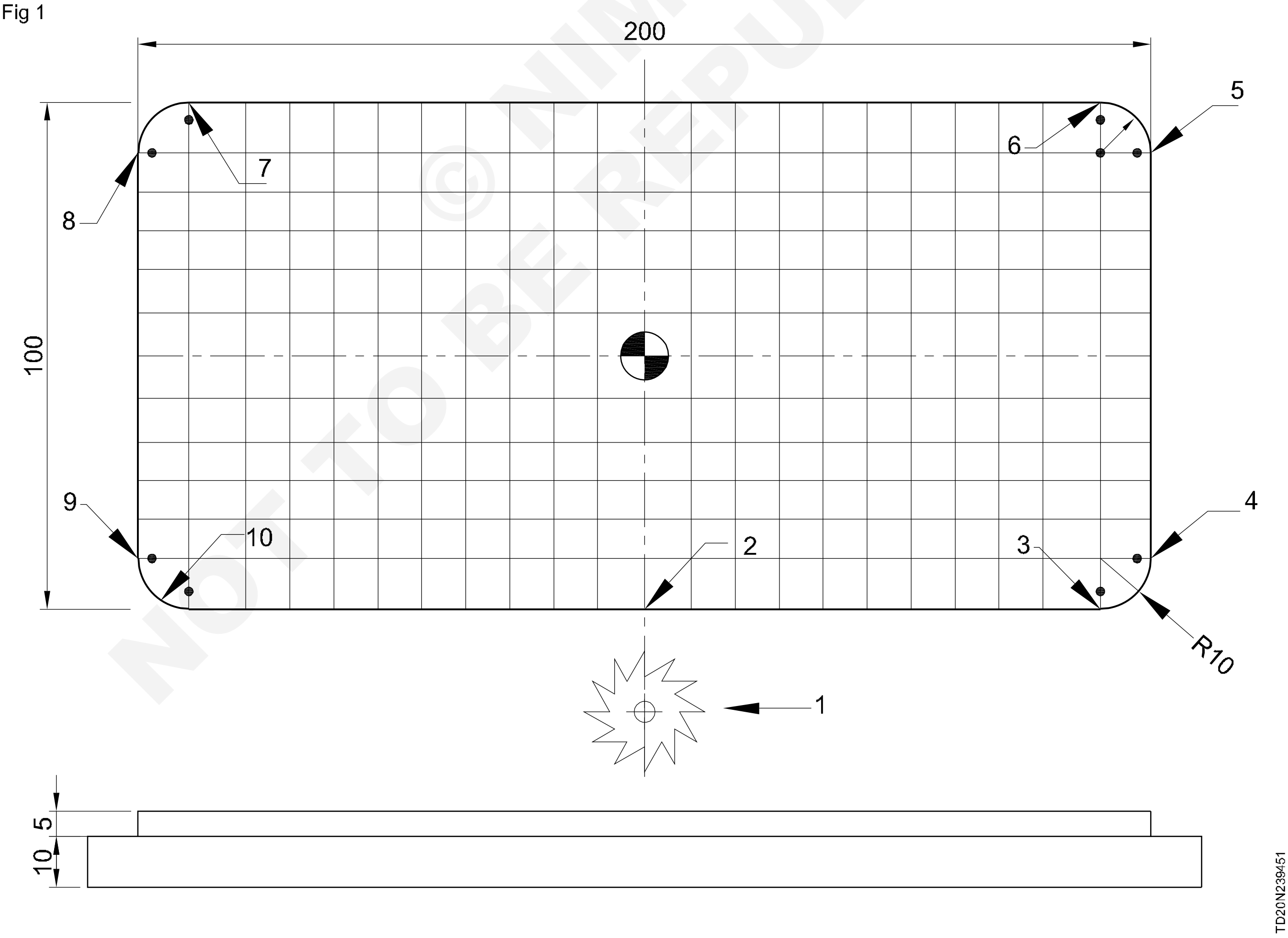
Once the command G41 or G42 is used the cutter will continue to take a compensated path. But when we want the cutter centre to revert back to the original programmed path the effect of G41/G42 has to be cancelled and this is done by the command G40.

Programming examples and 'G' code format

Objectives : At the end of this lesson you shall be able to
- **prepare the programme for the given component using G and M codes**
- **brief function of each codes**
- **explain the drilling cycle structure.**

Programming example (Fig 1)

Fig 1



Material : 200x100x15 mm Aluminium Plate.

Program Notes

1 Move zero point centre of the job (x,y axis).

2 Programs are absolute and incremental co-ordinate system.

3 Cutting tool Ø22.0 flat end mill a rule right hand helix.

4 Machining job by counter clock wise direction.

5 Cutter radius compensation used.

1111 - Program number

N1 G00 G91 G21 G49 G80 G28 Z0

N1-Block Number

G00-Rarid Positioning

G91-Incremental co-ordinates system

G21-Metric (mm)

G49-Tool length Compensation Cancel

G80-Canned Cycle Cancel

G28-Return to reference position

Z0-Return to reference position z-axis

N2 T01 M06

T01- tool number 1 (Ø20.0mm)

M06- tool change (ATC)

N3 G00 G90 G54 G43 H01 Z 200.0 M01

G90-Absolute co ordinate system

G54- Wook co ordinate system

G43- Tool length Compensation' +ve' direction

H01 -Put with off set column number in tool length off set

Z 200.0- above the tool 200.0 mm from top of the work surface.

M01-Optional stop.

N4 G17 G40 X0.0 Y-65.0 M03 S1200

G17-X and Y Plane selection

G40-Cutter radius compensation Cancel

X0.0 -tool comes over point 1 in X axis

Y-65.0- tool comes over point 1 in Y axis

M03- Spindle starts Clock wise

S1200 -Spindle speed 1200 RPM

N5 Z 5.0 M08:

Z5.0 -Tool comes 5.0mm above the work surface

M08- Coolant ON.

N6 G94 G01 Z-5.0 F 50.0:

G.94-Feed mm/minute

G01-Linean interpolation

Z-5.0 Tool feeds in work.

F-50.0-Tool feed 50.0 mm/minute.

N7 G42 G01 Y- 50.0 D16.F 300 (D=10.0)

G42-Cutter moves Compensation right side

Y-50.0-Tool moves along Y axis to point 2

D16- Off set Column number

F300.0- Tool feed rate.

(D10.0)- Cutter radius Value.

N8 X 90

X 90.0- Tool moves X axis point 3

N9 G03 X10.0 0Y-40.0R10:

G03 - Tool moves counter clock wise point 4

X 100.0 - End point of are in X axis

Y-40.0- End point of are in Y axis

R 10- Radius of arc.

N10 G01 Y 40.0

G001 - liner Interpolation.

Y40.0 - Tool moves along Y axis point 5

N11 G91 G03 X-10.0 Y 10.0 R10:

G91-Incremental Co-Ordinate

G03 - Tool moves counter clock wise direction point 6

X-10.0-End point of are in 'X' axis

Y-10.0- End point of are in 'Y' axis

R 10 - Radius of arc

N12 G90 G001 X-90.0:

G90-Absolute Co-ordinate

G01-Linear interpolation

X-90.0- Tool moves X axis point 7

N13 G03 X-100.0 Y 40.0 I0.0 J-10;

G03-Tool moves counter Clock wise direction point 8

X-100.0-End point of arc in 'X' axis

Y-40.0- End point of arc in 'Y' axis

I0.0- Arc Centre in X axis

J-10- Arc Centre in Y axis

N14 G01 Y-40.0;

G01 - Linear Interpolation

Y-40.0- Tool Y axis point 9

N15 G03 X 90.0 Y-50.0 R10;

G03 - Tool moves Counter Clockwise direction point 10

X- 90.0-End point of arc in 'X' axis

Y-50-End point of arc in 'Y' axis

R 10- Radius of arc

N16 G01 X0;

G01 -Liner interpolation

X0- Tool moves point 2

N17 G40 Y- 65

G40-Cutter radius Compensation Cancel

Y-65-Tool moves point 1

N18 Z 5.0 F 10000.0 M09;

Z 5.0-Tool moves 5.0 mm above the work piece

F 1000.0-Feed 1000.0 mm/minute

M09 - Coolant off

N19 G00 Z 200.0 M05;

G00 -Rapid Traverse

Z 200.0 -Tool moves 200.0 mm above the work piece

M05 - Spindle stop

N20 G91 G00 G28 Z0.0 Y0.0;

G91 - Incremental Co-Ordinates

G00 - Rapid Traverse

G28 - Return to reference position

Z 0.0 - Z axis return to home position

Y0.0- Y axis return to home position.

N21 M30;

M30-End of program with rewind.

Fanuc 'G' codes format are listed in Table 1

The symbol in the list represent the following

IP_ : X __ Y __ Z __ A ____

As seen above the format consists of a combination of arbitary axis addresses among X, Y, Z, A, B, C, U, V and W.

x : First basis axis (X usually)

y : Second basis axis (Y usually)

z : Third basis axis (Z usually)

α : One of the arbitrary addresses

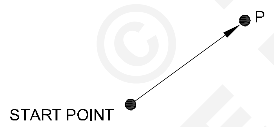
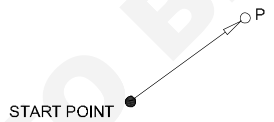
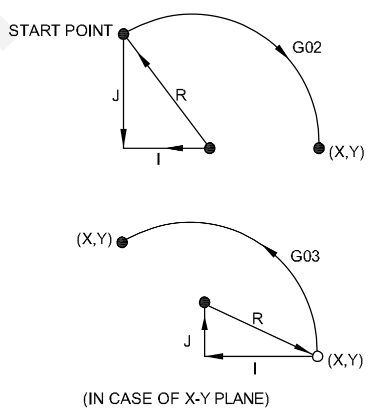
β : One of the arbitrary addresses

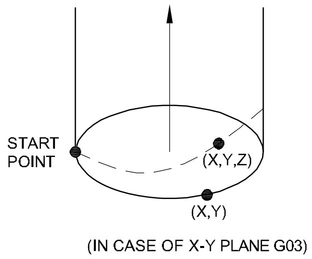
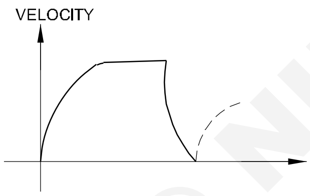
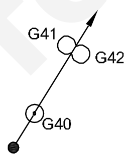
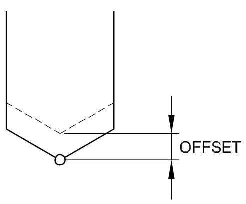
X : X axis or its parallel axis

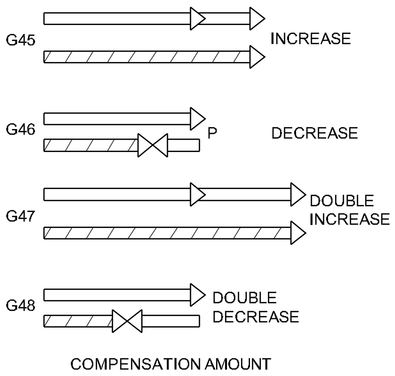
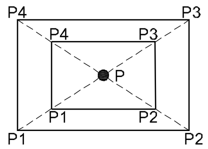
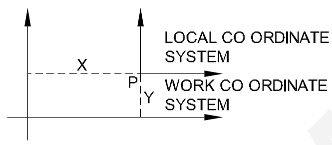
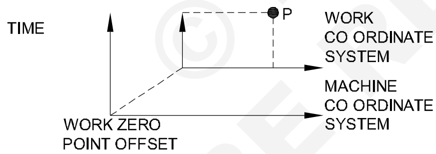
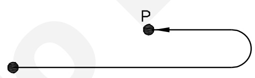
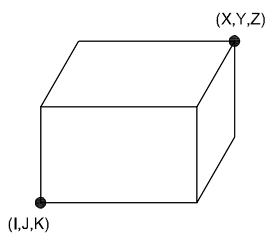
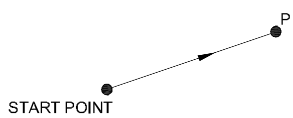
Y^P : Y axis or its parallel axis

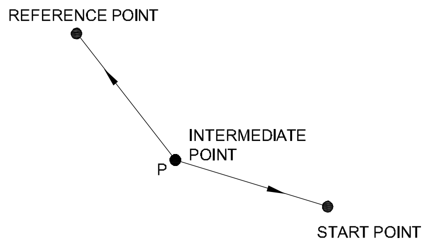
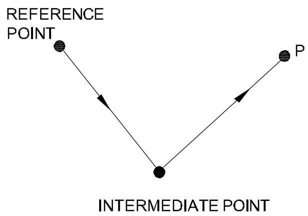
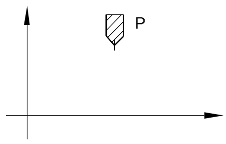
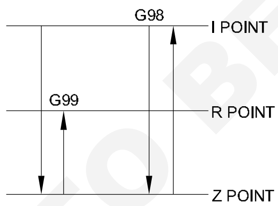
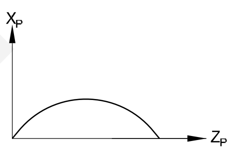
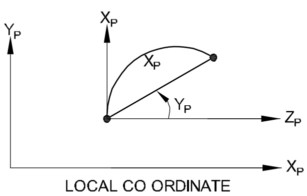
Z^P : Z axix or its parallel axis

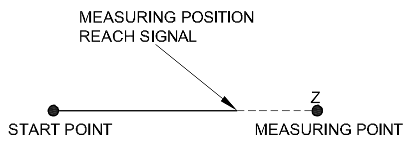
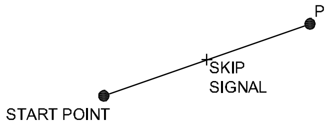
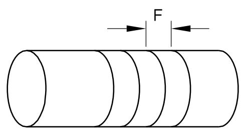
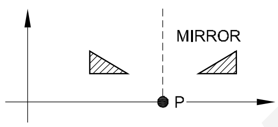
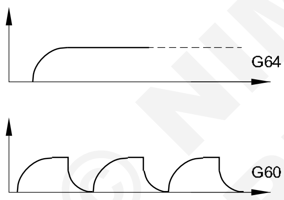
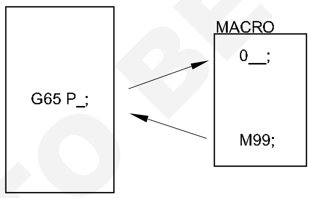
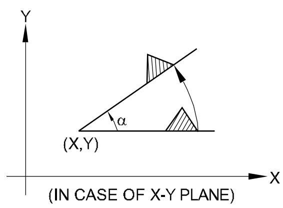
Table 1

Functions	Illustrations	Tape format
Positioning (G00)		G00P ____ :
Linear interpolation (G01)		G01P ____ F ____ :
Circular interpolation (G02, G03)		$G17\begin{Bmatrix}G02\\G03\end{Bmatrix}X_P - Y_P - \begin{Bmatrix}R\_\\I\_J\_\end{Bmatrix}F\_\_;$ $G18\begin{Bmatrix}G02\\G03\end{Bmatrix}X_P - Z_P - \begin{Bmatrix}R\_\\I\_K\_\end{Bmatrix}F\_\_;$ $G19\begin{Bmatrix}G02\\G03\end{Bmatrix}Y_P - Z_P - \begin{Bmatrix}R\_\\J\_K\_\end{Bmatrix}F\_\_;$

Functions	Illustrations	Tape format
Helical interpolation (G02, G03)		$G17\begin{Bmatrix} G02 \\ G03 \end{Bmatrix} X_P - Y_P - \begin{Bmatrix} R\_ \\ I\_J\_ \end{Bmatrix} \alpha \_\_ F\_\_;$ $G18\begin{Bmatrix} G02 \\ G03 \end{Bmatrix} X_P - Z_P - \begin{Bmatrix} R\_ \\ I\_K\_ \end{Bmatrix} \alpha \_\_ F\_\_;$ $G19\begin{Bmatrix} G02 \\ G03 \end{Bmatrix} Y_P - Z_P - \begin{Bmatrix} R\_ \\ J\_K\_ \end{Bmatrix} \alpha \_\_ F\_\_;$ α : Any axis other than circular interpolation axes
Dwell (G04)		Per second dwell $G04\begin{Bmatrix} X\_\_\_ \\ P\_\_\_\_ \end{Bmatrix};$ Per revolution dwell $G95\ G04\begin{Bmatrix} X\_\_\_ \\ P\_\_\_\_ \end{Bmatrix};$
Exact stop (G09)		$G09\begin{Bmatrix} G01 \\ G02 \\ G03 \end{Bmatrix}\_\_\_\_;$
Change of offset value of program (G10)		Geometry offset amount G10 L10 P____ R____; Wear offset amount G10 L11 P____ R____; Work zero point offset amount G10 L2 P____ IP____;
Cutter compensation (G40 - G42)		$\begin{Bmatrix} G17 \\ G18 \\ G19 \end{Bmatrix}\begin{Bmatrix} G40 \\ G41 \\ G42 \end{Bmatrix} IP\_\_\_\_\ D\_\_\_\_;$ D : Tool offset no.
Tool length compensation (G43, G44, G49)		$\begin{Bmatrix} G43 \\ G44 \\ G43 \\ G44 \end{Bmatrix}\begin{matrix} \alpha\_\_\_ H\_\_\_; \\ H\_\_\_\_\_; \end{matrix}$ H : Tool offset number G49 ; .... Cancel

<table>
<tr><th>Functions</th><th>Illustrations</th><th>Tape format</th></tr>
<tr><td>Tool offset
(G45 - G48)</td><td>
</td><td>{G45
G46
G47
G48} IP ___D ___;
H : Tool offset number</td></tr>
<tr><td>Scaling (G50, G51)</td><td>
</td><td>G51 IP ____ P ____ ;
P : Scaling magnification
G50 ; .... Cancel</td></tr>
<tr><td>Setting of local co ordinate system (G52)</td><td>
</td><td>G52 IP _____;</td></tr>
<tr><td>Commond in machine coordinate system (G53)</td><td></td><td>G53 IP _____;</td></tr>
<tr><td>Selection of work coordinate system (G54 - G59)</td><td>
</td><td>{G54
:
G59} IP ___;</td></tr>
<tr><td>Single direction positioning (G60)</td><td>
</td><td>G60 IP _____;</td></tr>
<tr><td>Inch/metric conversion (G20, G21)</td><td></td><td>Inch input G20 ;
Metric Input G21 ;</td></tr>
<tr><td>Stored stroke check (G22, G23)</td><td>
</td><td>G22 X ___Y__Z__I__J___K____;
G23 ; .... Cancel</td></tr>
<tr><td>Reference point return check (G27)</td><td>
</td><td>G27 IP ______ ;</td></tr>
</table>

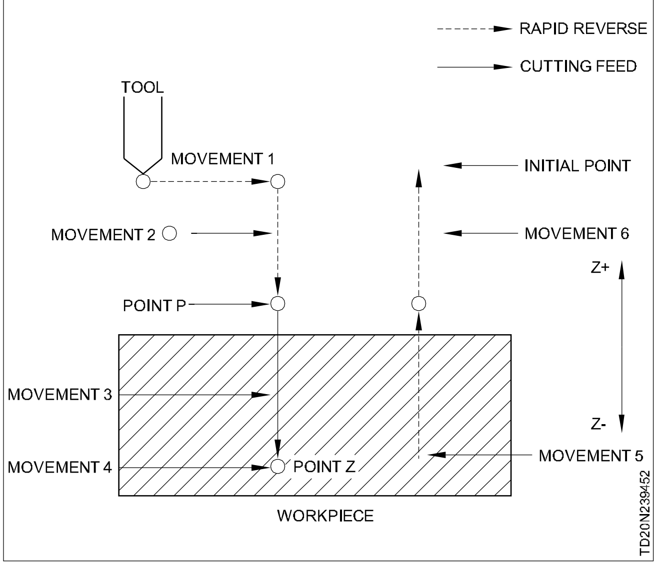
Functions	Illustrations	Tape format
Reference point return (G28) 2nd, 3rd, 4th reference point return (G30)		G28 IP____; G30P {2 / 3 / 4} IP____;
Return from reference point (G29)		G29 IP _____ ;
Absolute / incremental programming (G90/G90)		G90 _____________ ; Absolute G91 _____________ ; Incremental G90 ___ G91 ___ ; Combined use
Change of work coordinate (G92)		G92 IP ______ ;
Inverse time/ perminute feed/ perrevolution feed (G93, G94, G95)	1/min mm/min inch/min mm/rev inch/rev	G93 ___ F ____ ; Inverse time G94 ___ F ____ ; Feed per minute G95 ___ F ____ ; Feed per
Initial point return/ R point return (G98, G99)		G98 ______ ; G99 ______ ;
Hypothetical axis interpolation (G07)		G07 α0; {G17 / G18 / G19} {G02 / G03} X_P___Y_P___Z_P___; G07 α1; α; Hypotheticalaxis
Polar coordinate (G15, G16)		G17 G16 X_P ____ Y_P _____; G18 G16 Z_P ____ X_P _____; G19 G16 Y_P ____ Z_P _____; G15 ; .... Cancel

Functions	Illustrations	Tape format
Tool length measurement (G37)		G37 Z ______ ;
Skip function (G31) Multi step skip function (G31. 1 __ G31.3)		{G31 G31.1 G31.2 G31.3} IP ___F ___;
Thread cutting (G33)		Equal lead thread cutting G33 IP ___ F ___ Q ___ ; Q: Thread cutting start point shift angle inch thread cutting G33 IP __ E ___ Q ___ ; E : Threads per inch
Programmable mirror image (G50.1, G51.1)		G51.1 IP ____; G50.1 ; .... Cancel
Cutting mode/ Exact stop mode/ Tapping mode/ Automatic corner override mode		G64 ___ ; Cutting mode G60 ___ ; Exact stop mode G62 ___ ; Automatic corner override mode G63 ___ ; Tapping mode
Custom macro (G65, G66, G66.1, G67)		One shot call G65 P___ <Arugument assignment>; P : Program No. Modal call {G66 G66.1 G67;} P ___ < Arugument assignment >; .... Cancel
Coordinate system rotation (G68, G69)		G68 {G17 X_P ____ Y_P ____ G18 Z_P ____ X_P ____ G19 Y_P ____ Z_P ____} R α ; G69 ; ...... Cancel
Canned cycles (G73, G74, G76, G80 - G89)	SEE "CANNED CYCLE"	G80 ; .... Cancel {G73 G74 G76 G81 : G89} x ____ Y ____ z ___; P ___ Q ___ R ___ F ___ L ___ ;

G-Code	Group	Function
G61	15	Exact stop mode
G62		Automatic corner over ride
G63		Tapping mode
G64		Cutting mode
G65	0	Macro call
G66	12	Macro modal call
G67		Macro modal call cancell
G68	16	Co-ordinate rotation
G69		Co-ordinate rotation cancell
G73	9	Peckdrilling cycle
G74		Counter tapping cycle
G75	1	Plunge grinding cycle (O-GSC)
G76	9	Fine boring cycle
G77	1	Direct constant - Dim plunge ginding cycle (O-GSC)
G78		Cont. Feed surface grinding cycle (O-GSC)
G79		Inc. Feed surface grinding cycle (O-GSC)
G80	9	Cancell cycle cancell external op.function
G81		Drilling cycle or spot boring cycle
G82		Drilling cycle or counter boring cycle
G83		Peck drilling cycle
G84		Tapping cycle
G85		Boring cycle
G86		Boring cycle
G87		Back boring cycle
G88		Boring cycle
G89		Boring cycle
G90	3	Absolute command
G91		Increment command
G92	0	Setting for work co-ordinate system
G94	5	Feed per minute
G95		Feed per revelution
G96	13	Constant surface speed control
G97		Constant surface speed control cancel
G98	10	Return to initial point in canned cycle
G99		Return to R point in canned cycle
G107	0	Cylindrical interpolation
G150	19	Normal direction control cancel
G151		Normal direction control right side
G152		Normal direction control left side
G160	20	In-feed control function cancel (O-GSC)
G161		In-feed control function (O-GSC)

Drilling cycle structure

Fig 2



Movement 1: Positioning to hole machining start position

Movement 2: Moving of X,Y and Z axis.

Movement 3: Moving to point R at rapid traverse

Movement 4: Movement at hole bottom

Movement 5: Return to point R

Movement 6: Return to initial point at rapid travers

G81 - Spot Drilling Cycle

Format:

G98 or G99 G81 X__ Y__ Z__ R__ F__;

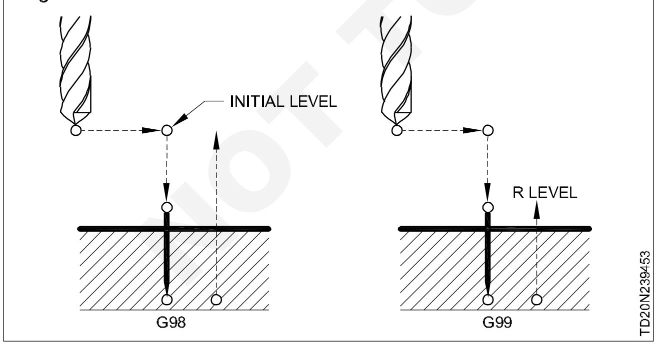
X,Y - Positioning points.

Z - Total hole depth.

R - Tool position Level

F - Feed

Fig 3



G82 -Counter Boring Cycle

Format:

G98 or G99 G82 X__ Y__ Z__ R__ P__ F__;

P - Dwell time in milliseconds.

G83 - Peck Drilling Cycle

Format:

G98 or G99 G83 X__ Y__ Z__ R__ Q__ F__;

Q - Depth of cut per pass in mm.

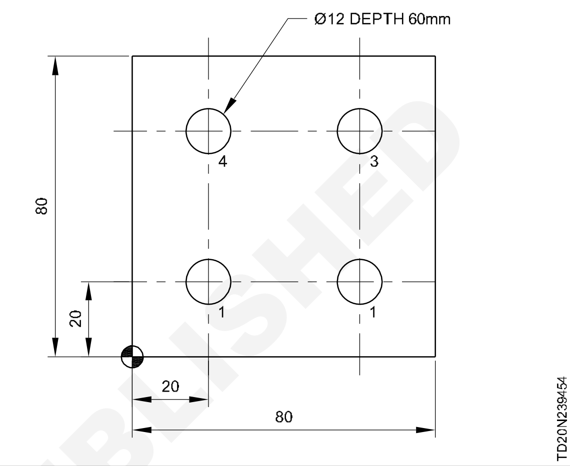
G73 - High Speed Peck Drilling Cycle

Format:

G98 or G99 G73 X__ Y__ Z__ R__ Q__ F__;

G83 & G73 - Peck drilling cycle

Fig 4



Points

1. X20.0 Y20.0
2. X60.0 Y20.0
3. X60.0 Y60.0
4. X20.0 Y60.0

O0003;

N5 G91 G28 Z0.0;

N10 G28 X0.0 Y0.0;

N15 M06 T1;

N20 G00 G90 G54 X20.0 Y20.0 S2000 M03;

N25 G43 H1 Z100.0;

N30 G00 Z50.0 M08;

N35 G98 G81 Z-3.0 R3.0 F100.0;

N40 X60.0 Y20.0;

N45 X60.0 Y60.0;

N50 X20.0 Y60.0;

N55 G00 G80 Z100.0 M09;

N60 G91 G28 Z0.0 M05;

N65 G28 X0.0 Y0.0;

N70 M01;

N75 M06 T2; (∅ 12mm drill)

N80 G00 G90 G54 X20.0 Y20.0 S1200 M03;

N85 G43 H2 Z100.0;

N90 G00 Z50.0 M08;

N95 G98 G83 or G73 Z-60.0 R3.0 Q4.0 F100.0; (Peck drilling cycle)

N100 X60.0 Y20.0;

N105 X60.0 Y60.0;

N110 X20.0 Y60.0;

N115 G00 G80 Z100.0 M09;

N120G91 G28 Z0.0 M05;

N125 G28 X0.0 Y0.0;

N130 M30;

Tapping cycles

G84 - **Right hand tapping cycle**

Format:

G98 or G99 G84 X__ Y__ Z__ R__ P__ F__;

X,Y - Positioning points.

Z - Total depth.

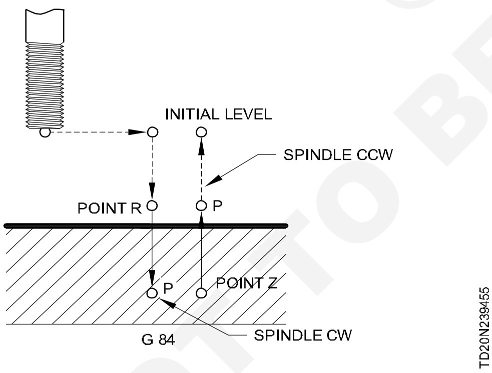
R - Tool position level

P - Dwell time in milliseconds.

F - Feed. (Pitch value)

G84 cycle is right hand tapping. In this the tool moves downward in clockwise direction (M03) in feed. At the bottom dwell time is performed, during that time spindle speed, feed stops and spindle automatically reverses its direction (M04). Return movement is also in feed. Spindle speed should be given less.

Fig 5



G74 - **Left hand tapping cycle**

Format:

G98 or G99 G74 X__ Y__ Z__ R__ P__ F__;

X,Y - Positioning points.

Z - Total depth.

R - Tool position level

P - Dwell time in milliseconds.

F - Feed. (Pitch value)

In this cycle the tool moves downward in counter clockwise direction (M04) with feed. At the bottom dwell time is performed, during that time spindle speed, feed stops and spindle automatically reverses its direction (M03). Return movement is also in feed.

G85 - Reaming or boring cycle

Format:

G98 or G99 G85 X__ Y__ Z__ R__ F__;

X,Y - Positioning points.

Z - Total depth.

R - Tool position level.

F - Feed.

G86 - BORING CYCLE (Rough)

Format:

G98 or G99 G86 X__ Y__ Z__ R__ F__;

X,Y - Positioning points.

Z - Total depth.

R - Tool position level

F - Feed.

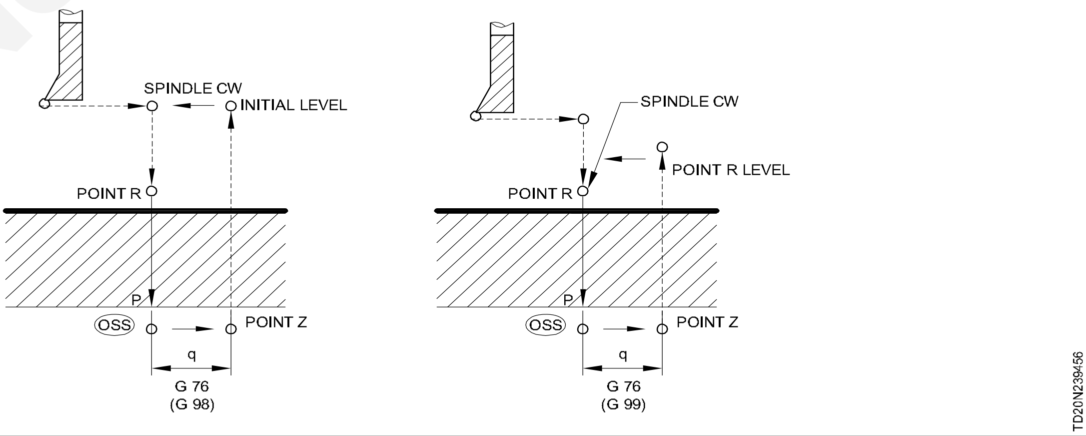
G76 - Boring cycle (Fine)

Format:

G98 or G99 G76 X__ Y__ Z__ R__ Q__ F__;

Q - Shift amount in mm. (value should be very less between 0.1 - 0.5).

Fig 6



Example:

Boring cycle

Tools

1 Ø36 mm drill

2 Ø39.5mm rough boring

3 Ø40mm fine boring (G76)

N5 G91G28Z0

N10 G28X0Y0

N15 M06 T5; (Ø39.5mm rough boring tool)

N20 G00 G90 G54 X50.0 Y50.0 S1200 M03;
N25 G43 H50 Z100.0;

N30 G98 G86 Z-35.0 R3.0 F100.0; (Rough boring)

N35 G00 G80 Z100.0 M09;

N40G91 G28 Z0.0 M05;

N45 G28 X0.0 Y0.0;

N50 M01;

N55 M06 T6; (Ø40mm fine boring tool)

N60 G00 G90 G54 X50.0 Y50.0 S1200 M03;
N65 G43 H06 Z100.0;

N70 G00 Z50.0 M08;

N75 G98 G76 Z-35.0 R5.0 Q0.2 F100.0; (Fine boring)

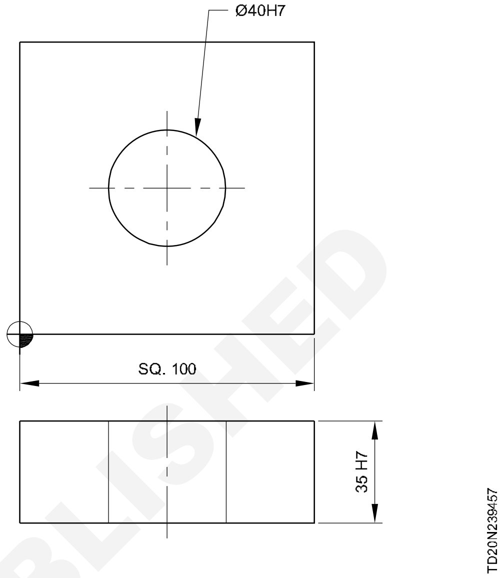
N80G00 G80 Z100.0 M09;

N85 G91 G28 Z0.0 M05;

N90 G28 X0.0 Y0.0;

N95 M30;

Fig 7

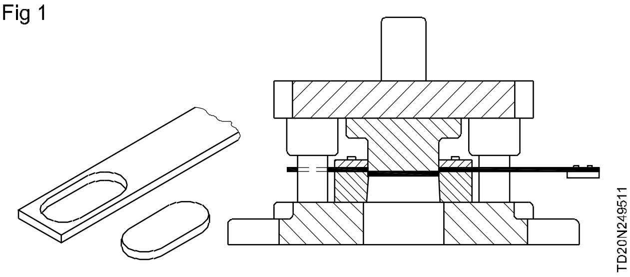


Introduction to tooling

Objectives: At the end of this lesson you shall be able to
- **state what is press tools**
- **explain moulds and its material**
- **explain die casting dies**
- **brief the functions of jigs and fixtures.**

Press tools : Press tools are tools made to produce a particular component in very large numbers, mainly out of sheet metal. The principle press tool operations are cutting operations and forming operations of sheet metal. Sheet metal components such as automobile parts, part of household appliances electronic equipments etc. are produced by press tools. (Fig 1)

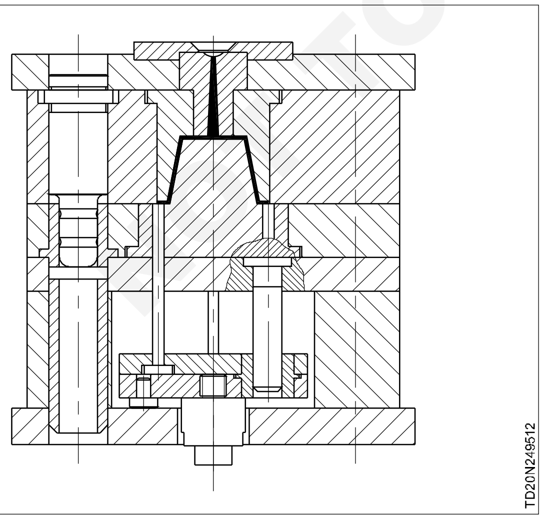
Fig 1



Moulds for plastics : Plastics are replacing wood, metal and other materials, because they perform better than the other materials in specific areas. Lot of plastic components are used in household articles, electrical switches and accessories, household appliances, automobile parts, electronic equipments etc.

In moulding of thermoplastics the plastic is melted and injected into the mould cavity resembling the shape of required component. Once the material solidifies the mould is opened and the component is taken out. A large number of components can be produced from a mould. (Fig 2)

Fig 2

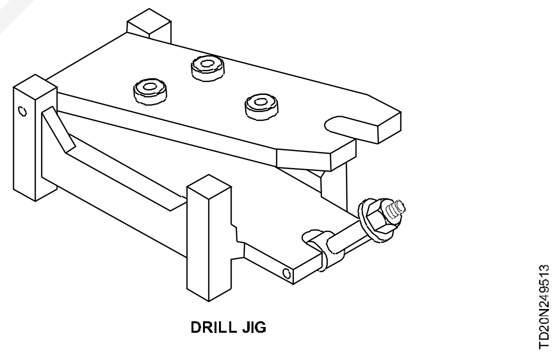


Die casting dies: Sand casting requires a sand mould to be made for each component. In die casting, a permanent die made out of tool steel is used. The cavity in the die resembles the shape of the component. The molten metal is made to flow into the cavity. When it solidifies, the die is opened and the component is taken out. A large number of components can be made from a die. Usually zinc, aluminium and its alloys are used to manufacture die cast components. Automobile engine parts, parts of household appliances etc. are made by die casting.

Jigs : Many machined parts require holes. The location of holes will be critical in many components. Drilling is the common method employed to produce holes.

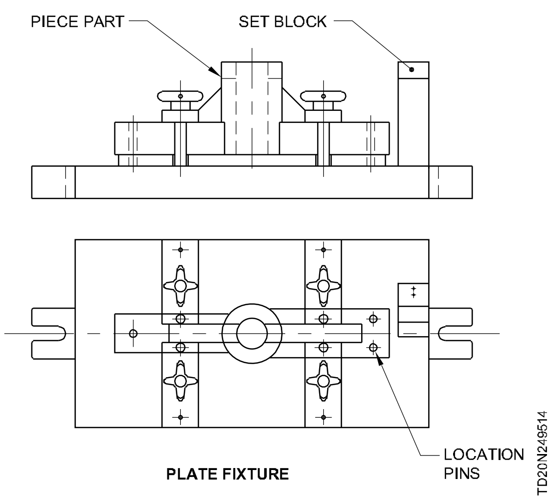
Getting repeated accuracy in large number of components by conventional marking and drilling is time consuming. Also accuracy cannot be guaranteed. A drill jig locates and guides the drill. This helps to drill holes with locational accuracy in a large number of pieces (the hole location will be identical in all the pieces). (Fig 3)

Fig 3



Fixtures : A fixture is a device for holding the workpiece during machining. When only a few parts are to be machined the job can be clamped to the machine table without using a fixture. But fixing and setting time will be more. When a large number of components are required, use of a fixture helps in clamping and locating the work faster. This increases productivity. (Fig 4)

Fig 4



Plastic material

Objectives: At the end of this lesson you shall be able to
- **differentiate between thermoplastics and thermo-setting plastics**
- **state the properties and uses of commonly used thermo-plastics**
- **state the properties and uses of commonly used thermoset plastics**
- **state the effect of fillers and additives on the properties of plastics**
- **state the properties of reinforced plastics**
- **state the function of mould release agents**
- **identify common thermo-plastics by simple tests.**

Introduction

Platic is a synthetic polymer of high molecular weight. It is composed of repeating organic chemical units. Polymer is a single large molecule. It is formed as a result of the union of two or more molecules of simpler substances.

Plastics are broadly classified into two

– thermoplastics

– thermosetting resins.

Thermoplastic resins consist of long molecules. They may have side chains or groups that are not attached to other molecules. (i.e they are not cross-linked). They can be repeatedly softened and hardened by heating and cooling. Usually thermoplastic materials are in granular form. Heat softens the material so that it can be formed. Subsequent cooling hardens it to the final desired shape. No chemical change takes place during forming.

In thermosetting resins, reactive portions of the molecules form cross links during polymerisation. Once polymerised or cured the materials cannot be softened by heating. Normally they are available in powder form. They are cured in the mould at high pressure and temperature. As technology progresses the line between thermoplastic and thermosets has become less distinct. More and more processes are developed which make use of the economic processing of thermoplastics and then converting the material to a thermoset.

Extruded polyethylene wire coating is cross-linked after extrusion chemically or by irradiation. The polyethylene is converted to thermoset material by this process. It cannot be melted by subsequent heating.

Materials and machinery have been modified to provide the economics of thermoplastic processing of thermosetting materials.

eg: Injection moulding of thermosets.

Some plastic materials are members of both groups.

eg: Thermoplastic polyesters and thermoset polyesters.

Thermoplastics: Commercial grades of thermoplastic materials are in granular form. The shape and size of the individual granules differ according to the type of material. The particular form of granulate is dictated primarily by processing requirements. They may also depend upon the production technique during manufacture. The nature of the material should be such that it can be plasticised at temperatures within a practical range to a homogeneous melt to permit even filling of the mould cavity. The material must be stable under temperature changes. Chemical decomposition should not take place during processing.

The material will deteriorate

– if it is kept inside the cylinder at normal temperatures for too long a time due to the stoppage in the cycle

– exposed to high temperatures inside the cylinder.

Thermoplastics of low thermal stability will quickly show signs of decomposition. This is known as burning. The degraded material may run through the product as streaks.

A completely clean, free running granulate, without any tendency to agglomerate is the most important requirement for optimum processing. The size and the shape of the granule should be such that optimum melt conditions are obtained. A constant bulk density of granulate is important on machines where volume feeding is employed. The nature of the surface of the granule should be such that absorption of moisture will be maintained at a tolerable level even during lengthy periods of storage. A high absorption of water leads to the production of steam in the heating cylinder. This leads to the production of faulty products. Many plastics require intensive drying of the the granulate before use. Bulk quantities are dried in drying chambers. It can even discolour the product throughtout.

These effects are more apparent in transparent or light coloured plastics. Thermal instability in the material may separate out the chemical components in the plastics. These chemicals are often corrosive.

They may attack the interior of the heating cylinder and the mould. Poisonous gases which affect the operator may also be formed.

Ingredients are added to the material to alter its colour or other properties. These substances include dyes, filters, plasticisers, dispersers (substances designed to increase the adhesion of the pigment dye) and materials to aid free flowing properties. These materials should possess the same thermal stability as the bulk material. They must be free from constituents which evaporate during processing.

For the products to be of uniform size and shape the material must exhibit small and even shrinkage on cooling.

Plasticising the material: The material in granular form is converted to a homogeneous plastic state suitable for injection to take place in the machine. The material passes through three different stages. The change from one stage to the next stage is not sharply defined owing to the differing length of the molecular chains.

When softening temperature is reached the material passes from solid state to thermoelastic state. The intermolecular forces are further reduced. When the melting temperature is reached the molecules are completely mobile and the material will be in a thermoplastic state.

Metal Flow Index (MFI): MFI indicates the flow capacity of different grades of thermoplastics. It is inversely related to the molecular weight. A high MFI grade will flow faster and has a low molecular weight. It is expressed as the amount of material extruded in ten minutes through an orifice of standard dimensions under a standard load and at a particular temperature.

Spiral flow: It is a way of expressing the flow of a particular grade of plastic inside the mould. It is determined on a standard mould having a spiral cavity marked in cm.

Flow length to wall thickness ratio (L/T ratio or flow ratio) the L/T ratio expresses the flow behaviour of the material inside the mould. It is defined as the ratio of the largest flow path to the thickness of the moulding under optimum moulding conditions.

The higher the flow ratio the higher is the tmperature required to obtain fast mould filling. Fast mould filling keeps the components free from stresses.

Thermoplastics

Polystyrene (PS): Possesses high gloss sparkle and transparency. Has low water absorption and good electrical insulation. Can be coloured by adding dry colours. The mouldings can be decorated by screen printing, hot failing. Ultrasonic welding or solvent welding can be done. The L/T ratio is between that of Polypropylene/Acrylonitrile Butadine Styrene. The material attracts dust and should be kept covered.

Applications: Polystyrene is used in the manufacture of parts which require a combination of colour clarity, stiffness, low cost and good appearance.

Thin walled and rigid containers for packing, toys, tumblers, soap dishes, paper weights, refrigerator components, fitting for lamps, desk calendars etc., are a few of the common items manufactured using polystyrene as raw material. But their resistance to heat, solvents and impact loads is interior. Addition of glass fibre to PS improves the impact strength and heat distortion temperature.

Low Density: Polyethylene (LDPE): LDPE is a crystalline thermoplastic. It is not available in transparent grades. It is tough and has moderate strength, it can be coloured. Oxidation occurs when the material is overheated in the barrel. it is therefore necessary to add antioxidants.

Applications: Shopping baskets, waste paper baskets, artificial flowers and fruits, hair brushes etc. are some of the consumer items manufactured. It is a common material for blow-moulding. It is also used in the manufacture of pipes 10 mm to 150 mm dia. for transporting water, packaging films and for coating on paper, textiles etc.

High Density Polyethylene (HDPE): Compared to LDPE, HDPE has higher density, rigidity, tensile strength, hardness, heat distortion temperature and chemical resistance. They possess low MFI and their L/T ratio is also lower than that of LDPE.

Applications: The property of high stiffness combined with high impact strength and low weight makes it is popular material for moulding articles like bottle crates, containers etc and pipes from diameter 25 mm to 80 mm for carrying water and chemicals. Packaging films, woven sacks (silt film) monofilament for fishing nets, ropes, cane for furniture etc. are also manufactured from HDPE.

Polypropylene (PP): Polypropylene is a stiff and strong material. It possesses high gloss, resistance to scratch and high barrel and mould temperature.

Applications

Automobile Industry

- Radiator cooling fans
- Tail light housings
- Carburetors, air filters etc.

Textile Industry

- Bobbins, cones, ring tubes etc.

Electrical and Electronic Industry

- Cooling fans for motors
- regulator cover for fans
- Knobs etc.

Chemical Industry

- Valves, pipe fittings
- Casings and impellers for centrifugal pumps etc.

Consumer Products

- Camera parts, time piece casings, vacuum flask components etc.

Household items

- Plugs, soap boxes, combs, hair brush, tooth brush handle, plates, cups, tumblers, saucers etc.

Medical Products

- Syringes, cotton bud sticks, sterilization containers, flasks etc.

Acrylonitrile Butadiene Styrene (ABS): It is a hard tough material with good resistance to heat and impact. It has low water absorption and good electrical insulation properties. ABS can be plated. The surface is resistant to scurring and marking. Granules are pre-dried at 80°C for 4 hours to remove moisture.

Applications: Used in the manufacture of automotive components, road signs, heavy domestic appliances and fittings, radiator grills, air conditioner grills, control panel housings, helmets, furniture items, bathroom fittings, knobs and cabinets for radio, TV, tape recorders, typewriters etc.

Polymethyl Methacrylate (PMMA): Also known as ACRYLIC, PMMA is a hard rigid material with excellent optical clarity, weather resistance and depth of colour. It has low water absorption and good dimensional stability. It is unaffected by contact with human tissues. It is stiffer than ABS or PS. It is an amorphous material. Slight traces of moisture result in frothing on the cylinder of the machine and splash marks in the mouldings. The material is pre-dried at 70° for 4 hours. Buffing can be done on cut edges to get high polish. Components can be decorated by screen printing and vacuum metalisation. They can be joined by ultrasonic welding or by painting the edges with chloroform or with a solution made from PMMA in methylene chloride.

Applications: For street light fixtures, sign boards, instrument panels, dials etc. for canopies cabin windows for air crafts, boat wind shields, contact lenses, parts of musical instruments etc.

Polyvinyl Chloride (PVC): PVC is available in two varieties

- rigid PVC
- plasticised PVC.

Rigid PVC: Rigid PVC is strong, stiff and resistant to oils. But it is difficult to process. It has a low impact strength and low heat distortion temperature. On heating PVC, hydrochloric acid is evolved. This attacks the machine and the operator. Processing is made easy by the addition of stabilisers (tri-basic lead sulphate) lubricants (bulyl stearete) processing aids and low molecular weight resins. Impact strength can be increased by mixing other plastics (15% ABS). Heat distortion temperature is improved by using fillers (10% china clay).

Rigid PVC is very stiff flow material. High temperature cannot be used because of decomposition. The L/T ratio is 60:1.

Plasticised PVC: A plasticiser is added to PVC to get a soft and flexible product. The flexible or plasticised PVC is very easy to process. a number of additives like fillers, stabiliser, lubricants etc. are also added. The type and concentration of plasticiser affects the properties of the moulded product. L/T ratio is 180:1.

Applications: PVC is used in the production of moulded articles which should posses high abrasive resistance, toughness, non-inflammable and good resistance to climatic conditions.

Rigid PVC is used in pipes and pipe fittings for water, waste water etc. Plasticised PVC is used for the manufacture of washers, footwear, footwear soles, and for insulating electrical conductors, medical tubings, blood storate bags, staircase railing, lining of chemical vessels, etc.

Poly Carbonate (PC): PC has high touchness, transparency and temperature resistance. It is rigid upon 140°C. It possess very good insulation characteristics and is self-extinguishing. It has excellent optical properties.

Generally PC has good room temperature resistance to water, dilute organic and inorganic acids, oxidizing and reducing agents, neutral acid salts, animal and vegetable oil and fats. The polymer has high resistance to staining by tea, coffee, fruit juices, ink, soaps and detergents.

Application: PC has the largest application in electronic industry. This is because of its good insulation characteristics, flame-resistance and durability. Housings for domestic appliances are also made from PC. Terminals, contact strips, fluorescent lamp starter enclosures and switch plates, high molecular weight film used in capacitor manufacture are also made from PC. Tail-light lenses for cars, football helmetsand body for electric kettles are also made from PC.

Polymide (PA): Polymide is known as Nylon. Nylons have superior abrasion resistance and toughness with low melting point, and can be processed at a lower temperature. At 1 mm thickness L/T ratio is 1:100.

The machine selected for processing Nylon should have a nozzle shut off device to avoid drooling of the melt because of the low viscosity. The material has short transition range from melt to solid. The injection speed should be relatively fast. Nylon should be stored under strict moisture-proof conditions. Pre-drying at 60°C for two hours is essential.

Components made from PA possess good mechanical strength, cold impact strength, high rigidity, heat resistance, abrasion resistance and dimensional stability.

Application: Components manufactured with Nylon as raw material are, transparent parts for medical use, gears, electrical components, instrument casings, packaging films, filaments for fishing nets, ropes etc.

Thermosetting Materials: Thermosetting materials possess

– high thermal stability

– resistance to creep, deformation underload

– high dimensional stability

– high rigidity and hardness

– light weight

– excellent electrical insulation properties.

Thermosetting moulding compounds consists of two major ingredients.

- A resin system containing curing agents, hardners, inhibitors and plasticisers.

- Fillers or reinforces.

The resin system determines the

– cost

– dimensional stability

– electrical qualities

– heat resistance

– decorative possibilities

– flammability.

Fillers and reinforces affect all these properties in varying degrees. Strength, toughness and sometimes electrical qualities are improved much by their addition.

Alkyds: Alkyds are primarily used in electrical industry. They cost less and possess good insulating properties. Their mouldability is excellent, cure time short and moulding pressures are low. They are stable up to a temperature of 120 to 150°C. Glass or synthetic fibres or mineral fillers are used to improve the strength. Casings for missiles, pipes, tanks, pressure vessels, and jigs and fixtures are manufactured from this.

UREAS (Urea formaldehyde): The properties are

– excellent colourability

– moderately good strength

– low cost

– poor dimensional stability

– poor impact strength.

The material is used for producing lighting fixtures, wiring accessories, buttons etc.

Phenolics (Phenol formaldehyde): The properties are

– low cost

– good insulation property

– excellent heat resistance

– good mechanical properties

– excellent mouldability.

They are limited in colour (black or dark brown) they are generally filled with saw-dust or flock. They are suitable for use at temperatures upto 150°C. They are used for manufacturing circuit breaker insulation, capacitors and resistor encapsulation, switch gear components etc. Phenol formaldehyde is used in moulding switch gear, relay systems, connectors etc.

Allylics: Diallyl pthalate (DAP) is the most common moulding material in the Allylic family. They are relatively costlier. They possess excellent dimensional stability and high insulation resistance. They can withstand temperatures of 150 to 175°C. They are reinforced with glass, asbestors, acrylic and polyester fibers.

Epoxies: They offer excellent electrical insulating properties and dimensional stability. They possess high strength and less moisture absorption. They are used in the moulding of various electrical and electronic components. Expoxy glass reinforced plastics are used for manufacturing aircraft components, filament-wound rocket motors, etc. Heat resistant grades are filled with mineral or glass fibre. They can withstand temperatures in the ranges of 200 to 260°C.

Melamines: They possess the following properties

– Extreme hardness

– Excellent and permanent colourability

– Non-cracking characteristics

– Self-extinguishing

Dishes and other house ware articles and electrical items are manufactured from this plastic.

Polyesters: Polyesters resins can be formulated to be

– brittle and hard

– tough and resilient

– soft and flexible when reinforced with glass fibres.

They exhibit

– excellent mechanical properties

– high strength to weight ratio

– good chemical resistance.

They are used in liquid resin form for casting furniture parts. Glass filled polyester is used for manufacturing automobile body parts, boat parts, hulls etc.

Fillers and Additives: Plastics are large polymeric molecules. Materials are added to improve the properties of plastics. The following properties can be changed by the use of fillers and additives.

- Colour

- Flexibility

- Rigidity

- Flame resistance

- Weather resistance

Processability

Plastic additives may be grouped into two main categories. Some additives modify the base polymer's characteristics by physical means.

eg: plasticisers, lubricants, impact modifiers, fillers and pigments.

The second group of additives achieve their effect by chemical reaction.

eg: flame retardents, stabilisers, ultraviolet absorbers and anti-oxidants.

Fillers: Fillers are classified as inorganic, organic, mineral, natural and synthetic types. Fillers are commonly used with thermosetting materials such as phenolics, ureas and melamines. Theya re also used with thermoplastics. Fillers are also called extenders. Theyincrease the bulkiness and decrease the cost of plastics. Properties of an extended plastic are usually interior. The use of fillers is often limited to less critical application. A filler usually comprises between 10 and 50% of the mass of the mix.

The widely used filler for thermosetting plastics is powdered wood. It consists of finely ground powder of hard woods or nut shells. They are of low cost, light in weight, strong (due to their fibrous nature) and easy to compound. But they have low thermal resistance, low dimensional stability and poor electrical insulating properties. Purer forms of silica, mica, quartz, inorganic pigments, refined metal oxieds, sulphates, etc are used as mineral fillers. Asbestors (mineral fibre) combines the advantages of an inorganic filler with those of a fibre, (use of asbestos is discouraged on health grounds). The factors for choosing a filler are

– cost, availability and uniformity
– compatibility or meltability with the resin
– moisture absorption
– physical properties
– thermal stability and mould temperature
– resistance to chemicals
– abrasiveness
– effect of plastic flow characteristics.

For getting the desired qualities in plastics, based on end use, different materials are used as fillers.

The following list gives different qualities, purposes vis-a-vis the type of filler materials used

Purpose/Quality	Filler
Bulk	Powdered, wood Saw-dust Wood pulp Purified cellulose Mica rock Sisal jute
Reinforcement	Glass fibre Asbestos fibre Cellulose fibre Cotton fabric Paper Synthetic
Hardness	Inorganic Metallic oxides Powdered metals Graphite Silica
Thermal insulation	Asbestos Diatomaceous earth Ceramic oxides Silica
Chemical resistance	Glass fibre and fabrics Synthetic fibres and fabrics Graphite Metallic oxides
Appearance	Colour pigments Dye stuffs Carbon black Powdered metals Phosphorescent minerals Woven fabrics

Plasticisers: Plastics are plasticised to increase their flexibility, resilence and melt flow. Addition of plasticisers is necessary in the manufacture of plastic sheeting, tubing, film and other forms of plastics. Plasticisers enable the molecular chains of polymers to move freely with respect to one another. The plasticisers thus act as an internal lubricant. It is not chemically linked to the plastic. Many plasticisers are liquids and some are solid. But they melt at the compounding temperature. Plasticisers are colourless, odourless and possess good thermal stability. But they decrease the strength characteristics, heat resistance, dimensional stability and solvent resistance of resins.

The popular general purpose plasticisers are

– pthalates
– epoxies
– phosphates
– adipate diesters
– sebacates etc.

Colourants: Colours varying from pastels to deep shades can be produced on a variety of plastics by the addition of colourants. Colourants used are dyes and pigments of organic and inorganic nature. Dyes are fairly soluble in plastics while the pigments are not. The choice depends on resin compatibility or the need for stability.

The colourants should be stable: Dyes, have lower thermal stability and they will be discoloured on overheating. Colour concentrates are supplied with the plastics by the manufacturers.

Ultraviolet absorbers: Virtually all plastics degrade when exposed to sunlight. They loose their colour and physical properties. PS, PVC, ABS and Polyesters are affected much.

The choice of ultraviolet absorber depends on

– the application

– the basic resin

– the ultimate effect

– the required durability of end product.

Ultraviolet absorbers are used up in time.

Flame Retardants: Flame retardants affect combustion in plastics. They work on four basic principles

– they insulate or

– create an endothermic cooling reaction or

– coat the product and exclude oxygen, or

– actually influence (negatively) combustion through reaction.

Heat stabilisers: Stabilisers prevent the degradation of resin during processing when melts are subjected to high temperature. They also extend life of the end products. PVC can be easily degreaded during processing and is the prime consumer of heat stabilisers. The commercial stabilisers are based on solid barium, cadmium, lead etc.

Antioxidants: Antioxidants protect the materials from deterioration through oxidation. The oxidation is brought on by heat, light or chemically induced mechanism. Oxidation causes brittleness, melt flow instability, loss of tensile properties and discolouration.

Antistatic Agents: Antistatic agents are also called destatisisers. They are used to reduce the build up of electrostatic charges on the surface of plastic materials by increasing surface conductivity. Plastics susceptible to electrostatic charges are PE, PP, PS, PA, Polyesters, Urethanes, Celluloses, Acrylics and Acrylonitries. Antistatic agents are selected on the basis of application, durability and effectiveness at low humidity.

Lubricants: Lubricants are additives used to help the processing of plastics. Internal lubricants act within the plastic to reduce the forces acting between the molecules. External lubricants reduce the adhesive of plastics to the metal surface of the process machinery and moulds.

Reinforced plastics: Reinforced plastics consist of a thermoplastic or thermosetting material combined with another stronger and stiffer material. The reinforcing material is usually in fibre form. The fibre carries as much the load as applied to the composite as possible. The primary duty of plastic matrix is to transfer the load from the point of application to the load bearing fibres. The effectiveness of the reinforcement depends on:

- the degree of adhesion between plastic and fibre

- relative volumes of reinforcement and plastic matrix

- the length of fibre used

- the physical and chemical properties of fibre and plastic.

Reinforcement: Glass filaments are used widely. They possess high tensile strength. Glass fibres are manufactured by extruding molten glass through a die. Then it is drawn down to a diameter of 0.01 mm.

Ceramic fibres provide

– highest possible strength

– stiffness

– good strength at elevated temperatures

Carbon and boron fibres are also used.

The effectiveness of fibre reinforcement depends on the bond between the plastic matrix and fibre. Glass fibres are cooled with substances which act as coupling agents to encourage good bonding.

Mould Release Agents: Reinforced plastics tend to stick to the mould. A mould-release agent is used to facilitate the removal of parts. The release agent forms a barrier film between the mould surface and the part.

– External release agents which are supplied to the mould surface.

eg: regenerated cellulose, poly vinyl alcohol (PVA) and nylon solutions etc.

– Internal release agent which are combined with resin.

eg: zinc and calcium stearates, silicon oils, high melting points waxes etc.

Properties of Crystalline and Amorphous Plastics

Property	Crystalline	Amorphour
Melting or softening	Fairly sharp melting point	Softens over a range of temperature.
Density	Increases as crystallinity.	Lower than crystalline material
Heat content	Greater	Lower
Volume changes on heating	Greater	Lower
Shrinkage	Greater	Lower
Compressibility	Often greater	Sometimes lower

Typical crystalline plastics: PE, PP, PA, acetals and thermoplastic polyesters.

Typical amorphous plastics: PS, PVC, SAN, ABS and acrylics.

Properties of plastics

Material	Density g/cm3 or kg/dm3	Bulk factor	Heat distortion temperature	Moulding temperature	Specific heat capacity kJ/kg	Total heat content kJ/kg
ABS	1.01	1.8-2	98	225	1470	302.4
Acetal (homo-polymer	1.42			210	1470	441
Acetal (x polymer)	1.41	1.8-2	140	225	1470	446.2
Acrylic	1.18	1.8-2	85	225	1470	302.4
Nylon 66	1.14	2-2.21	180	280	1680	567
Nylon 610	1.13			245	1680	487.2
Nylon 11	1.05			220	2436	655.2
Nylon 6	1.13		160	250	1596	525
Nylon 66GF (Glass filled)	1.38		230	290	1260	432.6
Polycarbonate	1.20	1.75	140	300	1260	352.8
Polyethylene		1.84-2.3	60	280		
MF 12	0.92		60	280	2310	810.6
MF 120	0.918			220	2310	672
Polyethylene HD	0.94	1.75-1.9	90	240	2310	718.2
MF 17	0.918			260	2310	764.4
Polyethylene oxide	1.06	1.3-2.2		310	1344	390.6
Polypropylene	0.9	1.9-1.96	120	250	1932	546
Polystyrene	1.05	1.9-2.15	90	200	1344	239.4
PVC (Rigid)	1.44	2.3	60	180	1008	159.6
PVC (Flexible)	1.30	2.2	70	160	1680	235.2
Cellulose acetate	1.28	2.4	70	195	1512	264.6
CAB	1.19	2.2	70	200	1470	264.6
SAH	1.09	1-2.15	90	220	1386	277.2

Identification of thermo plastics

No.	Name	Abbrevation	Relative density	Ease of Ignition	Whether self-extinguishing	Character Flame	Odour of burning	Behaviour material
1	Acetal	POM	1.42	Moderate	No	Pure Blue No carbon	Very sharp formalde hyde smell	Melts blackens
2	Acrylonitrile Butadiene Styryene	ABS	1.02	Moderate	No	Yellow Smoky	Flowery	Softens, bubbles blackens.
3	Metha acryl butadiene	MABS	1.07	Readily burns.	No	Yellow Smoky	Faint Pleasant.	Softens, bubbles blackens.
4	Polyamide (Nylon)	PA	1.04-1.14	Moderate.	Mostly yes.	Blue with yellow top	Burnt hair.	Melts and Froths
5	Poly carbonate	PC	1.2	Difficult	Yes	Yellow smoky.	Pleasant (Faint Phenolic)	Softens, bubbles carbonises

No.	Name	Abbre-vation	Relative density	Ease of Ignition	Whether self-extin-guishing	Character Flame	Odour of burning	Behaviour material
6	Cellulose acetate	CA	1.28	Readily burns.	No (Tri acetate yes)	Yellow flame. Moderate blacksmoke	Sharp like vinegar	Melts and drips.
7	Cellulose Acetate	CAB	1.19	Readily burns.	No	Dark Yellow blue edge Moderate smoke	Champhor	Melts and drips
8	Cellulose	CN	-	Highly inflamable	No	Bright yellow	Champhor	Burns violently.
9	Poly ethylene	PE	0.92	Readily burns	No	Blue with yellow top	Parafin wax	Melts and drips
10	Polymethyl metha acrylate	PMMA	1.19	Readily burns.	No	Blue with yellow top slight carbon	Fruity	Chars, melts bubbles.
11	Poly propylene	PP	0.9	Readily burns.	No	Blue with yellow top	Parafin wax.	Melts.
12	Poly Oppressive Styrene	PS Melts	1.05	Readily burns.	No	Yellow dense black smoke sooty	Flowery.	and bubbles.
13	Styrene Acrylonitrile	SAN burns	1.09	Readily smoky	No	Yellow smoky	Flowery	Melts and bubbles
14	Poly Vinyl chloride	PVC	1.4	Moderate	Yes	Yellow	Acidic	Softens and blackens
15	Poly Vinyl chloride (Plasticised)	PVC	1.3	Moderate	Yes	Yellow	Acidic	Melts and drips.

Hand injection mould

Objective: At the end of this lesson you shall be able to
- **state the function of various elements of a hand injection mould.**

Constructional details of Basic Injection Mould Hand Injection Mould)

Injection moulding machine, injection force is applied manually.

Hand injection mould is preferred under the following reasons.

– Mould cost is to be less.

– Dimensional accuracy of the component is less.

– The number of component to be produced is small.

– The component profile is small.

A basic injection mould consists of (Fig 1)

– cavity plate

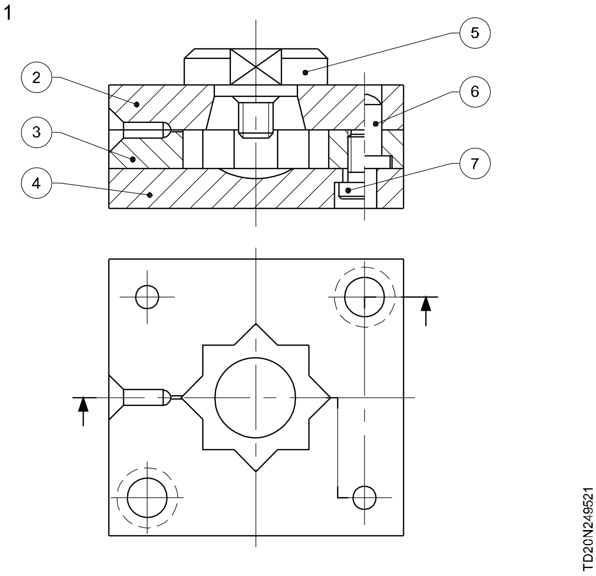
– core

– locating pins

– screws

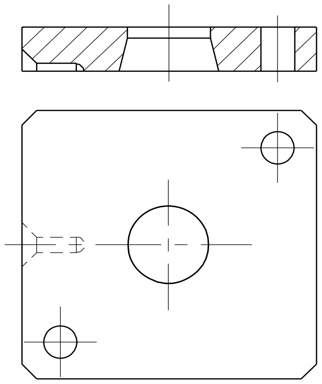
– feeding system.

Fig 1



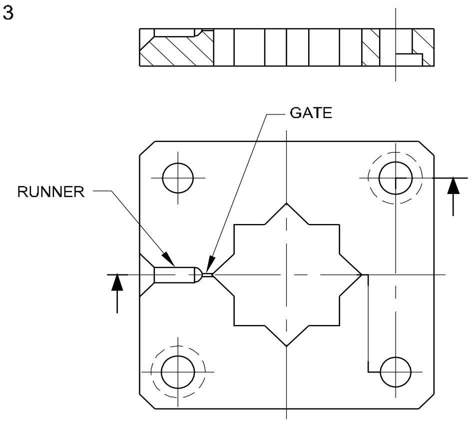
Cavity plate (Fig 2,3 & 4): The external profile of the component is formed in the cavity plates.

Fig 2



CAVITY PLATE

Fig 3



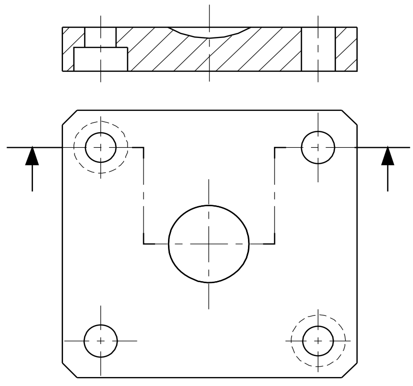
CAVITY PLATE

Core insert: The internal form of the component is provided on the core. (Fig 5)

Locating pins: The function of the locating pins is to accurately locate the mould halves. (Fig 6)

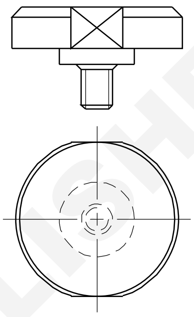
Screws: These are used to clamp the plates together.

Fig 4



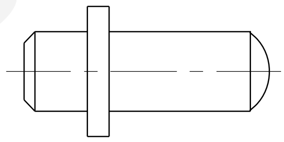
CAVITY PLATE

Fig 5



CORE INSERT

Fig 6



LOCATING PIN

Feed system: The feed system consists of two grooves termed as runner and gate. The runner is machined in between the mould plates. The gate connects the cavity to the runner. (Fig 3)

Injection moulding machine

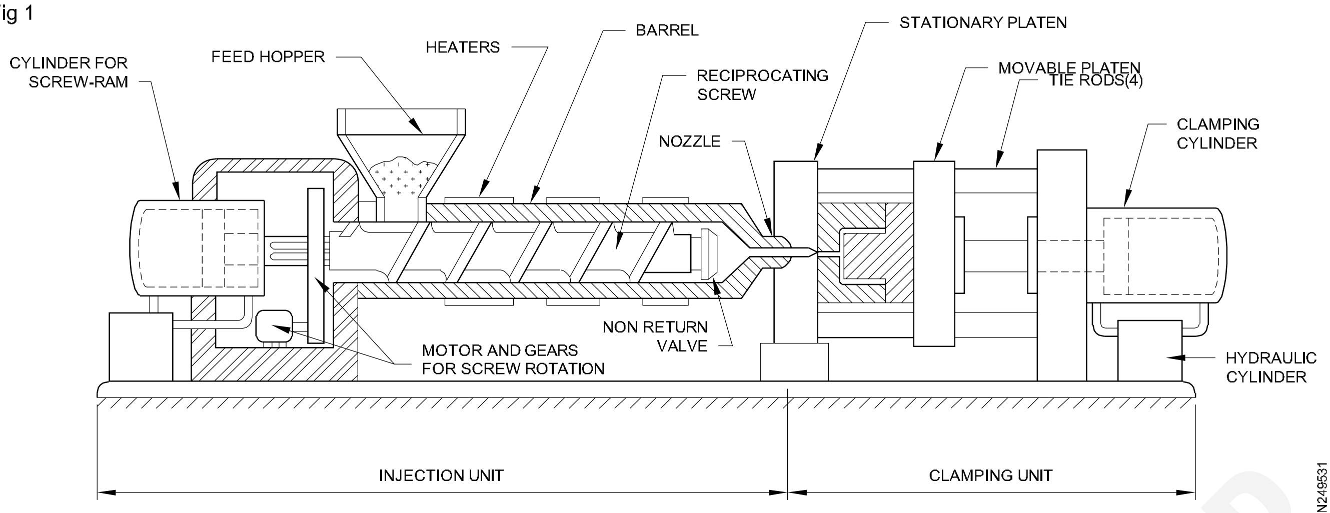
Objectives: At the end of this lesson you shall be able to

- **state the constructional features of an injection moulding machine**
- **state the functions of different types of nozzles**
- **state the specification of an injection moulding machine**
- **state the clamping force of an injection moulding machine**
- **state the plasticizing capacity of an injection moulding machine**
- **state the clamping force of an injection moulding machine**
- **differentiate between `shot capacity' and `injection velocity' of an injection moulding machine.**

The basic parts of a modern injection moulding machine are as follows.

- A feed hopper for holding and feeding the plastic material.
- A reciprocating screw and a barrel where material is heated and brought to the stage of injection.
- An arrangement for applying pressure to the screw to force the plastic material into the mould.
- A fixed and moving plate on which the mould is mounted. The moving plate will have ejecting arrangements.
- Mould clamping arrangements either by toggle or direct hydraulic lock which holds the mould halves together during injection.
- A system of controls for operating, plasticising, injection and mould closing mechanism in the correct sequence and with variable arrangements, for regulating during the process cycle. (Fig 1)

Fig 1



INJECTION MOULDING MACHINE

The screw (Fig 2): The screw of an injection moulding machine is divided into three sections or zones.

– the feed section

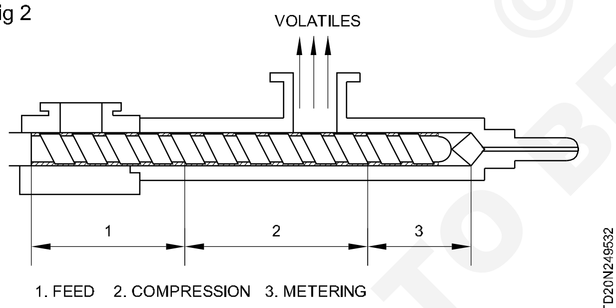
– the compression section

– the metering section.

The feed section transports the plastic material from the hopper to the heated portion of the barrel. The plastic granule is compressed to a homogeneous melt in the compression section.

The final mixing and heating of the material into a homogeneous melt is carried out in the metering zone.

Fig 2



The drive for the rotation of the screw during the feed cycle is obtained from a hydraulic motor or a variable speed electric motor.

Check valves are used on the screw to avoid the material coming back along the screw during injection stroke.

The nozzle: The nozzle is connected to the end of the barrel. Through the nozzle the softened material passes into the mould. General purpose or standard nozzles are made:

– in varying diameters to accommodate the cylinder bores and sprue diameters of moulds

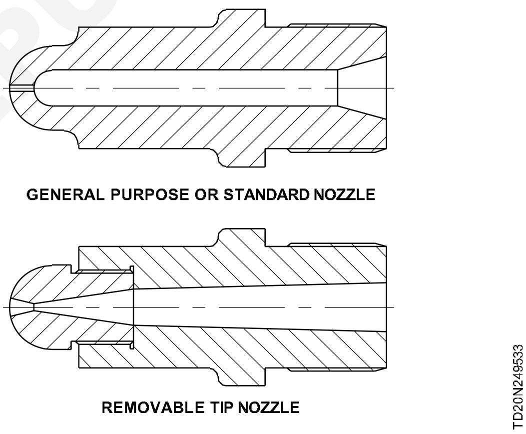
– in varying lengths depending on the heating requirements.

Standard nozzles are made with straight bore, restricted at the outlet end and tapered as required to match the cylinder bore at the inlet end.

To facilitate the melt to be injected at the optimum temperature, band heaters, internal catridges or heating coils are installed in the nozzles.

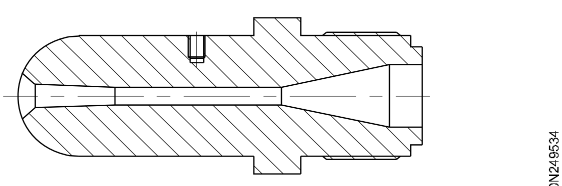
Depending on the type of resin other nozzle configurations are also used. (Fig 3)

Fig 3



Materials like nylon have very low melt viscosity. This necessitates the use of nozzles known as reverse taper nozzle. The reverse taper is used to freeze off the melt or prevent drooling between shots. (Fig 4)

Fig 4



REVERSE TAPER OR MELT VALUE NOZZLE USED FOR NYLON ABS

The nozzles can also be used for cellulose and ABS because of better flow, ease of cleaning sprue breaks without long strings.

For corrosive polymers like PVC, nozzles made of stainless steel are used. Chrome-plated nozzles are not preferred because the plating peels off from corners.

Purging: Purging is the process of cleaning one colour or type of material from the cylinder of the machine by forcing it out with new colour or material.

Machine specifications: Injection moulding machines are commonly classified according to the clamping force that they can generate and the amount of material that they can inject.

Injection capacity or shot capacity: It is the maximum volume of material that can be injected by the screw during one cycle of operation.

Plasticising capacity: It is the amount of material that can be processed by the machine per hour. It is expressed in kg/h.

Injection rate or injection velocity: It is the maximum rate at which the screw can inject or shoot material from the barrel during one shot.

Injection pressure: It is the pressure by which the material is injected through the nozzle. Unit. MPa.

Clamping force: It is the maximum force that the clamping system can exert on the mould. It is also the maximum force by which the mould halves can be closed together. It is given in KN or MN.

Maximum daylight: It is the farthest distance that the machine platens can be separated from each other. It can be obtained by adding the maximum mould thickness to the maximum opening stroke.

Different materials require different injection pressures

The following classification is given for general guidance.

High pressure (HP)	– Nylon
	RIGID PVC
	S A N
	A B S
	ACRYLIC
	P C
Medium pressure (MP)	– H D P E
	P P
	P S
Low pressure (LP)	– SOFT P V C
	L D P E

Normally different types of pressure heads capable of being fitted on the same barrel and screw are available. It is advisable to have atleast two types of heads for processing different materials.

Mould construction

Objectives: At the end of this lesson you shall be able to
- **state the different parts of an injection mould**
- **state the function of the different parts of an injection mould.**

Basic terminology

A mould is divided into two parts.

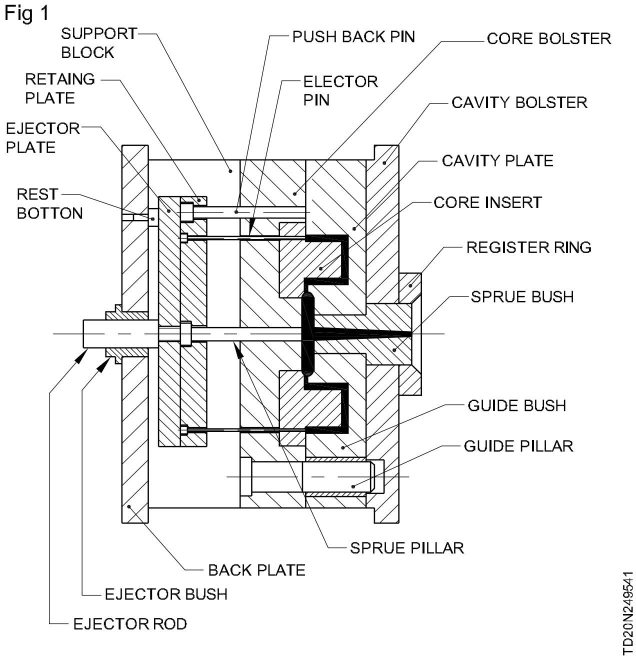
- Fixed half
- Moving half

The half that is attached to the stationary platen of the machine is termed the fixed half.

The half that is attached to the moving platen of the machine is known as the moving half.

Generally the core is situated in the moving half. This is because, it is easier to provide the ejector system in the moving half. (Fig 1)

Fig 1



Impression (Fig 2): The injection mould contains within it an impression.

Plastic material is injected and cooled in this impression.

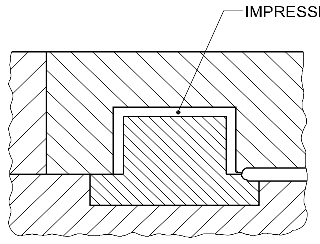
The impression gives form to the moulding.

The impression is formed by two mould members.

The cavity: It forms the external shape of the moulding.

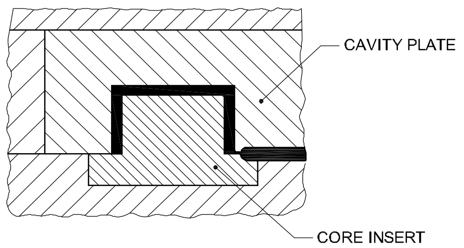
The core: It forms the internal shape of the moulding.

Fig 2



Cavity and core plates (Fig 3): The basic mould consists of two plates.

Fig 3



The plate in which the cavity is formed is known as the cavity plate and the plate from which a core projects is termed the core plate.

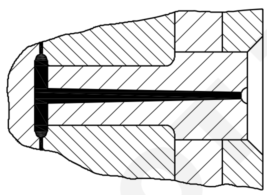
When the mould is closed the two plates come together forming a space between the cavity and core.

This is the impression.

The portion adjacent to the impression and that butt together on core and cavity plate is called the parting surface.

Sprue bush (Fig 4): The plastic material is delivered to the nozzle of the machine as a melt. It is transferred to the mould through a passage. The passage is a tapered hole within a bush. The material in this passage is termed the sprue. Sprue bush is made from nickel chrome steel and is hardened. The backward movement of the sprue bush is prevented by stepping the end and fitting a register ring. (Fig 4)

Fig 4



SPRUE BUSH

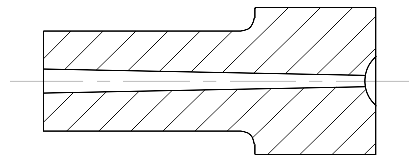
The register ring serves a dual purpose of securing the sprue bush and mould location. The internal aperture of the sprue bush has an included taper of 3 degrees to 5 degrees. This facilitates the removal of the sprue from the bush. There are two basic types of sprue bushes.

– Sprue bush with a spherical recess. It is used when a spherical front ended nozzle is employed. (Fig 5)

– Sprue bush with a flat rear face. It is used when a corresponding nozzle is used. (Fig 6)

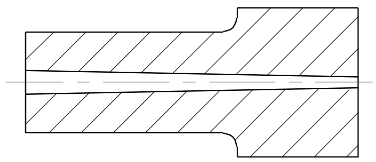
To obtain a leak-free joint when spherical seating is used, the alignment between the nozzle and the bush aperture should be perfect.

Fig 5



SPHERICAL SEATING

Fig 6

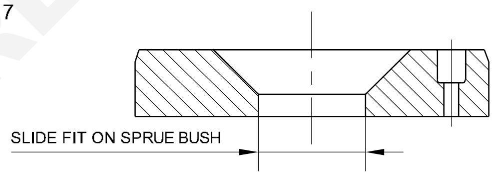


FLAT SEATING

But in the case of flat faced nozzle no leakage occurs even if the two apertures are slightly out of line. It can also be located on the stepped end of the sprue bush. In such cases the register ring secures the sprue bush in position.

Register ring (Fig 7): The alignment between the nozzle and the sprue should be correct for the easy flow of material. The register ring aligns the mould to the machine. The register ring is a flat circular member fitted on to the front face of the mould. When the mould is mounted on the machine the register ring fits into a circular hole which is accurately machined in the injection platen on the cylinder nozzle axis. This ensures that the aperture in the nozzle is in direct alignment with the sprue bush hole. The register ring is located on the outside diameter of the sprue bush.

Fig 7



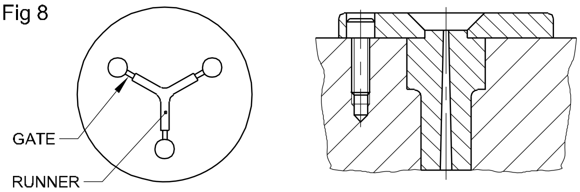
REGISTER RING DETAILS

Runner and gate system (Fig 8): The material can be directly injected into the impression.

– Through the sprue bush

– Through a runner and gate system.

Fig 8

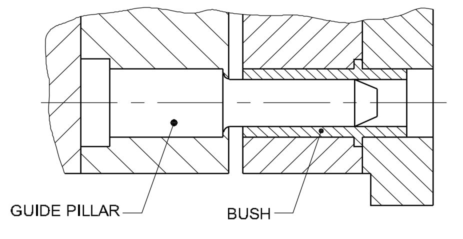


REGISTER RING USED FOR MOULD LOCATION AND SCREWING SPRUE BUSH

The runner is a channel machined into the mould plate to connect the sprue with the entrance to the impression. The gate is a channel connecting the runner with the impression. It has a small cross-sectional area when compared with the rest of the feed system.

Guide pins and bushes (Fig 9): The cavity and core should be in perfect alignment to get mouldings of uniform wall thickness. This is achieved by providing guide pins and bushes in the mould plates. The working diameter of the guide pin is smaller than its fitting diameter. If the working diameter is bent, it can be easily removed without damaging the fitting hole. The guide bush provides a wear-resisting working surface for the guide pillar.

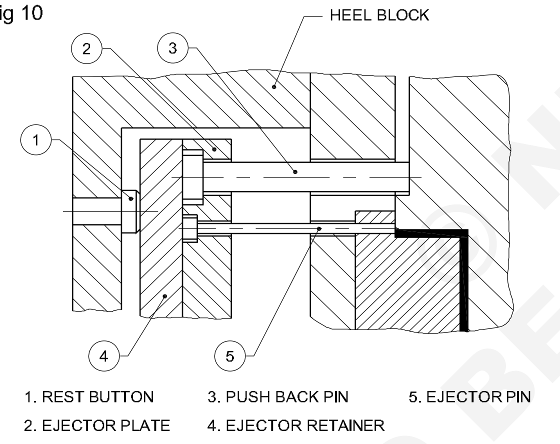
Fig 9



Heel blocks (Fig 10): Heel blocks are mounted on to the back plate. Heel blocks support the mould plate. They provide space for the ejector plate assembly to fit and operate. They are also called as support blocks.

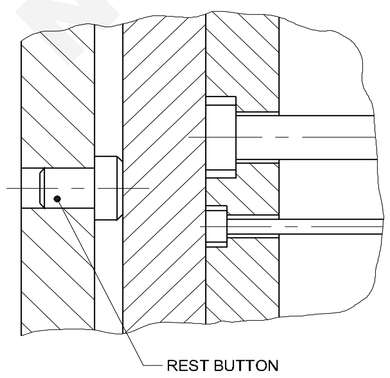
Ejector plate (Fig 10): The ejector plate transmits the ejector force from the actuating system of the machine to the ejector element.

Fig 10



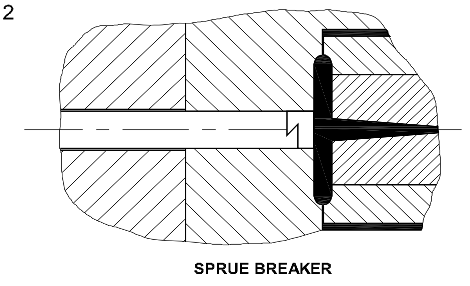
Rest buttons (Fig 11): With a large ejector plate system it is preferable to provide rest buttons or stop pins on the underside of the ejector plate. This reduces the seating area. The possibility of the ejector elements remaining slightly above due to the foreign matter being trapped behind the ejector plate is avoided by using the rest buttons.

Fig 11



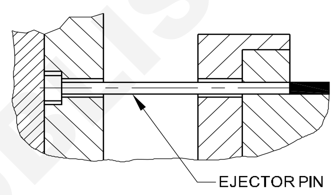
Sprue breaker (Fig 12): It is necessary to remove the sprue from the sprue bush after each injection for further injection of melt into the mould. Sprue breakers or sprue hookers are used to pull the sprue out of the sprue bush when the mould opens.

Fig 12



Ejector (Fig 13): The ejector is that part of the ejector system which applies the ejection force of the moulding. There are different types of ejectors used (ejector pins, ejector plates and sleeve ejectors). The ejector is selected depending on the component characteristics.

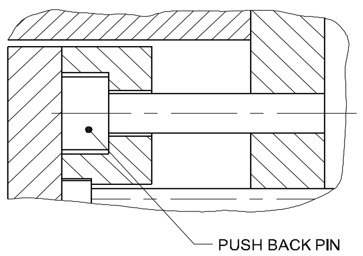
Fig 13



Ejector retainer (Fig 10): The ejector retainer is a plate securely attached to the ejector plate for the purpose of retaining the ejector.

Push-back pins (Fig 14): The push-back pins push the ejector assembly to its rear position as the mould is closed. Moulds used in injection moulding machines having hydraulic ejection actuating system do not require push back pins.

Fig 14



Attachment of mould to machine platen: There are two ways of attaching the mould halves to the platen of the injection moulding machines.

– directly bymeans of bolts

– indirectly by means of bolts and clamps.

For direct bolting clearance holes or slots are provided in each mould half to correspond with the holes tapped in the machines platen. In indirect clamping the mould is clamped on the platen by means of clamps and bolts.

Parting surface

Objectives: At the end of this lesson you shall be able to
- **choose parting surfaces for shaped components**
- **state the reason for providing relief for the parting surface**
- **state the effect of air traps in the mould**
- **state the function of air vents**
- **determine the position of air vents.**

Injection mould parting surface

Generally injection mould has two major components : the movable halve and the fixed halve, during injection moulding process, these two halves are pressed tightly and form a sealed hollow to avoid the melt flow leakage and maintain certain holding pressure, the pressed surface are called parting surface, usually parting surface are the biggest projection outline of the part.

The portion of both the mould plates adjacent to the impressions which butt together is the parting surface. The parting surface forms a seal and prevents the loss of plastic material from the impression.

Parting surfaces are classified as flat and non flat.

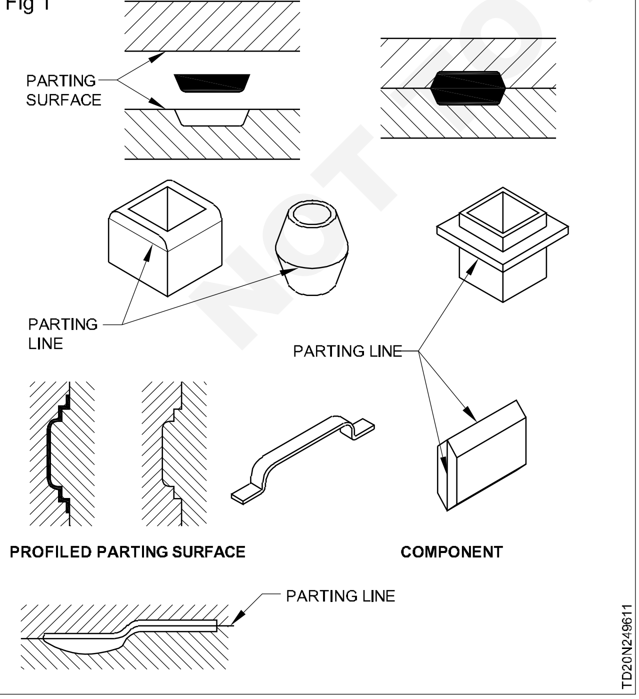
The non flat parting surfaces include stepped, profiled and angled surfaces.

The nature of the parting surface depends on the shape of the component.

A parting surface must be so chosen that the moulding can be ejected easily from the mould.

If the parting surfaces are not properly matched the material from the impression will escape through the gap. This material is called flash.

Fig 1



Types of parting surface

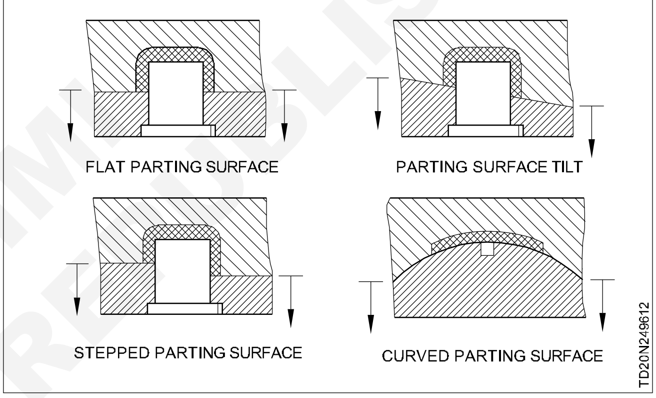
Flat parting surface (Fig 2a)

Parting surface tilt (Fig 2b)

Stepped parting surface (Fig 2c)

Curved parting surface (Fig 2d)

Fig 2



Parting surface design principles

Parting surface design directly affects the product quality, ease of construction and operation of the injection mold, the mold design is one of the key factors to sucess.

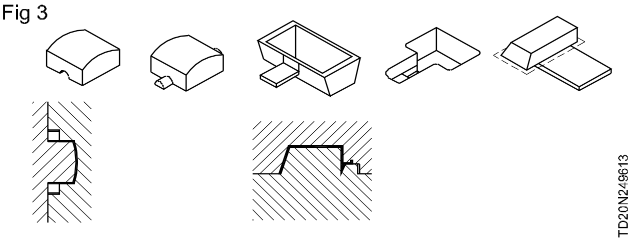
Design parting surface should follow the following principles

- **Design the injection mould structure as simple as possible:** Avoid or reduce side parting as much as possible to reduce the complexily of mould making.
- **Ease of de-moulding:** As far as possible, for instance, the plastic parts should stay in fixed halve of the injection mould and ejected by ejection system, avoid side de-moulding action.
- **Ensure the dimensional accuracy of the moulded parts:** Try to make the parts with similar dimensional accuracy requirements in the same module to reduce manufacturing and assembly errors.
- **Ensure appearance requirements of the moulded parts:** Flash or mismatch occurs usualy on parting surface, so the parting surface should not be designed on any high quality requirement surfaces.

- **Ensure dequate venting:** If possible, parting surface should be place at the end of the mold flow. This is going to be a good advantage of filling and venting.

Complex edge forms : components whose edge form is not constant leads to complex parting surfaces.

Fig 3

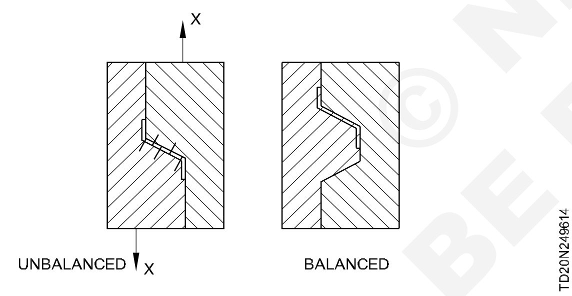


Local stepped and profiled parting surfaces

For components with a few small irregularities (which otherwise are a regular form) it is necessary to provide a stepped or profiled parting surface. This is achieved by localising the change in the parting surface to permit the major portion of the surface to be kept flat.

Balancing mould surfaces: When the parting surface is not flat, unbalanced forces are to be considered. In a mould with a stepped parting surface the plastic material under pressure within the impression will exert a force which will tend to open the mould in the laterial direction (x). The forces on the mould can be balanced by reversing the step as shown in the (Fig 4). When balancing is not practicable very sturdy guide pillars should be provided.

Fig 4



Relief of parting surfaces: Bedding down a parting surface over the entire surface is not practicable. It will be extremely expensive. It also affects the functioning of the mould, effect of injection pressure and locking force with respect to the area of contact between two surfaces.

p = f/a

where

p = theoretical injection pressure(MPa)

f = the applied force (MN)

a = the area of the injection ram (m^2).

The actual pressure exerted within the impression is considerably less than the theoretical value for the following reasons:

- the melt is non-newtonian
- the viscosity of the melt progressively increases as it passes through the mould due to cooling
- the acutal pressure within the impression depends on the length of the flow path (i.e., sprue, runners etc.).

In practice 25% to 45% of the theoretical value is used. The effective injection pressure is transmitted to the projected area of the impressions, the runners and gates. This produces a forcewhich tends to open the mould. This tendency to open is resisted by a locking force. the clamping force should exceed the opening force (i.e. the locking force).

The parting surface adjacent to the impression and runner is bedded down on a relatively small area. This is to safeguard against very high opening forces developed due to flash.

The surrounding surface is relieved. This small area adjacent to the impression and the runner is termed as the 'land'.

The corners of the mould are left high in order to withstand the large clmaping forces.

Venting: when plastic material enters an impression, air is displaced. Normally the air can escape between the two mould plates. But if the plates are perfectly flat the air will be trapped within the impression. This results in discoloration, sink marks, incomplete filing etc. Vents are provided in the mould to allow air to escape freely.

The vent is a shallow slot. It is not more than 0.05mm deep and 3mm wide. If the depth is more the plastic material can pass through the slot. This will leave a flash mark.

Positions where vents are required are

- on symmetrical mouldings at the point further most from the gate
- at the point where flow paths are likely to meet
- at the bottom of projections.

In the third case it is necessary to provide the vent through the mould plate. This is achieved by incorporating an ejector pin in the required position. The air escapes through the minute gap between the ejector pin and the mould plate hole.

Core and cavity dimensions

Objectives: At the end of this lesson you shall be able to
- **state shrinkage in mouldings**
- **state the factors governing shrinkage**
- **determine core and cavity dimensions**

Core and cavity dimensions

When a moulding cools, it contracts by an amount depending on the material being processed. Each material has its own characteristics shrinkage. The chart gives the percentage of shrinkage for some of the commonly used materials.

Material	Percentage of shrinkage
ABS	0.4 - 0.6
Polystyrene (PS)	0.5 - 0.6
Acrylic	0.4 - 0.8
Polycarbonate	0.6 - 0.8
Delrin(Poly acetal)	1.5 - 2.5
Nylon	0.4 - 1.0
Poly Propylene (PP)	1.0 - 2.0
HDPE	1.5 - 3.0
Polyethylene (PE)	
LDPE	1.5 - 2.5

Moulding shrinkage is affected by various conditions, the main ones being:

Shrinkage increases with

- increase of material temperature
- increase of mould temperature
- increase in section thickness

Shrinkage decreases with

- higher cavity pressure (injection pressure)
- longer injection time
- pressence of fillers

To assess the exact amount of shrinkage it is very important that the production is nearer to the optimum and that the cycle has settled down to steady conditions. The moulding should be checked only when it is fully set. Measurements should be made in all dimensions because shrinkage varies with the direction of flow. Differential shrinkage rates in the longitudinal and transverse planes may give rise to distortion of flat mouldings. Because of all these factors it is difficult to give the exact amount of shrinkage for moulding. generally the grade of mouldings are IT-11.

Values of shrinkage

Material	Moulding shrinkage mm/mm
ABS	
High impact	0.005 - 0.007
Heat resistant	0.004-0.005
Medium impact	0.005
Acetal	0.020-0.035
Acrylic	
Easy flow	0.002-0.007
General purpose	0.002-0.009
Heat resistant	0.003-0.010
High impact	0.004-0.008
Cellulose acetate	
Hard	0.002-0.005
Medium	0.002-0.005
Soft	0.002-0.005
High acetyl	0.002-0.005
Cellulose acetate butyrate	0.002-0.005
Nylon	
Nylon 6.6	0.010-0.025
Nylon 6	0.007 - 0.015
Nylon 6.10	0.010 - 0.025
Nylon 6.11	0.010 - 0.025
Nylon 6.12	0.008 - 0.020
Transparent	0.004 - 0.006
Glass filled	0.005 - 0.010
Polyethylene	
Low density	0.015 - 0.035
High density	0.015 - 0.030
Poly propylene	0.010 - 0.030
Polystyrene	
General purpose	0.002 - 0.008
Heat resistant	0.002 - 0.008
Toughened	0.003 - 0.006
Polytetra fluro ethylene	0.050 - 0.100
Poly vinyl chloride	
Unplasticised	0.002 - 0.004
Rigid	0.002 - 0.004
Semi-rigid	0.005 - 0.025
Flexible	0.002 - 0.006
Syrene acrylonitrile	0.002 - 0.006
Poly carbonate	0.006 - 0.007

Example of shrinkage for component and cavity are given Fig 1&2

Fig 1

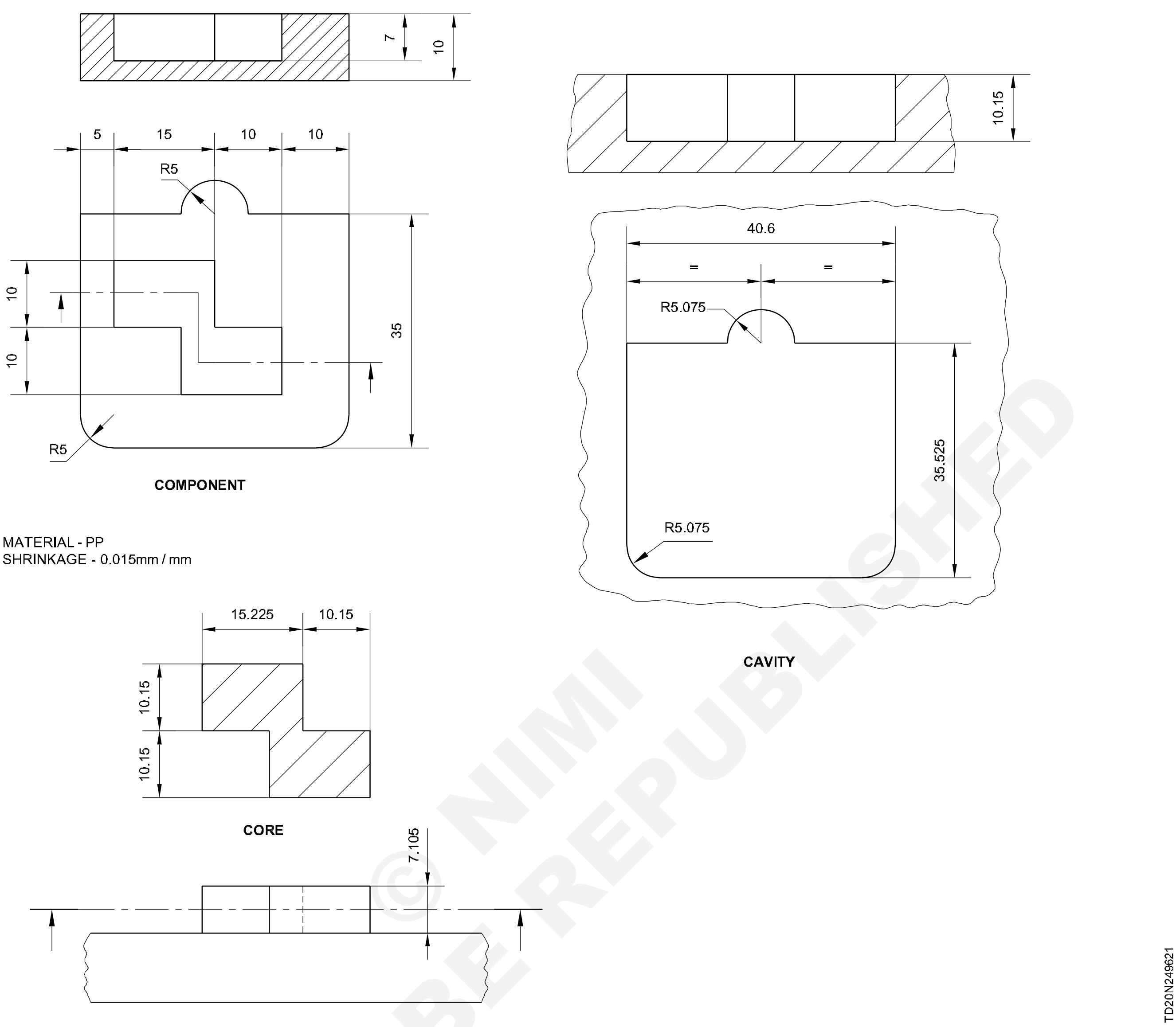
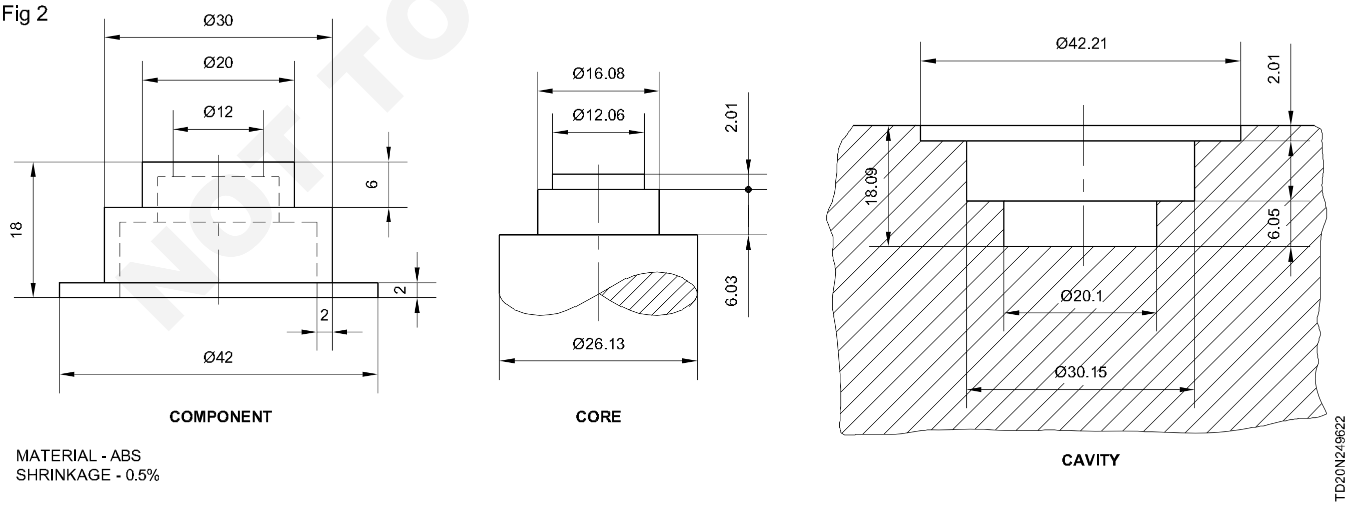

7
10
10.15
5
15
10
10
R5
10
10
35
R5
COMPONENT
MATERIAL - PP
SHRINKAGE - 0.015mm / mm
40.6
=
=
R5.075
35.525
R5.075
CAVITY
15.225
10.15
10.15
10.15
CORE
7.105
TD20N249621

Fig 2


Ø30
Ø20
Ø12
6
18
2
2
Ø42
COMPONENT
MATERIAL - ABS
SHRINKAGE - 0.5%
Ø16.08
Ø12.06
2.01
6.03
Ø26.13
CORE
Ø42.21
2.01
18.09
6.05
Ø20.1
Ø30.15
CAVITY
TD20N249622

Ejection system

Objectives: At the end of this lesson you shall be able to
- **state the ejection process in an injection moulding machine**
- **state the types of ejection grids**
- **state the function of the ejector grid**
- **state the parts of the ejector plate assembly**
- **state the functioning of the ejector plate assembly in an injection mould**
- **explain the various return systems for an ejector plate**
- **state the function of rest buttons**
- **state the various ejection methods**
- **explain the stripper plate ejection method**
- **explain ejection from the fixed half.**

Ejection: All thermoplastic materials contract as they solidfy. The moulding will shrink on the core and will stick to it. The amount of shrinkage depends on the material used. The moulded part has to be positively ejected from the core. The injection moulding machine has facilities for the automatic actuation of the ejector system. This system is situated behind the moving platen. The mould's ejector system will be most effectively operated if it is placed in the moving half of the mould. Therefore, the core is also located in the moving half.

The ejector system consists fo the

- ejector grid
- ejector plate assembly
- ejector

Ejector grid: The ejector grid supports the mould plate and provides a space into which the ejector plate assembly can be fitted and operated. The grid consists of a back plate on which support blocks are mounted.

There are three alternative designs.

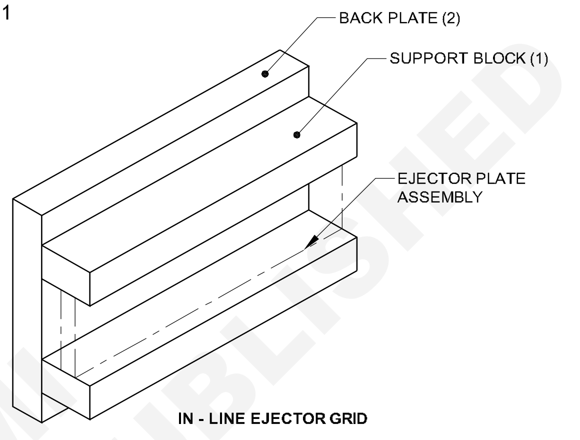
- the inline ejector grid
- the frame type ejector grid
- the circular support block grid

Inline ejector grid (Fig 1): This consists of two rectangular support blocks(1) mounted on a back plate(2). The ejector plate assembly is accommodated in the parallel space between the two support blocks. This design is suitable for small moulds. The distance between the support blocks should not be much. Thick mould plate or extra support blocks are used to prevent distortion of the mould plate by the ejection force.

Frame type ejector grid (Fig 2): The most common type is the rectangular frame type ejector. It consists of four support blocks mounted on a back plate. This type is used commonly for the following reasons.

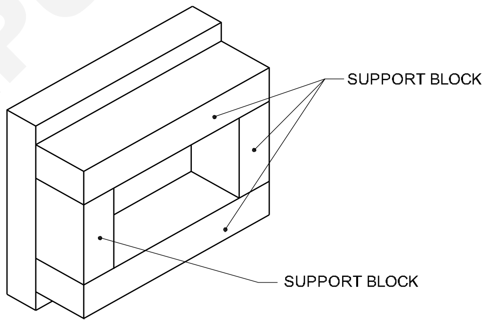
- It is simple and economical to manufacture.
- It provides support to the mould plate in a small mould.
- A rectangular shaped ejector plate can be used.
- The ejector plate assembly is completely enclosed. This prevents foreign bodies from entering the system.

Fig 1



IN - LINE EJECTOR GRID

Fig 2

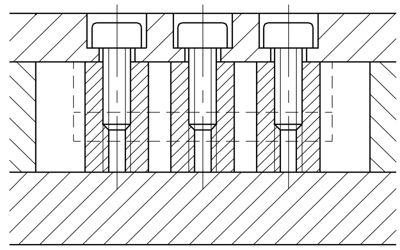


FRAME TYPE EJECTOR GRID

- When the outside shape of the mould plate is circular the ejector grid is also made circular. It is more expensive than the rectangular type. The effective support provided by the circular ejector grid decreases as the size of the mould plate increases. Additional local support blocks are to be provided.

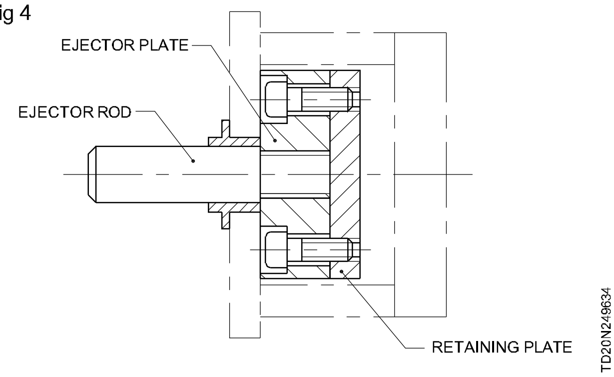
Circular support block grid (Fig 3): Circular support blocks are used to support the mould plate only. This system is used for large moulds when no extra support can be gained by including rectangular blocks.

Fig 3



Ejector plate assembly (Fig 4): The ejector element (pin, blade,sleeve etc) is attached to the ejector plate assembly. The assembly is contained in a pocket formed by the ejector grid directly behind the mould plate. The assembly consists of:

Fig 4



- ejector plate
- retaining plate and
- ejector rod

One end of the ejector rod is screwed into the ejector plate. The ejector rod is guided in an ejector rod bush fitted in the back plate.

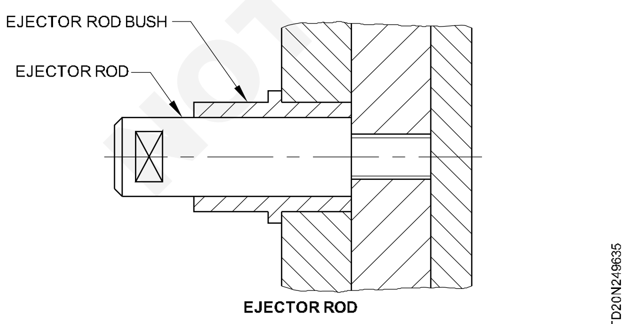
The ejector rod functions as an actuating member and also as a method of guiding the assembly.

Ejector plate: The ejector plate transmits the ejector force from the actuating system of the machine to the moulding. It will be more when the draft is less and the moulding is deep. The ejector plate must be thick enough to prevent deflection.

Retaining plate: Its purpose is to retain the ejector elements. The thickness of the plate depends on the depth of the head of the ejector element.

Ejector rod and ejector bush (Fig 5): The ejector rod is attached to the ejector plate by means of screw threads. The ejector rod bush is press fitted into the back plate. The ejector rod and ejector rod bush are case hardened to resist wear.

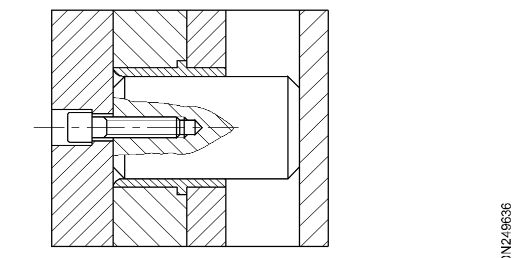
Fig 5



EJECTOR ROD

Guiding and supporting ejector plate assembly (Fig 6): The type of the guide system depends on the size of the mould. For smaller moulds the ejector rod is guided in an ejector rod bush fitted in the back plate. It maintains alignment and provides support for the ejector plate assembly. Guide pillars attached to the back plate and guide bushes provided in the ejector assembly is another method for aligning and supporting the ejector assembly. These guide pillars can be used as local support blocks also.

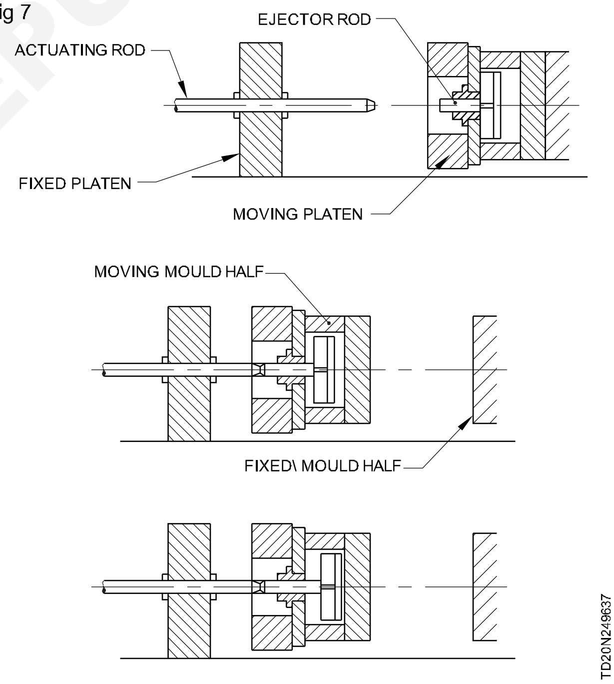
Fig 6



GUIDING AND SUPPORTING EJECTOR SYSTEM

Functioning (Fig 7): The ejection half of the mould is mounted on the moving platen of the injection moulding machine. The actuating rod can be adjusted for various alternative 'ejector strokes'. When the mould opens the ejector rod at some point of the stroke strikes the actuating rod and arrests the ejector assembly. The mould plate and the ejector grid continue to move to the left until the opening stroke is complete. This relative movement between the ejector plate assembly and the mould plate is necessary to operate the ejector element. In smaller machines the acuator rod passes through the centre of the moving platen. On larger machine several actuator rods are used so that a balanced force can be applied to the ejector plate.

Fig 7

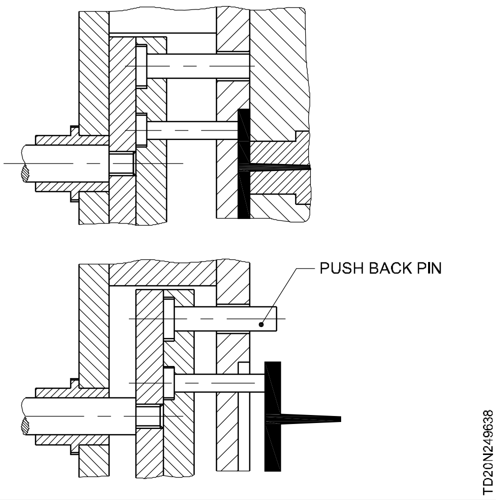


Ejector plate assembly return system: In a stripper plate mould the stripper plate is directly connected to the ejector plate by the rods. When the mould closes the stripper plate strikes the fixed mould plate. The stripper plate and the ejector plate return to their original position.

Push back pin return system (Fig 8): Push back pins are large diameter ejector pins. They are fitted to the ejector plate assembly. In the moulding position, the push back pins are flush with the mould plate surface. In the ejection position they project beyond the mould plate surface.

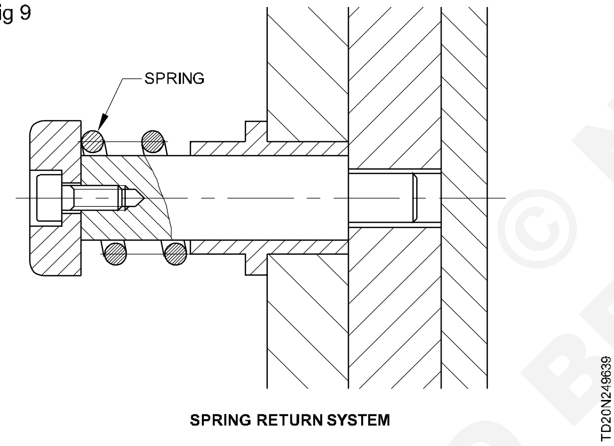
When the mould is being closed the push back pins strike the fixed mould plate and progressively return the ejector plate assembly to the rear position.

Fig 8



Spring return system (Fig 9): In small moulds where the ejection force is less, a spring can be used to return the ejector plate assembly. The spring is fitted on the ejector rod with a cap to keep it in position.

Fig 9

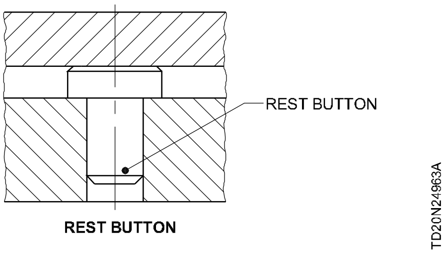


SPRING RETURN SYSTEM

Hydraulic return system: The hydraulic system of the injection moulding machine is made use of in this case. The ejection system of the mould is connected to the hydraulic system of the machine through a standard adapter. The ejection system can be operated independent of the mould opening. No push back pins are required to return the ejector system to its original positionafter ejection, because the hydraulic system pulls back the ejector system during its return stroke.

Stop pins or rest button (Fig 10): Stop pins are incorporated on the underside of the ejector plate when

Fig 10



REST BUTTON

- the ejector plate is large
- the ejector bar system is large

The heads of the stop pins should be relatively large to prevent them from digging into the back plate. Stop pins reduce the effective seating area. This prevents the ejector elements remaining slightly above due to foreign matter being trapped behind the ejector plate.

Ejection method

Ejection techniques: when a moulding cools, it contracts. For solid mouldings which do not have any internal form (eg. solid disc, cube etc.) the moulding will shrink away from the cavity walls (Fig 11). The ejection is simple in such cases. when the moulding has internal form, on cooling the component will shrink on the core (Fig 12). Positive type of ejection is necessary in such cases. The ejection methods are as follows.

Fig 11

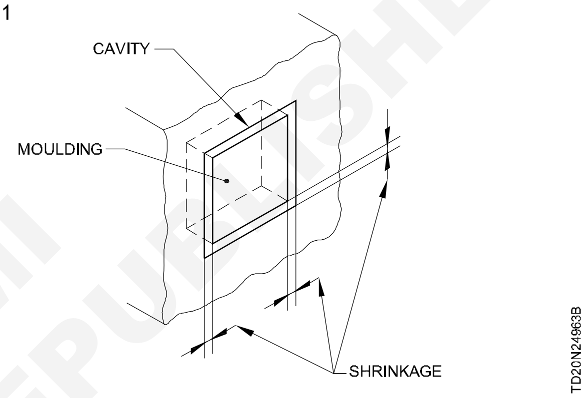
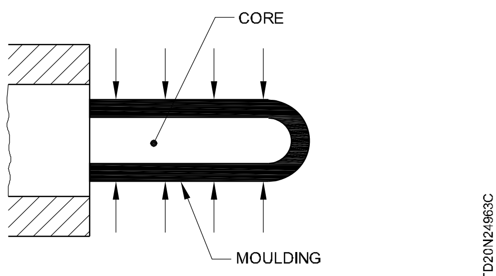


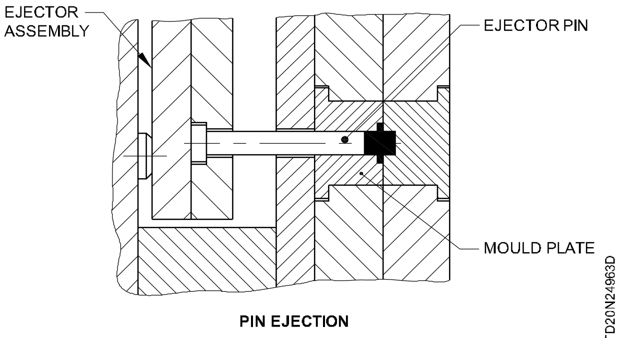
Fig 12



Pin ejection: This is the simplest and the most economical type of ejection. The moulding is ejected by the application of force by the ejectior pin (Fig 13). The ejector pin is attached to the ejector plate assembly. In operation it is moved forward relative to the mould plate (Fig 14). The ejection pin pushes the moulding from the cavity. Slide fit (H7/g6) is maintained between the ejector pin and its mating hole in the mould plate. This is to avoid the plastic material creeping into the clearance. The ejector pin is shouldered and proper seating is provided in the retaining plate.

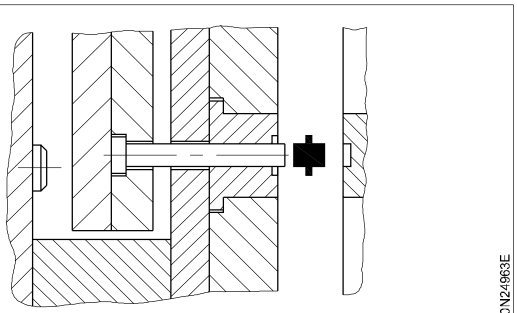
The accommodation provided must allow the ejector pin to float. The direction of movement of the ejector pin is controlled by the hole in the mould plate. If this hole is not at right angle to the mould plate face, this arrangement allows a relative lateral movement between the ejector pin and the retaining plate. This avoids the bending or breakage of the ejector pins.

Fig 13



PIN EJECTION

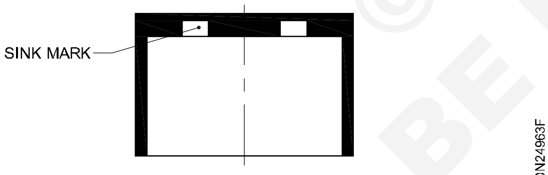
Fig 14



PIN EJECTION

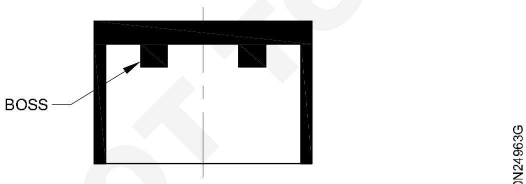
The pin should be flush with the top of the core. If the pin projects above, a sink mark will be formed on the moulding (Fig 15). If it is sunk below the core, a boss is formed on the moulding (Fig 16). The top surface of the pin must have the same surface finish as the rest of the impression. The location of the ejector pins and their number depends on the component shape and size. The aim should be to eject the moulding with as little distortion as possible.

Fig 15



EJECTOR PIN PROTRUDING OUT

Fig 16

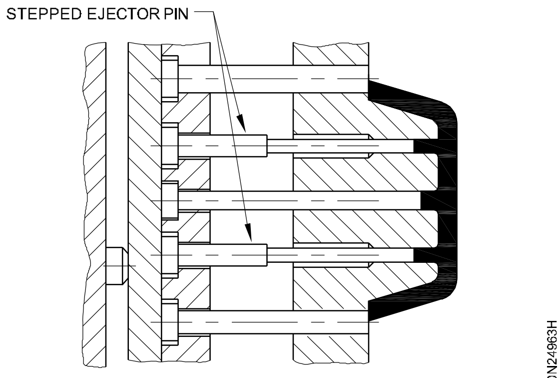


EJECTOR PIN SUNK INSIDE

It is advisable to use a maximum number of ejector pins in order to avoid distortion. When a small diameter ejector pin is required the working length is to be kept to a minimum. This is known as a stepped ejector pin (Fig 17). Ejector pins are hardened and tempered.

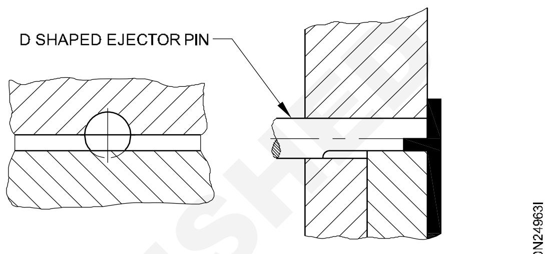
D shaped ejector pin: This is a flat sided ejector pin. It is made by machining a flat on a standard ejector pin. It is used for the ejection of thin walled box type mouldings. (Fig 18).

Fig 17



STEPPED EJECTOR PIN

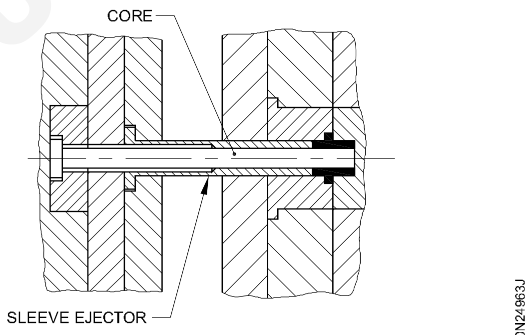
Fig 18



D SHAPED EJECTOR PIN

Sleeve ejection: In this method the moulding is ejected by means of a hollow ejector pin or sleeve (Fig 19). It is used in the following circumstances.

Fig 19

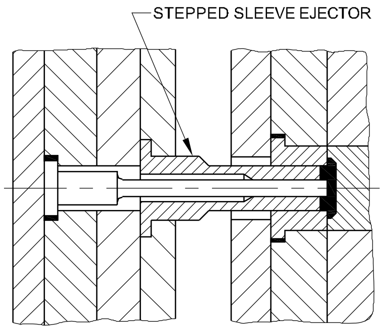


SLEEVE EJECTOR

- for the ejection of small cylindrical mouldings
- for the ejection of circular bosses on a moulding of any shape
- to provide positive ejection around a local core pin forming a round hole in the moulding.

The fit between the sleeve and the cacity insert and core pin is slide fit (H7/g6). The rear end of the sleeve is fitted to the ejector assembly. The sleeve and the core pin unit is allowed to float because the outside diameter of the sleeve is a slide fit (H7/g6) in the cavity insert. When the ejector assembly is actuated the sleeve is moved relatively to the core and to the cavity and the moulding is ejected. The ejection force is applied to a relatively large surface area. This design is restricted to circular types. When small diameter sleeve ejector is required the working length to be kept to a minimum. This is known as stepped sleeve ejector (Fig 20).

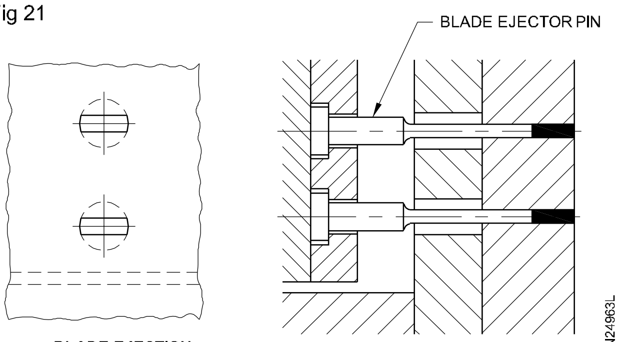
Fig 20



STEPPED SLEEVE EJECTION

Blade ejection: Slender parts are ejected by this methods. Ribs and other projections cannot be satisfactorily ejected by the standard type of ejector pins. This is basically a rectangular ejector pin. (Fig 21)

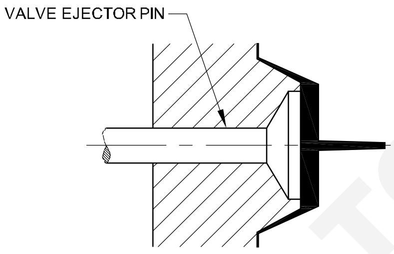
Fig 21



BLADE EJECTION

Valve ejection: This is a large diameter ejector pin. It is used for the ejection of large components. This ejector applies the ejection force on to the inside surface of the moulding. (Fig 22)

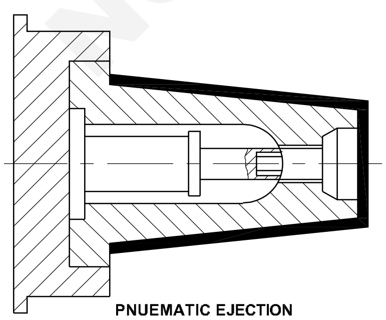
Fig 22



VALVE EJECTION

Air Ejection: The ejecting force is given by the compressed Air (Fig 23)

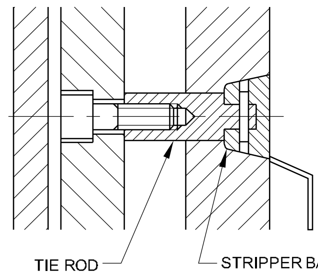
Fig 23



PNUEMATIC EJECTION

Stripper bar ejection: The ejector element pushes against the bottom edge of the moulding. The effective ejection area is greater in this case. This method of ejection is suitable for walled box type moulding. A single bar is used along each wall of the moulding. This reduces marks left on the surface of the moulding. The stripper bar is coupled to the ejector plate by a tie rod. (Fig 24)

Fig 24



STRIPPER BAR EJECTION

Stripper plate ejection: This method is adopted for ejecting:

- circular box type mouldings
- mouldings with thin wall sections.

The core will have slide fit (H7/g6) with the cavity of the striper plate (Fig 25). When the mould starts to open, the stripper plate moves back with the core plate. When the moulding is clear off the cavity, the movement of the stripper plate is stopped. The core plate continues to move backwards. (Fig 26)

Fig 25

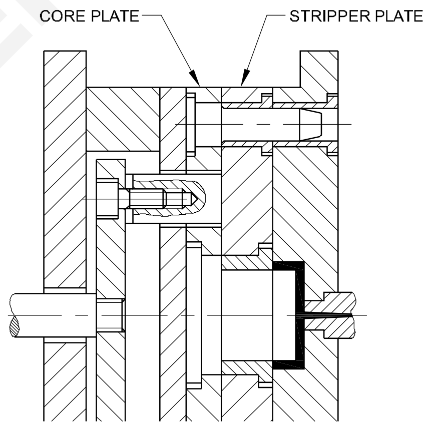
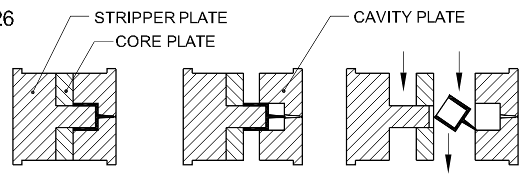


Fig 26



The core is withdrawn through the stripper plate and the moulding is ejected. The stripper plate mould consists of three plates. (Fig 27& 28)

Core Plate

Stripper plate

Cavity plate

Fig 27

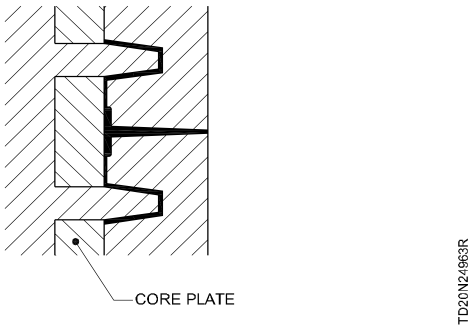
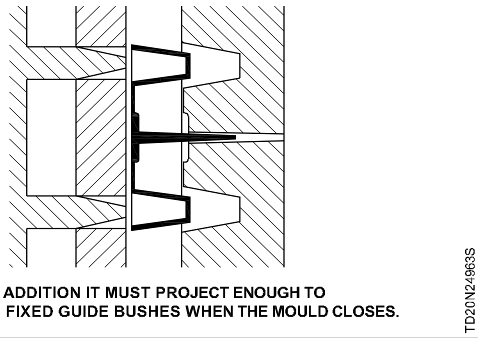


Fig 28



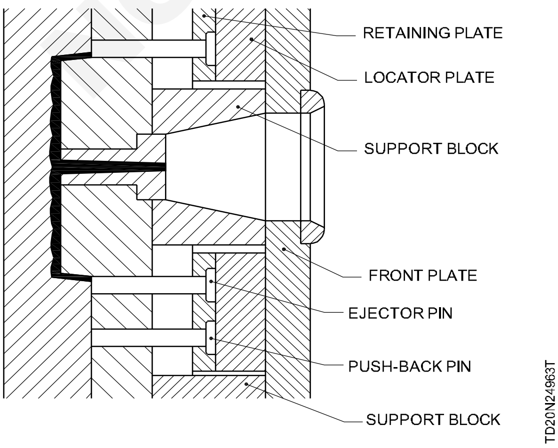
IN ADDITION IT MUST PROJECT ENOUGH TO ENTER THE FIXED GUIDE BUSHES WHEN THE MOULD CLOSES.

When the mould is opened there are two spaces. The spaces are termed ad day lights. The stripper plate is mounted on the guide pillars. The length of the working diameter of the guide pillar is longer for the following reasons:

- The guide pillar must support the stripper plate over the complete ejection stroke. It should also project enough to enter the guide bushes in the fixed half when the mould closes.
- The stripper plate should be thick enough to resist bending during ejection.

Ejection from fixed half: In some cases, it will be necessary to mount the core and the ejector system on the fixed mould half. This mould half must be sufficiently thick to incorporate an ejector grid and ejector assembly behind the fixed mould plate. The distance the material has to travel can be minimised by sinking the sprue bush deep into the mould plate assembly. The actuating mechanism must be incorporated in the mould since the facilities are not provided on the machine for actuating from the fixed mould half side. (Fig 29)

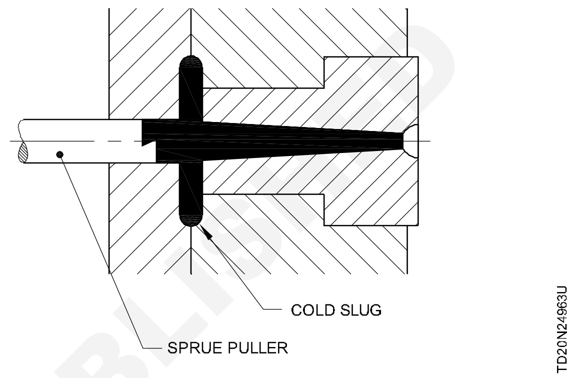
Fig 29



Sprue pullers: When the mould opens it is essential that the sprue is pulled positively from the sprue bush. In single impression moulds the sprue is pulled at the same time as the moulding is pulled from the cavity. But in a multi-impression mould the sprue is left in the sprue bush each time the mould is opened.

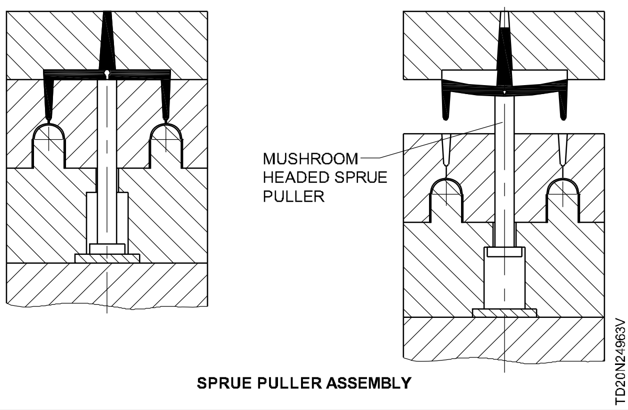
The common sprue pulling method utilises an undercut pin or an undercut recess situated directly opposite the sprue entry. The plastic material which flows in the undercut upon solidifying provides sufficient adhesion to pull the sprue as the mould is opened. The undercut forms part of the cold well design. The undercut may be of the reverse taper type. (Fig 30)

Fig 30



The cold slug well can have straight sides and grooves cut into it. A sprue pin is positioned behind the cold slug well, so that when ejection occurs the slug is ejected. A sprue pin is termed as a sprue puller when an undercut is provided on it. Another design utilises mushroom headed sprue puller. They are commonly associated with stripper plate design and under feed design. The mushroom head projects into the runner recess in both of the designs and is not contained in the cold slug well. (Fig 31)

Fig 31

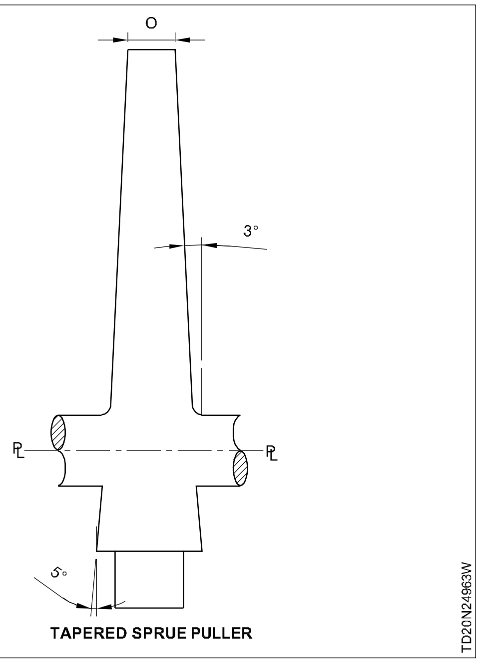


SPRUE PULLER ASSEMBLY

The typical sprue designs for various TPE hardness values are shown in the table below:

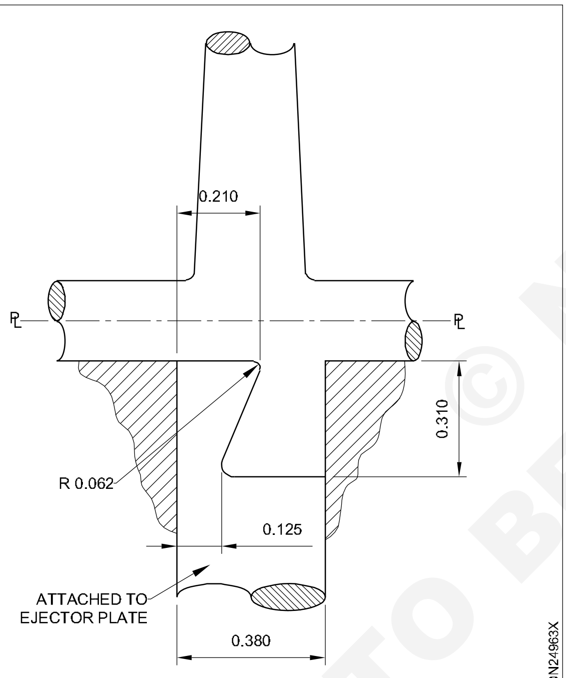
Typical TPE Hardness Range	Most Common Sprue Puller Types	Figure
> 50 Shore A	Tapered, Pin, Z-Type	32,33,35
40-70 Shore A	Undercut	34
5-40 Shore A	Pine Tree	36

Fig 32



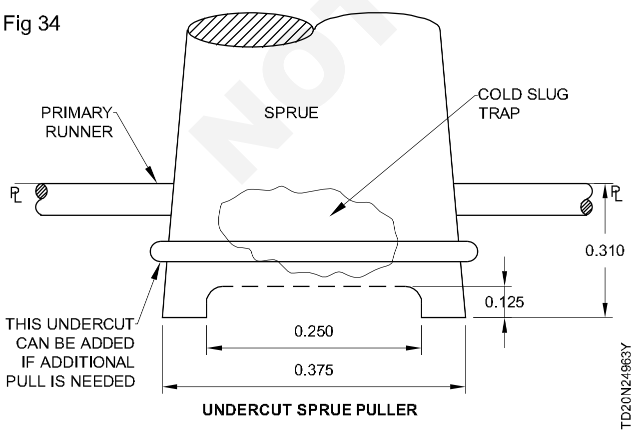
TAPERED SPRUE PULLER

Fig 33



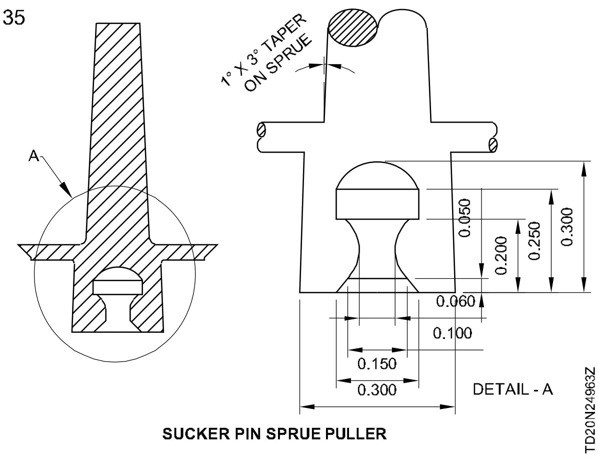
Z - PIN SPRUE PULLER

Fig 34



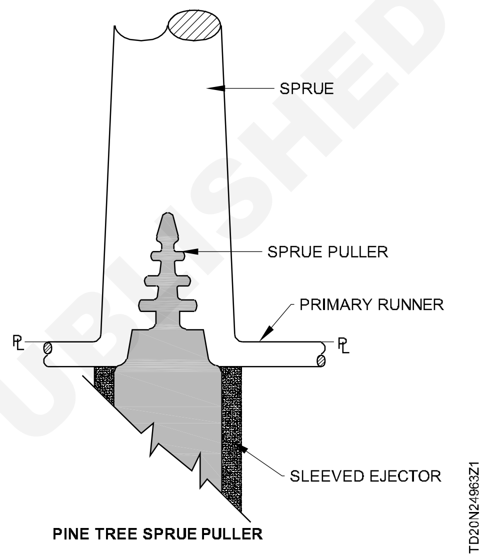
UNDERCUT SPRUE PULLER

Fig 35



SUCKER PIN SPRUE PULLER

Fig 36



PINE TREE SPRUE PULLER

Hot sprue bushings and extended nozzles may be used with GLS TPEs. In many molds, the sprue is the thickest wall section in the mold and will control the minimum cooling time. The use of a hot sprue, which may be viewed as an extension of the machine nozzle, can sometimes reduce cycle time. Extended machine nozzles may also be used to reduce sprue length and size. When hot sprues are used, the machine nozzle tip should be a free-flow nozzle rather than a reverse tip

*Spiral flow tests performed using 0.0625" thick x 0.375" wide channel at 400°F.

Sprue and Sprue Puller Design

The sprue should have sufficient draft, from 1° to 3° to minimize drag and sprue sticking. Longer sprues may require more taper (3° - 5°), as shown in Fig 32. Typically, the sprue diameter should be slightly larger than the nozzle diameter. An EDM finish is acceptable for most styrenic TPE materials. Permanent surface lubricant treatments have also been used successfully.

Sprue puller designs vary with the hardness of the material. The different sprue designs possible are shown in Fig 32 through 36.

Feed system

Objectives: At the end of this lesson you shall be able to
- **state the function of the various elements of the feed system**
- **state the various runner shapes and cross-section**
- **state the function of various types of gates**
- **select the best suited runner cross-section for the component**
- **calculate the runner size for an injection moulding machine**
- **differentiate between "centre gating" and "edge gating"**
- **name the different types of gates**
- **differentiate between `sprue gate' and "fan gate".**

The plastic material passes through the following before entering the impression.

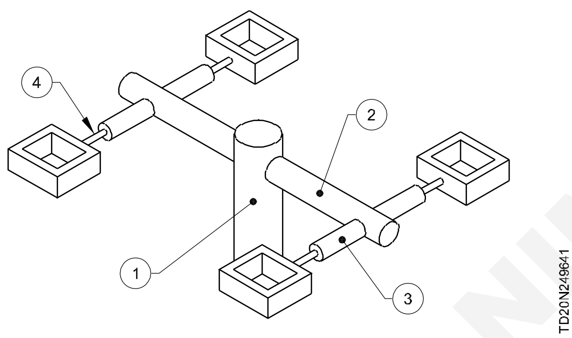
1 The sprue

2 Main runner

3 Branch runners

4 Gate (Fig 1)

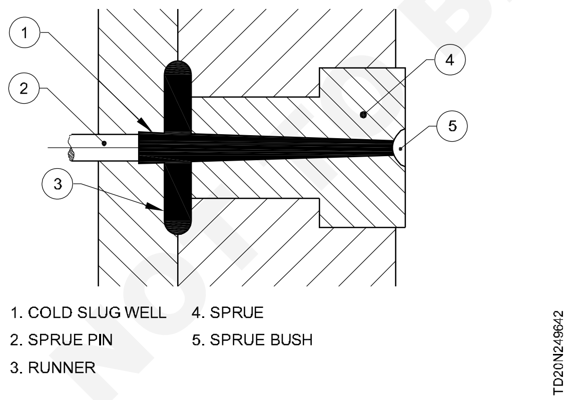
Fig 1



This is called the feed system.

Runner: The runner connects the sprue with the gate. The gate connects the runner and the impression. The runner will be positioned in the parting surface. (Fig 2)

Fig 2



Runner shapes and cross-section: The following are the runner shapes normally used.

- Fully round (Fig 3)
- Trapezoidal (Fig 4)
- Modified trapezoidal (Fig 5)

Fig 3

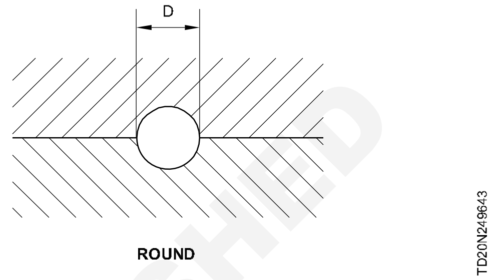


Fig 4

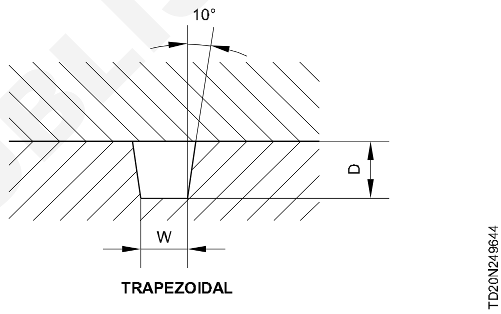
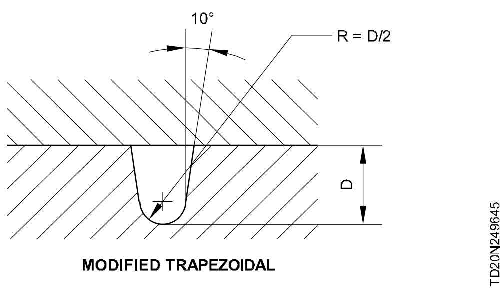


Fig 5



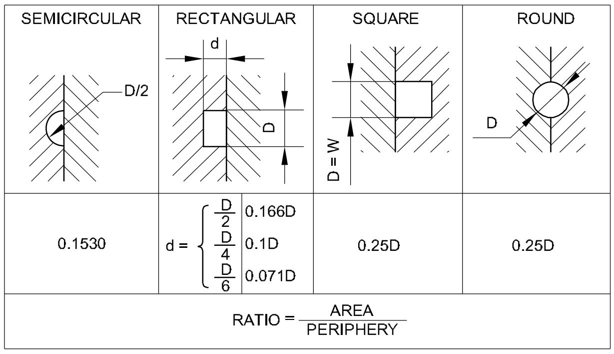
The runner should provide

- **maximum cross-sectional area:** This is necessary to avoid large pressure drop during injection.
- **minimum contact area:** This is necessary to avoid cooling and solidification of plastic during injection. The efficiency of the runner is given by the ratio area/ periphery. Higher the value, the greater the efficiency.

  By providing 10° angle on the runner wall, square section can be changed to a trapezoidal section. When the plastic melt flows through the feed system, the melt adjacent to the cold mould surface rapidly loses heat and solidifies. The material which follows passes through the centre of this solidified material.

For moulds with complex parting surfaces the semicircular channels of the round runner are used. For multiplate moulds the trapezoidal or modified trapezoidal section is used.

Fig 6



Runner size: The following factors are considered while deciding the runner size.

The cross-section and volume of the moulding: The cross-sectional area of the runner must be sufficient to permit the melt to pass through and fill the impression before the runner solidifies.

The distance of the impression from the main runner or sprue: As distance increases resistance to flow increases.

Runner cooling considerations: The larger the cross-sectional area of the runner, the longer the period the material takes to cool sufficiently to be ejected.

The solidified material acts as an insulator and maintains the temperature of the central region. Therefore, the gate should be positioned in line with the centre of the runner. This allows the gate to receive material from the central flow stream. This condition is achieved with fully round runner. In trapezoidal section the gate cannot be positioned in line with the central stream. The cost of mould having round runner is higher because the channels in the two mould plates are to be accurately machined and matched. A fully round runner system is preferred for simple two plate moulds with flat parting surface. The recommended runner diameter is 10 mm. But "for material with higher viscosity the runner diameter can be up to 12.5 mm".

Example: Rigid PVC, Acrylics

Runner diameter calculation

d = DIAMETER OF RUNNER IN mm

M = MASS OF THE MOULDING IN g

l = LENGTH OF RUNNER IN mm

Recommended runner diameters

Material	Recommended runner dia (mm)
ABS, SAN	4 to 10
ACETAL	3 to 10
ACRYLIC	7.5 to 10
NYLON	1.5 to 10
PC	4 to 10
PE	1.5 to 10
PP	4 to 10
PS	3 to 10
PVC (Plasticised)	3 to 10

Main Runner	PS		ABS	
Length (mm) from sprue	Main Runner	Sec. Runner	Main Runner	Sec. Runner
Upto 75	5	4	6.5	5.5
75 150	6.5	5.5	8	7
150 225	8	6	9.5	8
225 300	9.5	8	11	9.5
300 and above	9.5	8	12	11

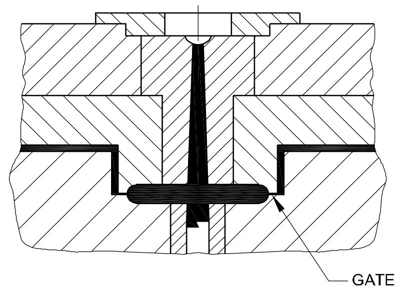
Long runners regulate increased diameter. All main runners in a mould should be equal in diameter. Diameter of all the secondary runners should be atleast 0.7 mm less than the diameter of the main runner. Mass of runner system in `g' for material density with 1g/cm.

DIA mm	Mass per 100 mm	
	Round	Trapezoidal
2	0.0314	0.0366
3	0.704	0.0823
4	0.1256	0.1463
5	0.1963	0.2286
6	0.2826	0.3292
9	0.6359	0.7407
12	1.1304	1.3167
15	1.7663	2.0574
18	2.5434	2.9626

The mass for runner for other materials can be obtained by multiplying the above values by the relative density of the particular material.

Gate (Fig 7): Gate connects the runner with the impression. Its cross-sectional area is very small compared with the rest of the feed system.

Fig 7



The small cross-sectional area is necessary for the following reasons.

- The gate should solidity soon after the impression is filled. This prevents the material from flowing out when injection pressure is withdrawn.
- De-gating will be simple.

– Only a small de-gating mark will remain on the component. Appearance of the component will not be spoiled.

– Multi-impression mould can be filled with better control.

The reduced cross-sectional area increases velocity. Due to this friction increases and this friction develops heat. This helps to maintain heat at the point of injection.

Caution

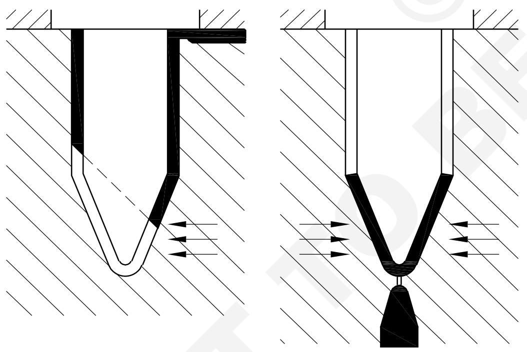
Care should be taken to avoid very high velocities. This may lead to super heating of the material.

Positioning of gate: The position of the gate should ensure even flow of material in the impression. The filling should be uniform. The advancing material front should spread out and reach the extremities of the impressions. At the same time, the gate position should avoid the formation of two or more advancing fronts. if two or more advancing fronts are formed they meet to form a weld line. The weld line will have interior mechanical strength. Normally the gate is positioned in the thickest part of the component. If material is fed from the thin part it cools faster and the mould will not be filled.

The gate should be located where it is easily accessible so that the de-gating will not damage the component.

Centre gating (Fig 8): Centre gating is ideal for sections with cylindrical cross-sections. Side gating in this case will lead to the formation of weld lines. When the core is slender side gating may deflect the core or may move it out of position. This will result in non uniform thickness of side walls.

Fig 8

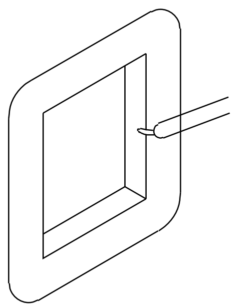


Edge gating (Fig 9): The edge gating should be positioned such that the flow immediately meets a restruction. If the gate is positioned in the centre, the melt on entering at high velocity jets and quickly solidifies on reaching the cool mould walls. (Fig 10)

More material then enters and flows around the original jetted material. Visible flow line will be formed on the component. The material along the flow lines is not homogeneous and is similar to weld lines. The mechanical strength of the component will be inferior.

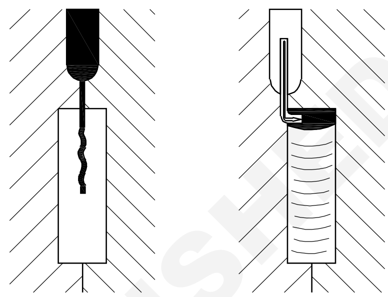
This can be overcome by overlap feeding. The injected material impinges on the opposite face of the impression. This forms an advancing front which fills the impression. The moulding will be free from flow lines.

Fig 9



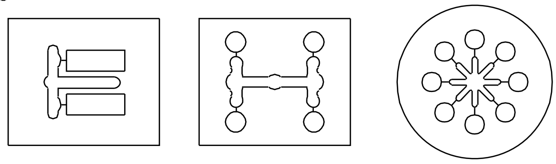
RECTANGULAR EDGE GATE
GATE MACHINED IN TO CAVITY PLATE

Fig 10



Balanced gating (Fig 11): The gating of multi-impression moulds should be balanced. It is necessary to have all the impressions filled simultaneously. This method is adopted when a balanced runner system cannot be used. If the runner layout is not balanced, once the runner system is filled the impression close to the sprue will fill first and those at the farthest end will fill last. Thus some impression may get over-packed while some will be left under filled. To achieve balanced filing greatest restriction to flow is caused to the impression nearest to the sprue. The restriction is reduced as the distance from the sprue increases. There are two ways to vary the restrictions.

Fig 11



– By varying the land length of the gate.

– By varying the cross-section of the gate.

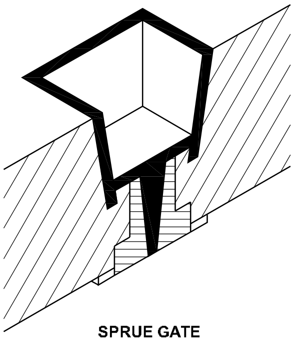
Balanced gating is achieved by trial and error. The trial is started by keeping the land length constant and keeping the width of gate to the minimum. The mould is tried out with a short injection stroke, so that short mouldings are obtained first.

Impressions which are filled will be known. The other gate widths are progressively enlarged, until balanced filling is obtained.

Types of gates

Sprue gate (Fig 12): When the moulding is fed directly from a sprue, the feed section is called sprue gate. This gate leaves a large gate mark on the moulding. This gate is preferred for tubular mouldings when more than one impression is required in a single two plate mould.

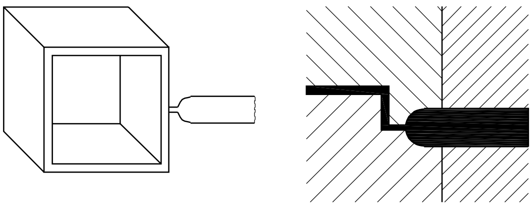
Fig 12



SPRUE GATE

Rectangular edge gate (Fig 13) : This is a rectangular channel milled in one mould plate to connect the runner to the impression.

Fig 13



MOULDING WITH GATE ATTACHED

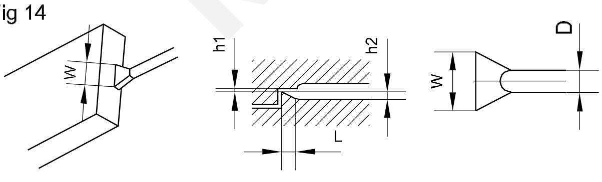
The advantages are

- the cross-section is simple and easy to machine
- gate dimensions can be maintained very accurately
- the gate dimensions can be quickly and easily modified
- all general plastics can be moulded through this gate.

The disadvantages with this type of gate is that it leaves a large gate mark on the component.

Fan gate (Fig 14): This is a type of edge gate. The fan shape spreads the melt flow as it enters the impression. Uniform filling is obtained. Flow marks are less. Fan gate is preferred for large thin-walled mouldings. This gate can be used for all types of conventional plastics. It leaves a large gate mark on the moulding.

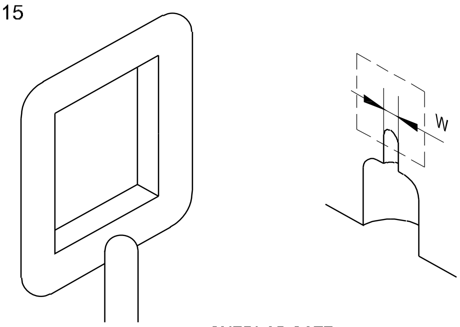
Fig 14



FANGATE

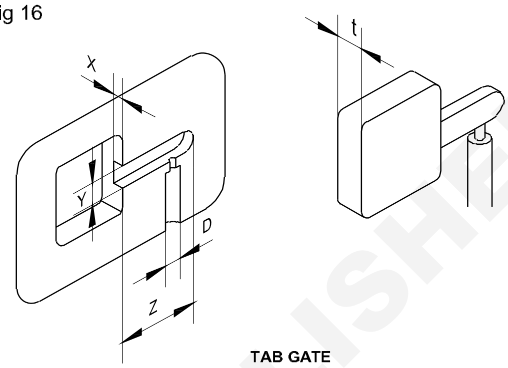
Over-lap gate (Fig 15): This is a variation of the rectangular gate. It is used for block type mouldings. The melt impinges against the opposite face and jetting effect is avoided.

Fig 15



OVERLAP GATE

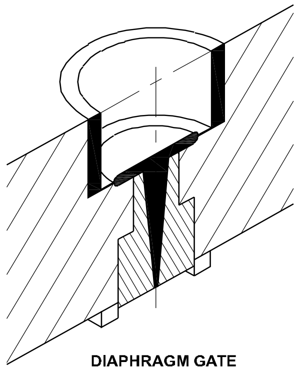
Fig 16



TAB GATE

Diaphragm gate (Fig 17): This gate is used for single cavity tubular shaped mouldings. The sprue leads to a circular recess which is slightly smaller than the diameter of the component. This acts as a runner and allows material to flow radially from the sprue to the gate.

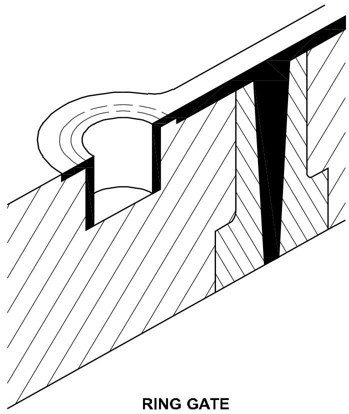
Fig 17



DIAPHRAGM GATE

Ring gate (Fig 18): The function is similar to diaphragm gate. The gate provides for all around feeding on the external periphery of the moulding.

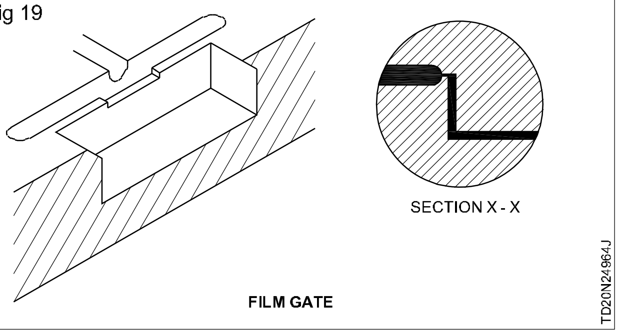
Fig 18



RING GATE

Film gate (Fig 19): It is a long rectangular gate. It is used for moulding large thin-walled mouldings. Warpage free components are obtained.

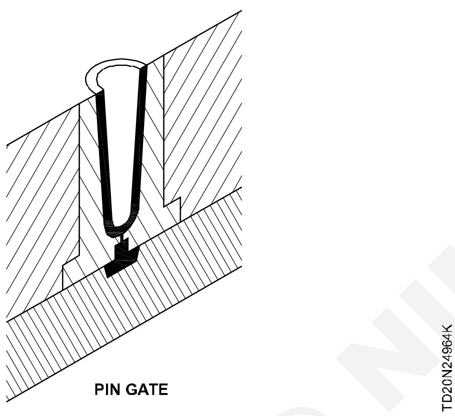
Fig 19



FILM GATE

Pin gate (Fig 20): This is a circular gate of very small diameter. It is used for feeding into the base of components. No visible gate mark will be present in the component after degating. Because of this it is preferred over sprue gates..

Fig 20

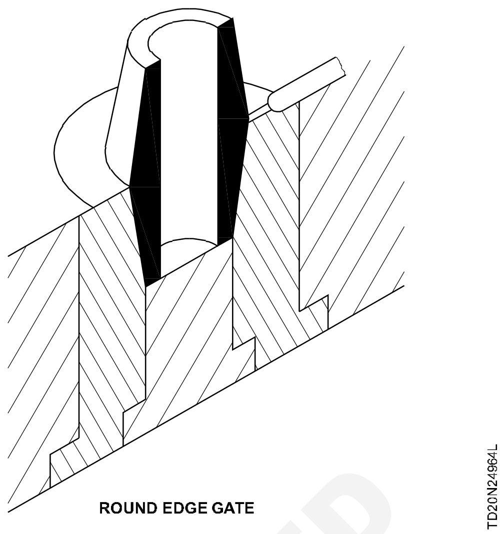


PIN GATE

Round edge gate (Fig 21): This gate is made by machining semicircular channel in both mould plates. This type of gate is generally used for components with minimum wall thickness of 4 mm. The relatively large gate ensures that the gate remains open sufficiently to allow the back pressure to be applied. This avoids sink marks.

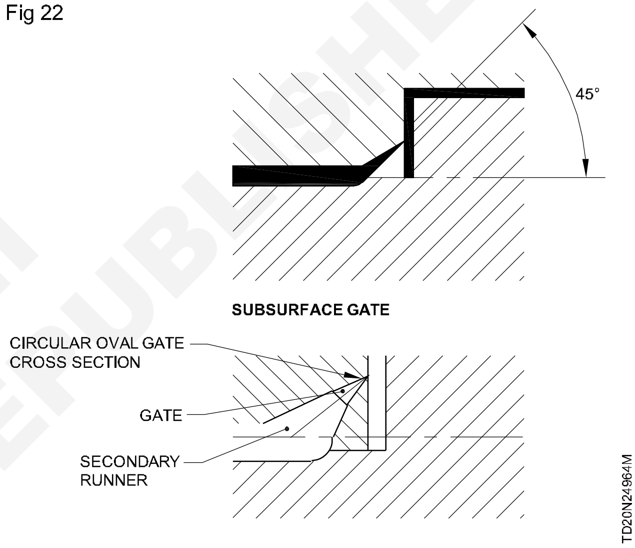
Sub-surface gate or submarine gate (Fig 22): This is a circular or oval gate. It feeds into the impression below the parting surface of the mould. Therefore it is called sub surface or submarine gate.

Fig 21



ROUND EDGE GATE

Fig 22



The advantages are

- the gate is machined on only one plate. Therefore dimensions can be maintained accurately.
- the gate gets sheared from the component during its ejection.
- only a very small de-gating mark is left in the component.

Shrinkage (Component Tolerance)

Objectives: At the end of this lesson you shall be able to
- **state the shrinkage in moldings**
- **list the factors governing shrinkage**
- **determine core and cavity dimensions**
- **state what is drafting in molding.**

Shrinkage in Moldings

When a molding cools, it contracts by an amount depending on the material being processed. This contraction is called shrinkage in other words, shrinkage can be defined as

The characteristic of a molding to contract in its size, while cooling.

The size of the molding hence is always smaller than that of the cavity and core

Factors governing shrinkage

SHRINKAGE of plastic depends on material and its processing. Various factors that affect shrinkage are:

1 Basic plastic material
2 Filler used and percentage
3 Part wall thickness
4 Melt temperature
5 Mould temperature
6 Injection pressure
7 Injection time
8 Hold on pressure
9 Hold on time
10 Gate areas

Shrinkage increases with

- Increase in melt temperature
- Increase in mould temperature
- Increase in wall thickness
- Low injection pressure

Shrinkage decreases with

- Low melt and mould temperature
- High injection pressure
- Long injection time
- Filler material and its content

Can be Specified as inch / inch / inch or in %

Shrinkage depends on the heat expansion and compression of the plastic during injection.

Heat expansion: Plastic has a large volume when it is hot and less volume when it is cold. The amount of expansion is directly proportional to the temperature of the plastic material. The amount of heat expansion is different for different plastics but it can also vary from supplier to supplier.

Compression during injection: The volume of plastic is decreased when it is under compression. The decrease is directly proportional to the injection and the holding pressure.

Relation between heat, pressure and shrinkage

The following relation exists between heat, pressure and shrinkage

1 Higher melt temperature with higher injection pressure gives less shrinkage.
2 Higher melt temperature with lower injection pressure gives more shrinkage.
3 Lower melt temperature with higher injection pressure gives the minimum shrinkage.

Core and Cavity dimensions

The ideal condition is to provide right core and cavity dimensions, so that, the part can be molded with the lowest temperature and fastest cycle time. Usually, this can be achieved by trial and error. This method is time consuming, but, it will give a dimensionally correct part. A designer can use the shrinkage data given by the manufacturer and also, his experience as starting point. The table below gives the shrinkage factor for different plastics to be used as guide line. All the part dimensions can be multiplied by this shrinkage factor to get core and cavity dimension.

Shrinkage for different plastics

Table 1

Sl.No.	Material	Shrink (mm/mm)
1	ABS(high impact)	0.005-0.007
2	ABS(heat resistant)	0.004-0.005
3	ABS(medium impact)	0.005
4	Acetal	0.02-0.035
5	Acrylic(general purpose)	0.002-0.009
6	Acrylic(heat resistant)	0.003-0.10
7	Acrylic(high impact)	0.004-0.008
8	EVA	0.010-0.030
8	Ionomer	0.003-0.020
9	Nylon 6/6	0.10-0.025
10	Nylon 6	0.007-0.015
11	Nylon 6/10	0.010-0.025
12	Nylon 11	0.010-0.025
13	Nylon 12	0.008-0.020
14	Nylon (glass filled)	0.005-0.010
15	Poly butylenes(molded)	0.02
16	Poly butylenes(aged)	0.04
17	Polycarbonate	0.005-0.007
18	PET	0.003-0.005
19	Polyester(0.5-1.25mm thick)	0.006-0.012
20	Polyester(1.25-2.50mm thick)	0.012-0.017
21	Polyester(2.50-4.50mm thick)	0.016-0.022
22	Polyester PBT	0.010-0.020
23	Polyester PET (30% GF)	0.001-0.002
24	Polyester PBT (30% GF)	0.003-0.005
25	Polyethermide	0.005-0.007
26	Polyethylene (low dense)	0.015-0.035
27	Polyethylene (high dense)	0.015-0.03
28	PPO/Styrene Co(Noryl)	0.005-0.007
29	Polypropylene	0.010-0.030
30	Polystyrene (GP)	0.002-0.008
31	Poly styrene(heat resistant)	0.002-0.008
32	Poly styrene(high impact)	0.003-0.006
33	Poly Sulphone	0.008
34	Poly urethane	0.010-0.020
35	PVC (Rigid)	0.002-0.004
36	PVC (semi-rigid)	0.005-0.025
37	PVC (Flexible)	0.015-0.030
38	SAN	0.002-0.006

Achievable tolerance for different materals

Tolerances given in Table 2 can be achieved at reasonable cost. Tighter tolerances can be achieved at greater cost.

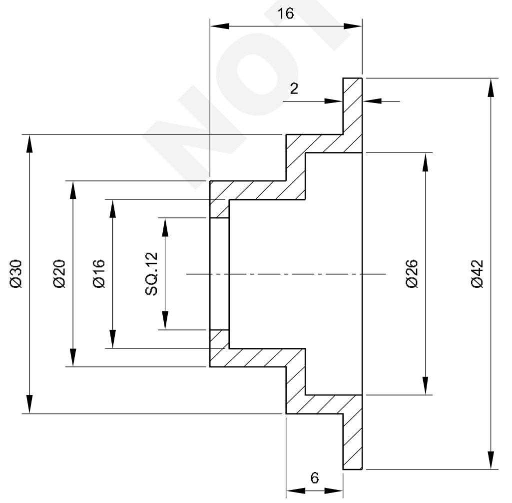
Table 2

Material	Min.	Add per mm. Dimension			Holes	Across	Flat	Concentric
	To. 25 mm	(25 - 75) mm	(75 - 150) mm	(150 - 250) mm	TO - 6 mm	PL - ADD	Total/ 25 mm	T.I.R./ 25 mm
A.B.S	0.051	0.025	0.038	0.051	0.025	0.051	0.051	0.051
Acetal	0.076	0.038	0.051	0.076	0.051	0.051	0.102	0.102
Acetate	0.051	0.025	0.038	0.051	0.051	0.051	0.051	0.051
Acrylic	0.051	0.025	0.038	0.051	0.051	0.051	0.051	0.051
Butyrate	0.051	0.025	0.038	0.051	0.508	0.508	0.051	0.051
E.V.A.	0.102	0.051	0.076	0.102	0.076	0.076	0.178	0.178
Nylon	0.076	0.038	0.051	0.076	0.051	0.051	0.127	0.127
Polycarbonate	0.051	0.025	0.038	0.051	0.038	0.051	0.051	0.051
Polyethylene	0.102	0.051	0.076	0.102	0.076	0.076	0.178	0.178
Polypropylene	0.102	0.038	0.051	0.076	0.076	0.076	0.127	0.127
Polystyrene	0.051	0.025	0.038	0.051	0.025	0.051	0.051	0.051
Propionate	0.051	0.025	0.038	0.051	0.051	0.051	0.051	0.051
S.A.N	0.051	0.025	0.038	0.051	0.025	0.051	0.051	0.051
Thermo elastic	0.152	0.051	0.102	0.127	0.102	0.076	0.254	0.254
Vinyl (Flexible)	0.152	0.051	0.102	0.127	0.102	0.076	0.254	0.254
Vinyl (Rigid)	0.102	0.025	0.038	0.051	0.076	0.051	0.076	0.076
Kraton	1.295	0.025	0.038	1.295	0.025	0.051	0.051	0.051

Example: Cavity/Core Dimensions

Calculate the dimensions for the cavity and core for the given component. (Fig 1)

Fig 1



The shrinkage is 0.5%.Therefore, the cavity and core dimension will as in Table 3.

Cavity Table 3

Component Dimension	Shrinkage allowance	The dimension to be maintained
ϕ42	42 x 0.5/100 = 0.21	ϕ42 + 0.21 = ϕ42.21
ϕ30	30 x 0.5/100 = 0.15	ϕ30 + 0.15 = ϕ30.15
ϕ20	20 x 0.5/100 = 0.10	ϕ20 + 0.10 = ϕ20.10
16	16 x 0.5/100 = 0.08	16 + 0.08 = 16.08
6	6 x 0.5/100 = 0.03	6 + 0.03 = 6.03
2	2 x 0.5/100 = 0.01	2 + 0.01 = 2.01

Core

Component Dimension	Shrinkage allowance	The dimension to be maintained
ϕ 26	26 x 0.5/100 = 0.13	ϕ 26 + 0.13 = ? 26.13
ϕ 16	16 x 0.5/100 = 0.08	ϕ 16 + 0.08 = ? 16.08
Sq 12	12 x 0.5/100 = 0.06	Sq 12 + 0.06 = 12.06
16	16 x 0.5/100 = 0.08	16 + 0.08 = 16.08
8	8 x 0.5/100 = 0.04	8 + 0.04 = 8.04
2	2 x 0.5/100 = 0.01	2 + 0.01 = 2.01

Draft

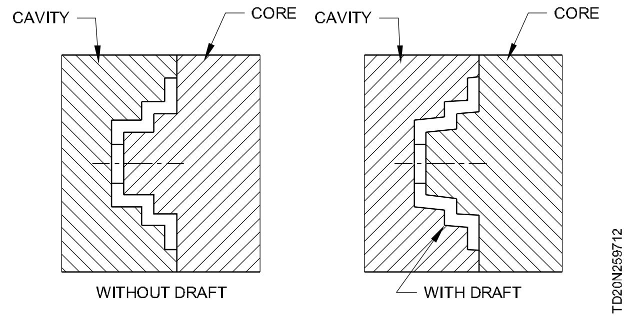
Draft is an angle applied to a plastic forming surface to ease the release of the moulded part without distortion. The same draft angle can be given to all interior and exterior surfaces to maintain the uniform wall thickness.

A draft angle can be specified as plus or minus. However the dimension of the draft should fall within the part tolerance.

Recommended draft angle

- degree --- Very small details under 0.040 tall that will get polished. The act of polishing will apply some draft.
- 1/4 degree --- Emergency use only. Deep ribs, one internal side of a box where the other sides have good draft, bosses ejected by sleeves.
- 1/2 degree --- Use sparingly and for good reason. Ribs, one internal side of a box, snaps, hooks, etc.

Fig 2



- degree --- Standard draft, all features.
- degree --- Standard draft, very light texture, cavity side to ensure good release.
- 3.0 degree --- Textured faces, faces that are in common with a shutoff.

Draft and its use is one of the most basic concepts in the design of the part, mould design and building and moulding.Improper use of draft will add continued cost and complexity to the manufacture of the plastic part. Proper application and use of draft will make moulding a difficult part easy and more economical

The reason for the using the same draft angle is to maintain a uniform wall thickness through the plastic part. A uniform wall is the most important building block, the most basic and important consideration for reducing stresses in the moulded block. Maintaining a normal wall thickness with the same draft angle should be the aim of the part designer.

Temperature controlling of moulds

Objectives: At the end of this lesson you shall be able to
- **state what is temperature controlled mould**
- **list the factors effecting the cooling of moulds**
- **explain the lay out and sizing of cooling channel**
- **brief the cooling of core, cavity , bolster and sub inserts**
- **brief the cooling requirement and its calculation.**

Introduction

Mould shapes the hot plastic injected into the shape of the impression. The heat of the melt is transferred to the mould and the moulding solidifies. Since, the mould is cold, the material solidifies fast initially. As the mould gets heated up, time required to cool the plastic also increases. If, the part is not sufficiently cooled inside the mould the ejecting force will deform the part. In a moulding operation, cooling time consumes almost 80% of the total cycle time. So, in order to considerably reduce cycle time, we should find a way to cool the mould fast. This can be easily achieved by circulating coolants around the part area.

Normally, water is used as coolant. We, also, should remember that if the material solidifies quickly, it may not reach the farther end of the impression. A compromise between these contradicting requirements should be considered. It is, also, important to note that approximately 70% of the heat is transferred to the core, 20% to the cavity and the remaining 10% is carried by the part while being ejected. The heat transferred to cavity is less, because, as soon as the melt touches the cold cavity surface, it shrinks and holds onto core surface. Therefore, we should concentrate more on core cooling than cavity cooling. Cooling should be uniform through out the part. Cooling channel should be placed closer to hotter regions and farther away in cold regions. Rapid and uniform cooling is achieved by a sufficient number of properly placed cooling lines. The operating temperature for a particular mould will depend on a number of factors, like type and grade of material to be moulded, length of flow within the impression, wall section of the moulding, length of the feed system etc. It is advantageous to use a slightly higher temperature than is required just to fill the impression, as this tends to improve the surface finish of the moulding by minimizing weld lines, flow marks and other blemishes.

The cooling holes or channels are termed 'water lines' and the complete system of waterways is termed as cooling circuit. Standard Units for the circulation of water are commercially available. These units are connected to the mould via, flexible hoses. The moulds temperature can be maintained within close limits with these units.

The desired location of the cooling lines in mould is in the core and cavity insert itself. They should be located close to the area where the maximum heat is. The holes should not be positioned too close to the impression as this causes a marked temperature variation across the impression, resulting in moulding problems. The recommended distance between holes is 2.5 to 3.5d. They also should not be placed close to the edge because of strength considerations. It is recommended that holes be drilled at least 6mm away from other openings in the case of longer holes, but for shorter ones it can be 4 to 5mm. The distance between the cooling channel and moulding surface is recommended as 1-1.5d. Standard drill diameter can be used.

The open end of the holes may be tapped for NPT (National Pipe Thread) or BSPT (British Standard Pipe Thread) as well as water connector. To improve the quality of thread the whole portion may be taperely reamed before threading. The standard taper is 3.5º per side

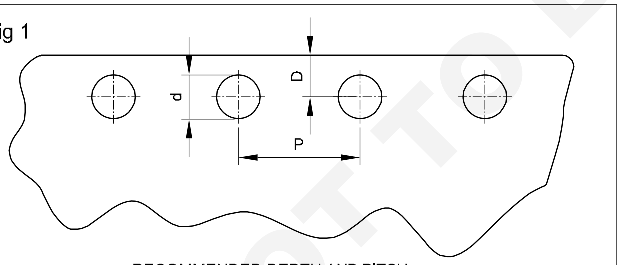
'O' ring fittings are used some times instead of taper thread fitting. They are costlier than taper fitting, but, have certain advantages:

Since there is no wedging action while tightening it can be very close to walls or other holes.

The chance of leaking is less. This is always recommended on moving parts where there is threaded plug coming off due to vibration.

The recommended depth and pitch is given below. (Fig 1)

Fig 1



RECOMMENDED DEPTH AND PITCH
d = DIAMETER OF WATERLINE = 6mm to 16mm
D = DEPTH OF WATERLINE = d to 2d
P = PITCH = 3d to 5d

Core and cavity plates should be at the same temperature. If there is considerable difference in temperature it may result difference in expansion of the plates which will cause misalignment and difficulty in mould opening. The parting line matching also will get affected.

Factors affecting the cooling of a mould

1 Inlet and outlet temperature difference. (Should be maintained with 3-5º)

2 Difference in temperature (coolant temperature versus Plastic melt)

3 Flow of coolant from IN to OUT.

4 Thermal conductivity of mould material.

5 Amount of heat to remove.

6 Total area of core and cavity.

7 Size and length of runner.

8 Arrangement and size of cooling circuit.

9 Diameter and number of inlet and out let.

10 Type of runner system. (Hot or cold)

11 Quality of coolant.

Layout and sizing of cooling channel in mould plates

Sometimes, in addition to, direct cooling of core and cavity inserts the mould plates are also cooled. The needs for separate cooling for mould plate cooling are:

1 When the channel is passing through the plate to the core or cavity insert.

2 When the core or cavity insert is cooled only by cooling the mould plate.

3 When slender core pins cooled by coming in contact with the cooling channel in the plate.

4 When simplicity of lay out is required.

5 When symmetry and number of circuits is required for even cooling.

Cooling integer-type mould plates

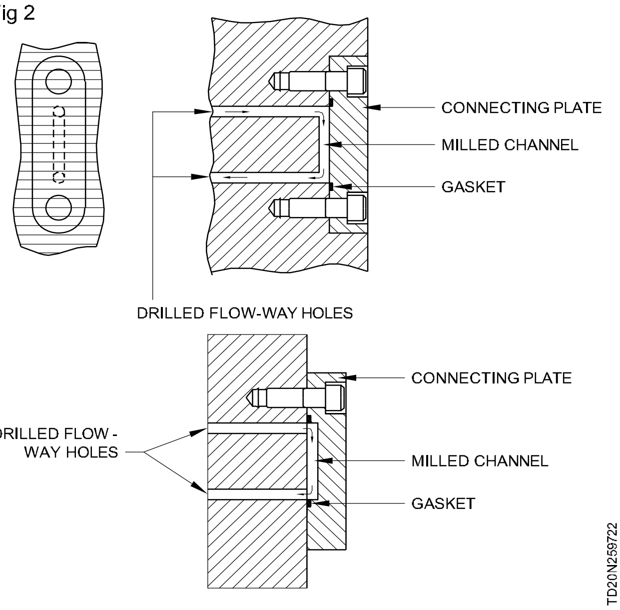
The temperature of a mould plate of the integer type is controlled by circulating water through holes drilled in the plate. The holes are normally interconnected to form a circuit. The circuit may be at one or more levels. The number of levels depends on the thickness of the mould plate.

Cooling integer type cavity plate

All the methods discussed for cooling integral plate are also applicable for cooling integral cavities. The easiest method is to drill two through holes, one on either side of the cavity and to connect these at one end by means of a flexible hose. The external connection can be avoided by interconnecting the two holes by means of an internal drilling. This forms a U - circuit and is useful for cooling long, narrow cavities. In case of a large area, shallow cavity the circuit can be positioned directly below the cavity. For deep cavities, the side walls also should be cooled. This forms a flow path of z-configuration. The mould plate is drilled, plugged and baffled. A multi-level system also can be is adopted.

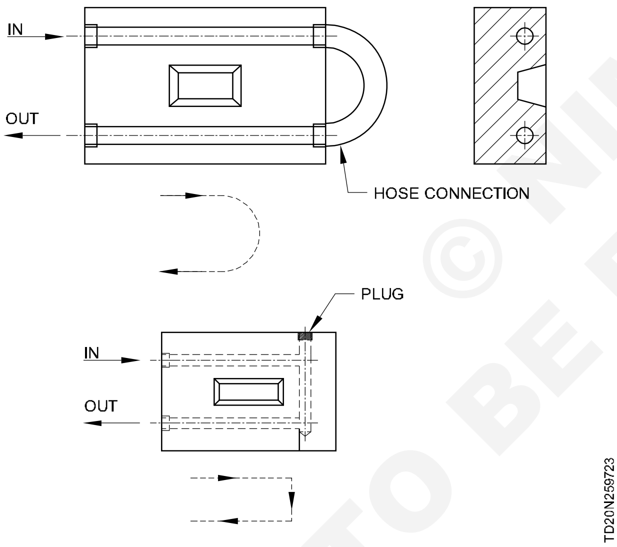
Through holes circuit connected outside (Fig 2)

Fig 2



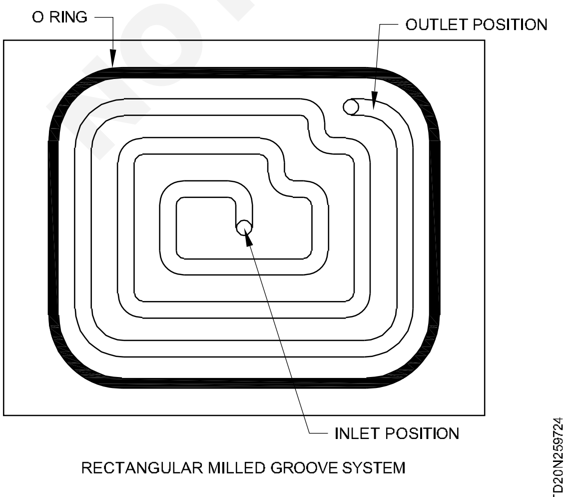
'U' Circuit (Fig 3)

Fig 3



Rectangular circuit (Fig 4)

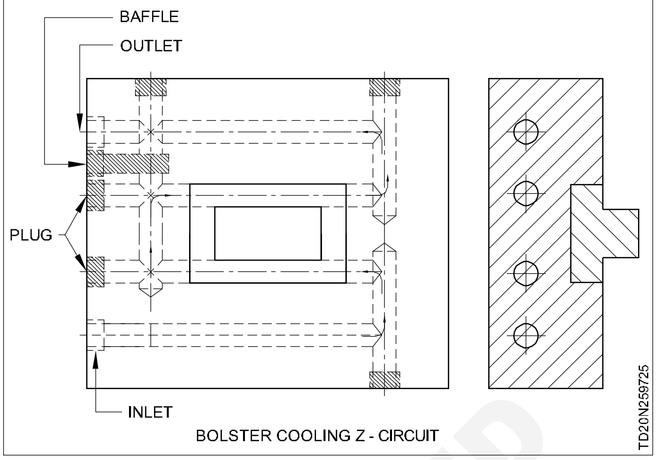
Fig 4



RECTANGULAR MILLED GROOVE SYSTEM

Rectangular wave circuit (Fig 5)

Fig 5



BOLSTER COOLING Z - CIRCUIT

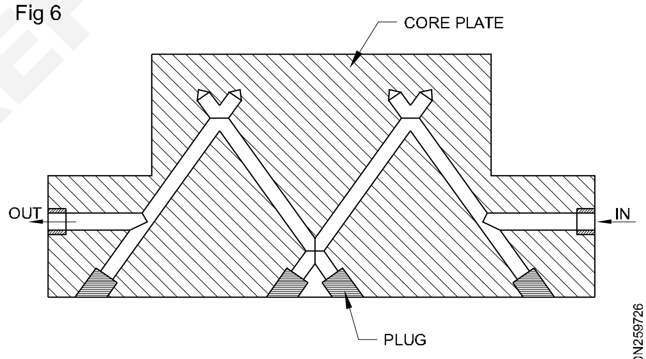
Cooling thick core plate

When the height is more, the single-level circuit is not sufficient to permit the coolant to transfer heat away from the core surface fast enough. The different methods used are:

Angled hole system (Fig 6)

Waterways are drilled at an angle from the underside of the core plate, so that, they interconnect at a point close to the surface. Each hole is plugged and inlet and outlet holes are drilled into the angled waterways.

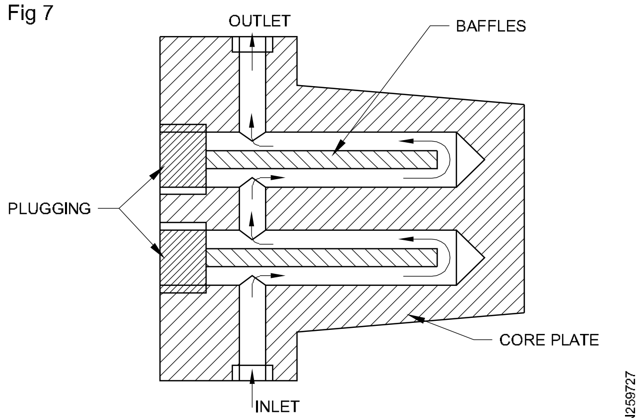
Fig 6



ANGLED-HOLE SYSTEM

Baffled - Straight hole system (Fig 7)

Fig 7



BAFFLED - STRAIGHT HOLE SYSTEM

Relatively, large diameter holes are bored at right angles to the rear face of the core plate, plugged and baffled. The borings are interconnected. Baffle drops typically designed where by several holes are in series. Therefore, they are not recommended for small cores of large length. The pressure drop in such designs results in low flow rates with large temperature rise in the cooling circuit. Bubblers, are often, preferred method for cooling small cores. Baffle hole dimensions are listed in Table 1.

Formula for equivalent hydra diameter

$$D_h = \frac{4A}{p}$$

$$\text{Where } A = \frac{\pi d^2}{4}$$

$$P = \pi d$$

Dh = Equivalent hydra diameter.

A = area cross section of the hole

P = Perimeter of the round hole

Table 1

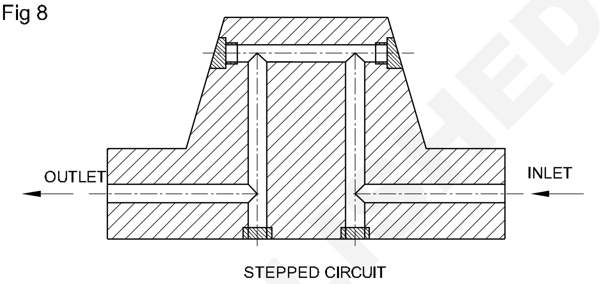
Baffle hole dimensions

Hole diameter (mm)	Inlet diameter (mm)	Plug size (inch) NPT	Blade Width (mm)	Relief (mm)
6	6	1/16 - 27	1.5	3
8	8	1/8 - 27	1.5	4
11	11	1/4 - 18	2.5	5
14	11	3/8 -18	2.5	7
17.5	11	1/2 - 14	2.5	9
24	14	3/4 - 14	3	12
28.5	17.5	1-11 1/2	3	14

Stepped circuit (Fig 8)

In this system, holes are drilled through the side wall of the core, parallel to the core face. These holes are carefully plugged and finished as they form part of the impression. For narrow cores, the above three circuits can be used singly. For wider cores, a number of identical circuits can be used at intervals along the core.

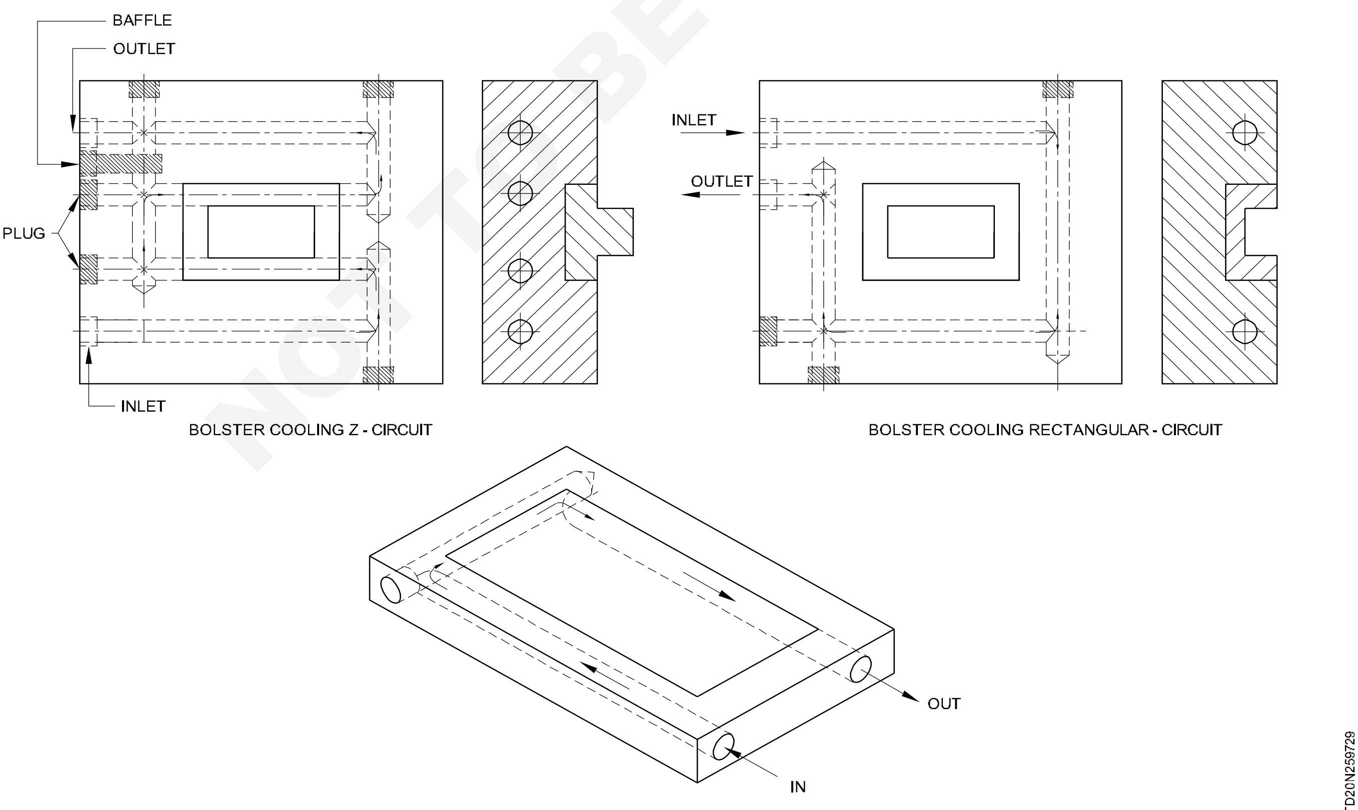
Fig 8



Cooling bolster (Fig 9)

The water ways are drilled through the bolster and are interconnected, either externally or internally to permit the circulation of a coolant. These water lines should be located close to the insert. A rectangular or z-shaped circuit is also used.

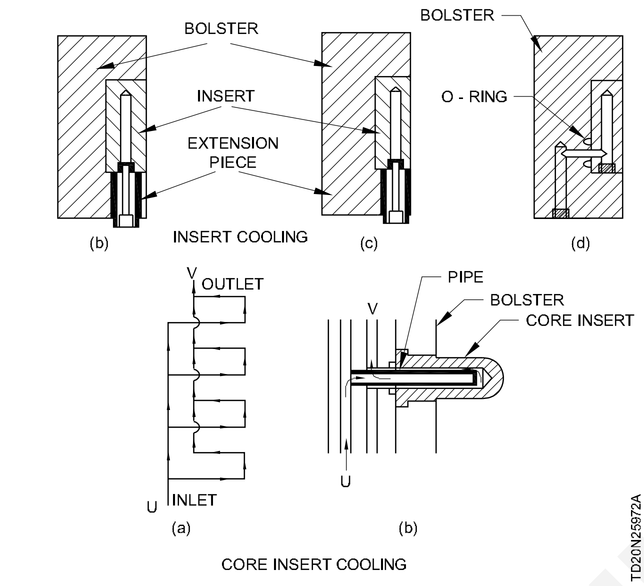
Fig 9



Cooling core and cavity inserts (Fig 10)

The method adopted for cooling inserts depends upon the shape of the insert. This is broadly classified as rectangular and circular. Rectangular insert circuit includes u-circuit, rectangular system, baffles, tubes, spiral, cascade etc and copper pipe system. For circular inserts in addition to the above coolant can be circulated around the insert through groves sealed with O-ring.

Fig 10

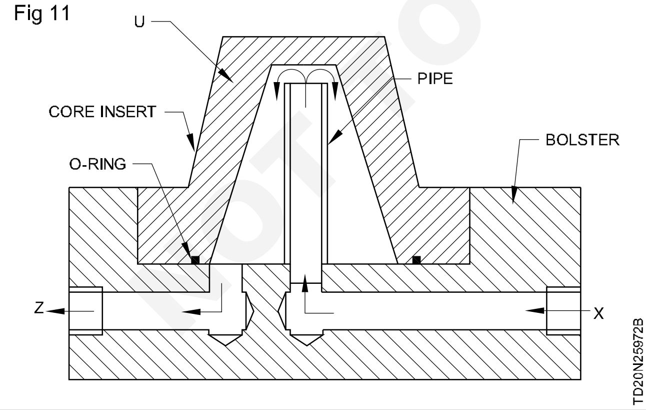


Deep core cooling system (Fig 11)

The important designs of cooling system are

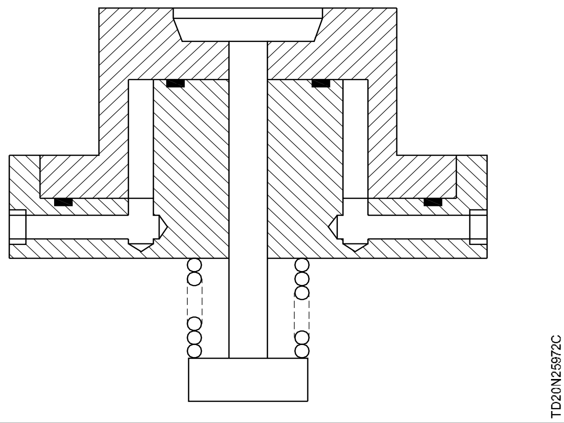
Deep chamber system: In this system a deep pocket is made below the moulding area. The insert fixed in a pocket and a 'o' ring is placed around the recess made for the water. A pipe is fixed at the centre of the recess as shown. The water entering through the pipe hit the centre of the recess and flashes out. The main problem with this system is the remarkable drop in flow rate.

Fig 11



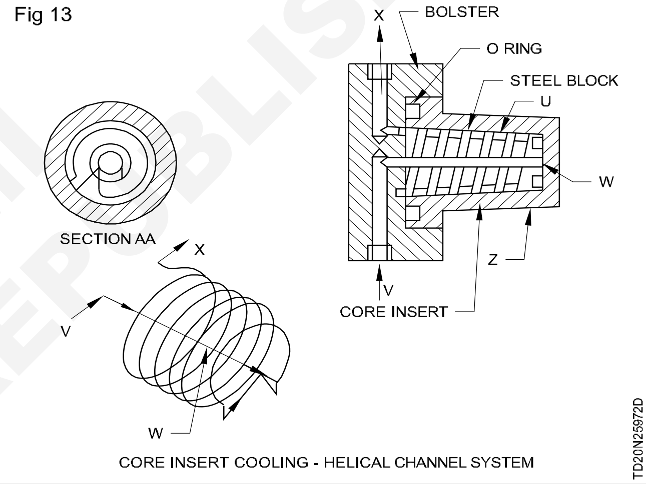
Deep chamber with central support system: This type of system has a center support for the material above the recess. 'o' rings are placed both on top and bottom as shown. This type of design is advantageous when a central support is required to avoid possible deflection or a valve type ejector is needed to eject the part. (Fig 12)

Fig 12



Helical channel system: (Fig 13) In this case the deep chamber is closed with an aluminum or brass plug with a helical channel. When single helix is used the water enters through the centre hole and spiral through the helix and comes out. In double helix the water can enter one helix and comes out through the other helix.

Fig 13



Baffled hole system: (Fig 14) In this system a brass sheet enters the hole dividing the volume into two equals. The sheet fixed to a tapered plug with marking on the reverse for positioning.

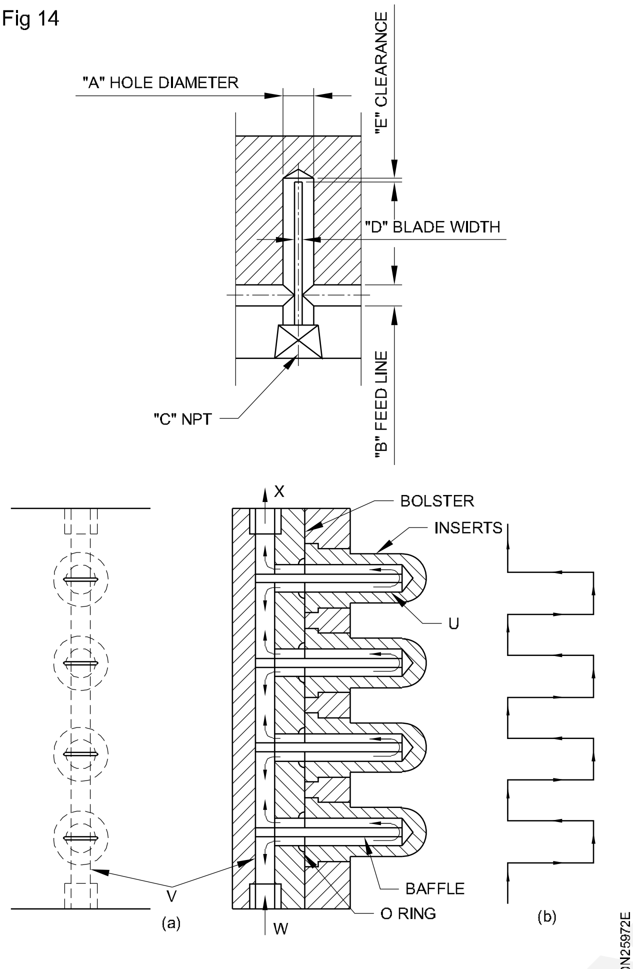
Bubbler system: (Fig 15) This design is same as the deep chamber system. But the pipe being small it is more suitable for small inserts. The tubes are either made of brass or copper.

Spiral plug system: (Fig 16) This one is same as helical system the deference being the pitch is like a thread. Both single and double starts are available.

Cascade water junctions: (Fig 17) In this the inlet and out let holes are made in a separate piece and it functions like bubbler. This is a self contained unit hence no 'o' ring required for fixing in the in the inert.

Heat transfer pins: (Fig 18) This type of systems is used when it is not possible to provide liquid cooling media. It is a simple high conductive metallic rod, one end inserted into the core insert and the other end touching water line. Some time the core itself made of high conductive materials like AMCO alloy or beryllium copper. Longer core pins in sleeve ejection are cooled by supplying water through the base plate.

Fig 14



BAFFLED HOLE SYSTEM FOR SMALL CORE INSERTS

Fig 16

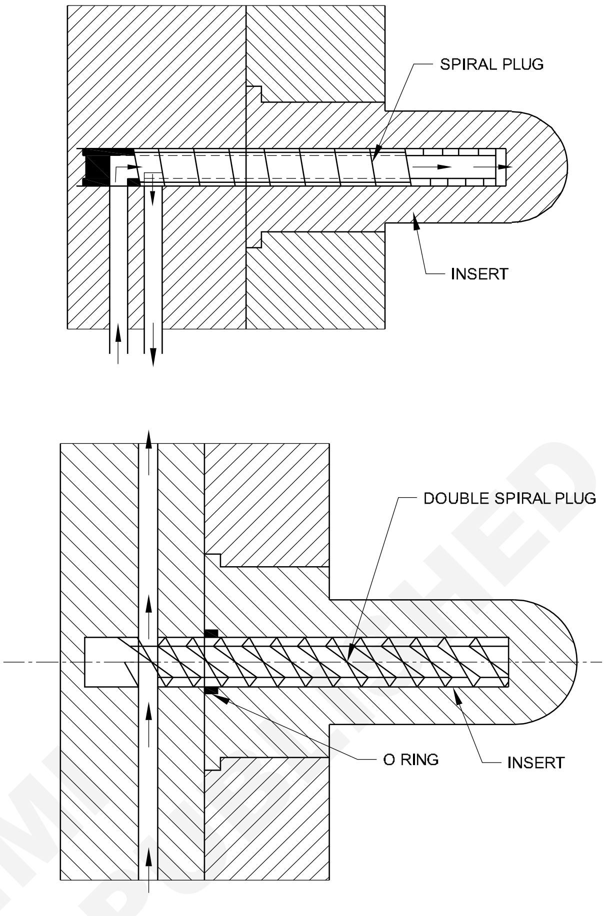


Fig 15

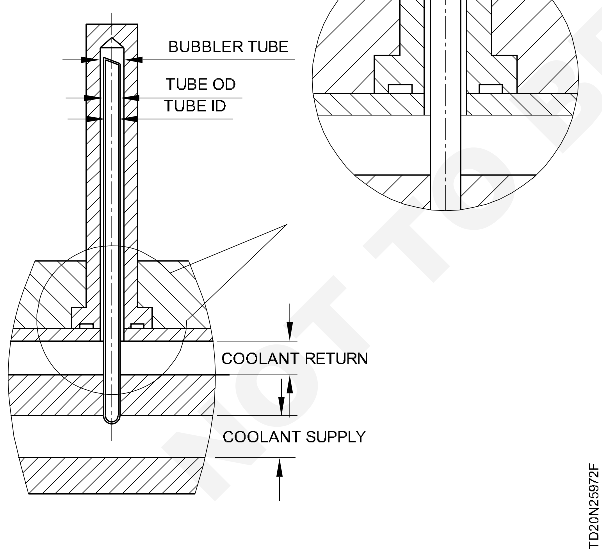


Fig 17

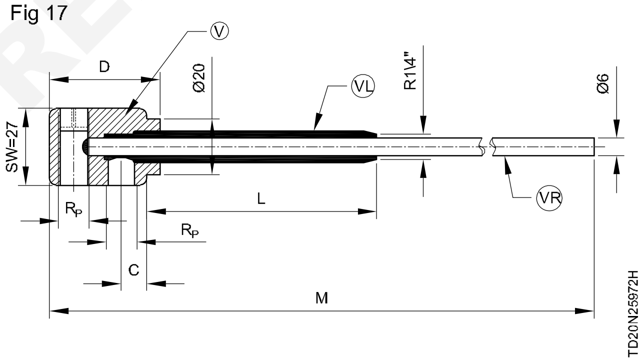
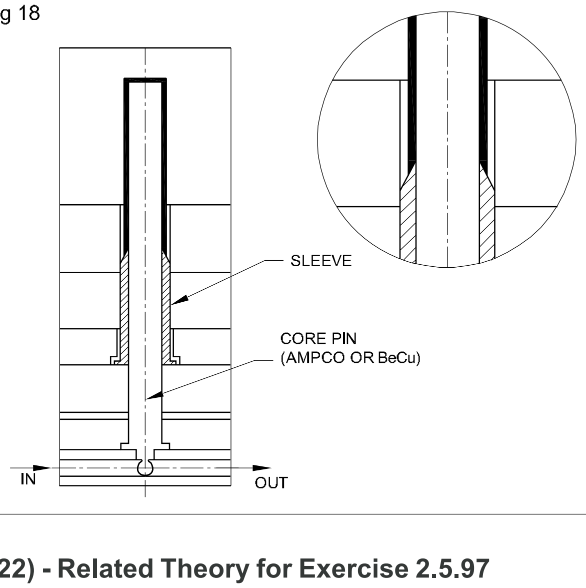
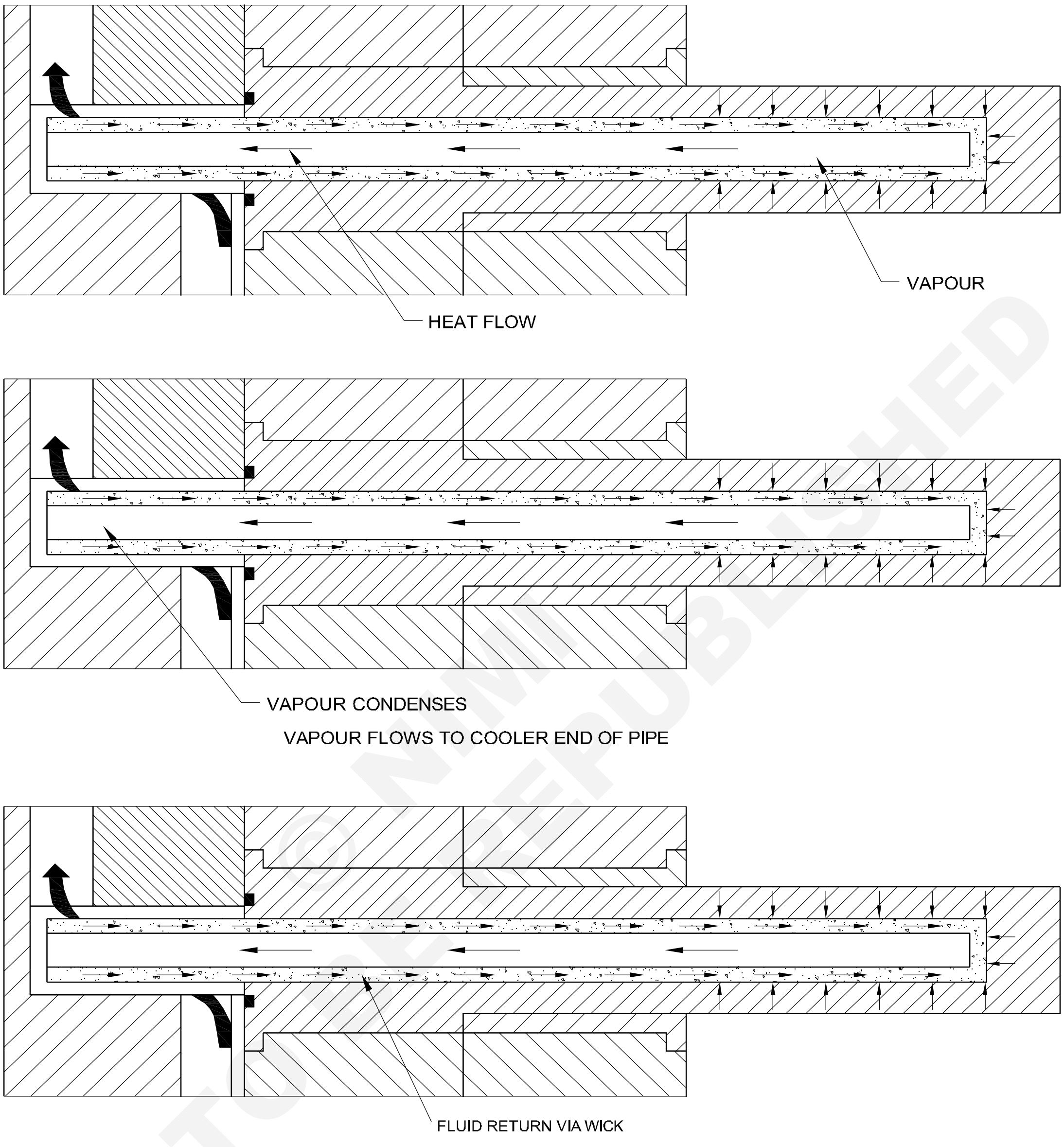


Fig 18



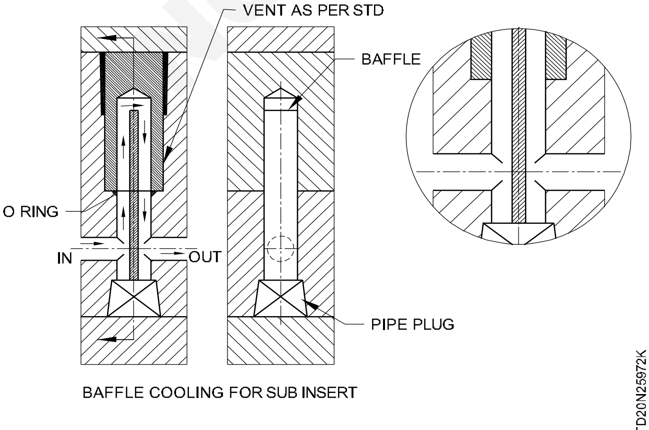
Heat transfer pipes: (Fig 19) these are pipes of standard length closed and sealed with gaseous medium for the very fast transfer of heat. The heat transfer pipes are fixed with less than 0.2mm clearance for better heat transfer.

Fig 19



Method of colling sub inserts (Fig 20)

Fig 20



Standard cooling items: Standard cooling items are available from various manufactures. Some of the items are discussed below.

Water connections

Moulds are drilled to provide a flow path through which water can be circulated. These holes are connected to the supply and return lines via standard connectors (hose connector). These are standard pipe fittings which are available in various sizes. It comes with two standard bore sizes. In one standard the bore diameter is less than the connecting hole. This type gives more strength to the threaded portion but it restricts the water flow. In the second type the bore matches that of plate but it weakens the threaded portion. The space available between the water ways also determines the size of the connector. A smaller connector can be inserted and removed with a smaller spanner. Different shapes and length of connectors are available. Some times the connector may be fitted in a counter bore so that it does not project out above the surface. This design prevents connector damage during shifting of mould.

The position of all inlet and outlet holes should be positioned either at the bottom or at the back of the mould. Connection at the front side will cause disturbance to the mould operator. The connections should not be at the top of the mould because in case of leakage water will get on to the polished face of the impression. All water connections are positioned such that no connectors and hoses interfere with the clamping of the mould on to the machine.

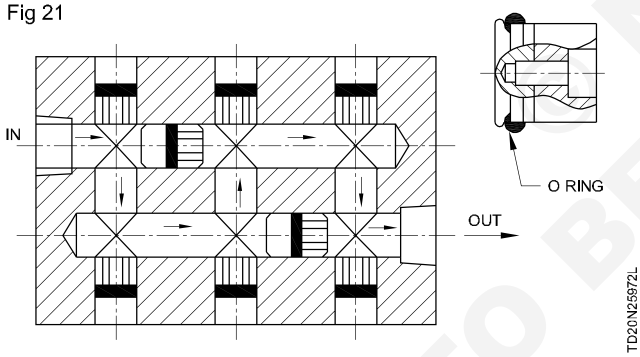
Taper plugs

Plugging the ends of certain holes is necessary to form a continuous circuit. Commercial plugs are used and these are available in the tapered BSPT or NPT range. The tapping size and length of thread is the same as that indicated for adapters.

Line plugs (Fig 21)

Thread-less brass plugs are used mainly used for diverting the water inside holes which are interconnected. It is an assembly of a serrated main body, an 'O-ring, a bush for compressing the 'o' ring and a screw to assemble these items.

Fig 21



Water manifold (Fig 22)

In this case all the inlets and out lets lines are connected to a manifold fixed on the mould. This will save connecting time during production as well reduce the chance of hose lines mix-up.

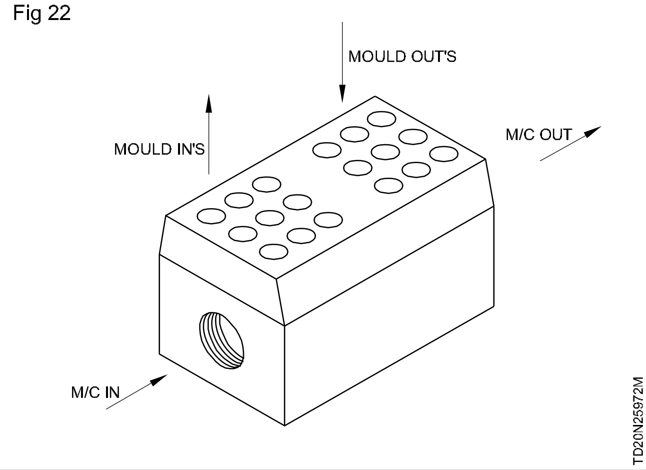
Mould cooling requirements and and calculations

Heat removal requirement of a particular moulding operation can be can be found out by two ways.

1 By detailed calculation based on the principles of heat transfer.

2 From empirical data accumulated by testing many different moulds, moulding different parts under different operating conditions.

Fig 22



There are three steps involved in the calculation of cooling time

1 Estimating the cycle time: Cycle time is usually given by the customer.80% of the cycle time is allotted for cooling.

2 Quantity of heat to be removed: depends on material and product requirements

3 Quantity of coolant required: depends on cooling efficiency

We can accurately calculate the heat input but the value depends on many factors which have to be assumed.

The estimated cycle time depends on

1 Geometry of part.

2 Wall thickness of part.

3 Ejection method adopted.

4 Machine parameters.(injection speed and pressure)

5 Characteristics of plastic material.(melt flow index, heat conductivity, moulding temperature)

The estimated cooling time depends on

1 Design cooling lay out.

2 Mould material.

3 Injection temperature.

4 Temperature the time of ejection.

5 Mould temperature.

6 Coolant temperature and pressure.

7 Surrounding temperature

Heat input required

$$Q_m = mxQ$$

Where, Q_m = Total heat required per hour (cal/h)

m = mass of plastic material injected into the mould per hour (g/h)

Q = total heat content of plastic material (cal/g).

Problem 1

A part made from ABS weighing 100g to be made in a 6 cavity mould at a cycle time of 12s. Specific heat of ABS is 0.49. Calculate the heat energy required if the melting temperature is 250ºc and room temperature is 28ºc.

Shot size = 100gx 6= 600 g

Shots/hour= 3600/12=300shots/hour

Mass moulded/hour = 600x300=18000g/h

Heat energy required per hour for raising the temp of the mass moulded by 1ºc = 18000g/h x 0.49cal/g =8820 cal/h

Total energy input required for the plastic during one hour= 8820 x (250-28) = 1958040cal/h

Quantity of heat to be removed

The heat added to the plastic must be removed from the moulding so that it is stiff enough to be ejected. It is not required to cool the mould to the room temperature for making the part stiff. So all the heat added to the plastic need not be removed by the coolant. The parts continue to shrink after ejection until it is completely cooled down. This will affect the dimension of the part. Hence ejection temperature may be critical for accurate parts.

The same formula given above can be used for calculating the heat to be removed prior to ejection.

Problem

In the above example calculate the heat to be removed per hour by the coolant if the safe ejection temperature is 70ºc.

Heat to be removed from mold = 8820cal/h x (250-70)

= 1587600cal/h

Quantity of coolant required

The heat to be removed from the plastic must be carried away by the coolant flowing through the mould and back to cooling tank. The temperature difference between inlet and outlet is generally 3-5ºc. But occasionally it may be reduced to 1-2ºc or increased upto 10ºc.

The removal of heat largely depends on the temperature difference between the injected plastic and the coolant. Using cold coolant only marginally reduce the cycle time. In most of the cases, the added cost of chiller may not be justified economically.

We have all ready calculated the heat to be removed. If we fix the inlet and outlet temperature of water we can easily calculate the amount of water required since the specific heat of water is one.

Problem

1 How many litters of water are to be circulated per hour in the above example if we maintain the inlet and out let temperature at 3ºc?

Heat removed per hour = 1587600 / 3 cal/h = 529200cal/h

Since the specific heat of water is 1:

Quantity of water required per hour= 529200g/h = 529.2 L/h of water.

2 Calculate the cooling hole diameter and find out whether turbulent flow of water will occur for the above example.

Calculation of cooling hole Diameter

Formula Q = A.V

Where Q = Flow rate

A = Area of cross section

V = Velocity

For this problem given data

$$Q = 529.2\ L/h = \frac{529.2}{3600}\ L/sec$$

V = 1.5 to 2.5 m/s (Taken from standard chart)

$$\frac{529.2}{3600} = A \times 2$$

$$A = \frac{529.2}{3600 \times 2}$$

$$\frac{\pi d^2}{4} = \frac{529.2}{3600 \times 2}$$

$$d^2 = \frac{529.9 \times 4}{3600 \times 2 \times 3.14}$$

$$d^2 = \frac{1058.4}{11304}$$

d^2 = 0.09363 meter

d^2 = 93.6 mm

<u>d = 9.67mm</u>

Calculation of Turbulent flow

$$\text{Reynolds number} = R_e = \frac{\text{Velocity} \times \text{Diameter in meters}}{\text{Kinematic viscosity}}$$

Velocity 1.5 to 2.5m/sec

Kinematic viscosity for 20 C water is 1.0124 x 10^{-6}

$$\text{Reynolds number} = R_e = \frac{2 \times 0.0093}{1.0124 \times 10^{-6}}$$

R_e = 18496.64164

Since, Reynolds number lies between 10,000 and 20,000 it is optimum

Hence, flow of water will be Turbulent

Note: This is only a theoretical value. It must be adjusted against the efficiency of heat transfer.

Efficiency of heat transfer

The quantity of heat that can be transmitted through the mould material can be calculated. The formula used to calculate the quantity of heat transfer (H) per hour according to core or cavity material is given below. Heat flow is directly proportional to the time, area of cross section of the cavity wall in contact with the moulding material and the difference in temperature of the molten plastic material and inversely proportional to the distance from the cavity surface to start of the water line.

$$H = \frac{KAT(t_2 - t_1)}{L}$$

Where

K = Thermal conductivity factor of the mold material. (Table 2)

A = Area of the cavity in contact with the molding material.

T = Time in hours (cycle time)

t2 = temperature of melt.

t1 = temperature of circulating medium

L = Distance from the face of cavity to the start of cooling hole.

Table 2

'K' value for different mould material (KW/h)

Stainless steel	2.93
Tool steel(H13)	3.515
Tool steel (P20)	6.153
Beryllium copper	18.17
Kirk site	18.17
Brass (60-40)	20.51
Aluminum	22.30
Copper (pure)	65.05

An alternate method for estimating the quantity of heat removed from mould per hour

Qm = m {C_p (temperature of material -temperature of mould) L}

Where Cp = specific heat capacity of plastic material (J/kg/K)

L = Latent heat of fusion of plastic material (J/kg)

Typical plastic material melting and mould temperatures (Table 3)

Plastic material	Melting temperature (ºc)	Mould temperature(ºc)
ABS med, high impact	227- 266	32- 82
ABS high heat grade	266 - 282	60 - 82
Acetal	193 - 216	60 - 82
Acrylic	216 - 252	49 - 82
CA, CAB	182 - 227	27- 54
EVA	177- 204	10 - 38
Ionomer	216 - 238	10 - 38
Nylon 66	260 - 293	49 - 82
Nylon 6	243 - 277	38 - 82
Nylon 6/10	238 - 266	38 - 82
Nylon 11	216 - 249	27 - 60
Nylon 12	204 - 232	27 - 60
PC low/med viscosity	282 - 299	71 - 93
PC high viscosity	310 - 338	82 - 121
Poly Easter - PBT	238 - 254	38- 60
Poly Easter - PET (bottle)	271 - 293	16 - 49
Poly Easter - PETG	249 - 271	21 - 49
Poly Easter - PCTG	271 - 293	21 - 49
Poly Easter - PCT(GF)	296 - 310	93 - 121
Poly Easter - PCTA(GF)	293 - 310	149 -160
Polyetherimide	360 - 382	104 - 149

Polyethylene (low density)	171- 227	10 - 38
Polyethylene (med density)	199- 254	10 - 49
Polyethylene (high density)	216 - 282	10 - 66
PPO/polystyrene copolymer	249 - 304	66 - 104
Polyphenylene sulfide	310 - 354	88 - 110
Poly propylene	216 - 271	16 - 66
Poly styrene GP	193 - 238	10 - 49
Ploy styrene impact mod	204 - 249	10 - 49
Poly sulphone	338 - 371	104 - 149
Poly urethane	199 - 227	21 - 49
PVC flexible, rigid	160 - 216	10 - 49
SAN	204 - 260	49 - 82
T/P Elastomer	171 - 227	21 - 49

Alternate method for estimating the quantity of water to be circulated per hour to remove heat

Q_w = K.m_1 (tout-tin) x C_p

Where,

Q_w = Rate of heat extracted per hour (kJ/h)

m_1 = mass of water circulated (kg/h)

C_p = specific heat capacity of water at component pressure (4200/kg/K)

(t out -t in - is usually 3 to 5°C)

K = constant to allow for heat transfer efficiency in steel.

Values of constant, K

Cooling channels bored in cavity plate or male core 0, 64

Cooling channels bored in back plate 0, 50

Cooling channels using copper pipes 0, 10

Since $Q_w = Q_m$

m_1 = m x Q / K (t out-1 in) x C_p

A method for estimating the cooling time

The principal factor that generally controls the cycle time is the cooling period, which is necessary to produce a warp free moulding. Cooling period of cycle Q is given by the formula given below.

$$Q = -\frac{t2}{2\pi\alpha} \text{loge} \left\{\frac{\pi}{4}\frac{Tx - Tm}{Tc\_Tm}\right\}$$

Where

Q = Cooling time in's'

t = Maximum part thickness in 'mm'

α = thermal diffusivity of material in mm^2/s

(Thermal diffusivity is calculated from density, specific heat and thermal conductivity of the material = (K /d c_p)

T_x = Heat distortion temperature of material in °C

Tm = Mould temperature in °C

Tc = Cylinder temperature in °C.

Note: Cooling time using this formula is some what conservative or long for thicker parts. This is because actual moulding of thick walled parts typically results in opening of the mould prior to full wall cooling. A hotter or slightly molten centre may exist on thick walled parts.

Thermal properties of selected resins (Table 4)

Table 4

Material	**Thermal Conductivity (Kw/hr)**	**Specific heat (Cal/g °c)**	**Density (g/cc)**	**Thermal diffusivity (mm^2/sec)**	**Deflection temperature @4.64kg/cm^2 (or)4.55bar**
ABS	0.032 - 0.056	0.490	1.060	0.1193546	95
CA	0.028 - 0.056	0.410	1.260	0.1167739	89
CAB	0.028 - 0.056	0.390	1.200	0.1290320	94

HIPS	0.007 - 0.023	0.501	1.050	0.0380644	85
IONOM	0.045 - 0.045	0.645	0.950	0.0954836	52
LDPE	0.061 - 0.061	0.760	0.920	0.1135481	45
MDPE	0.061 - 0.076	0.765	0.935	0.0703224	68
HDPE	0.084 - 0.095	0.870	0.960	0.1399997	86
PA6,6/6	0.045 - 0.045	0.731	1.140	0.7032244	180
PC	0.034 - 0.034	0.438	1.200	0.0851611	138
PET	0.045 - 0.045	0.502	1.330	0.0890320	67
PET (C)	0.045 - 0.045	0.548	1.360	0.0799998	122
PMMA	0.029 - 0.045	0.454	1.190	0.0890320	102
POM	0.042 - 0.042	0.715	1.420	0.0541934	169
PP	0.020 - 0.025	0.667	0.900	0.0496773	96
PP CO	0.015 - 0.031	0.640	0.900	0.0535482	84
PPO/PS	0.039 - 0.039	0.519	1.070	0.0929030	112
GF PPS	0.053 - 0.082	0.497	1.650	0.1070965	260
PS GP	0.018 - 0.025	0.480	1.060	0.0561289	82
PVC	0.026 - 0.026	0.383	1.290	0.0690321	69
PVC RIG	0.029 - 0.029	0.340	1.400	0.0793546	79
SAN	0.022 - 0.022	0.471	1.080	0.0567740	107

Injection moulding machines

Objectives: At the end of this lesson you shall be able to
- **state what is moulding machine**
- **list the type of moulding machines**
- **explain the terms related to the injection moulding machine**
- **specify the hand injection moulding machine**
- **brief the construction of the screw and its terminology**
- **explain the sequence of moulding operation.**

Indroduction

An injection moulding machine is a machine for converting, processing, and forming of raw plastic material of powder, pellets, or regrind into a part of desired shape and configuration. The process of injection moulding consists of heating plastic material until it melts, then forcing this melted plastic into a mould where it cools and solidifies. The process of converting the plastic raw material in to a melt of uniform temperature is known as plasticizing.

Types of plasticizing units used are:

1 Plunger injection.

2 Two -stage plunger.

3 Plasticizing screw (non-reciprocating) and plunger.

4 Reciprocating screw.

Reciprocating screw machines are widely used.

Types of injection moulding machines used are

1 Hand injection machine

2 Horizontal injection moulding machine.

3 Vertical injection moulding machine

4 Universal injection moulding machine

5 Multi color injection moulding machine.

6 Special purpose injection moulding machine

The mould is held between the clamping platens of the machine. The clamping in most machines is of two types - mechanical (toggle) and hydraulic.

An injection moulding machine can be operated in three different ways.

1 Manual operation: It is operation in which each function and the timing of each function are controlled manually by an operator.

2 Semi-Automatic Operation: It is an operation in which machine performs a complete cycle of programmed moulding functions automatically and then, stops. It then, requires an operator to manually start another complete cycle.

3 Fully - Automatic Operation: It is an operation in which machine performs a complete cycle of programmed moulding function repetitively, and stops only, if there is a machine or mould mal-functions or it is manually interrupted.

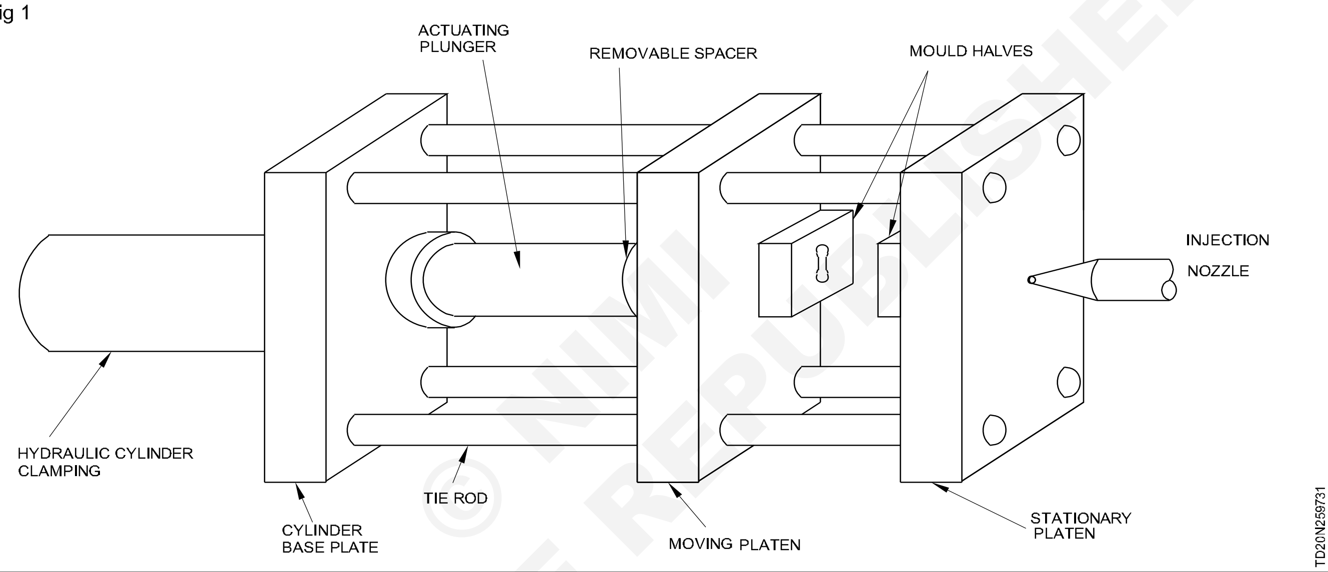
Injection moulding machine nomenclature

Clamping system teminology

Clamping unit (Fig 1)

It is the portion of the injection moulding machine in which the mould halves are clamped, and which provides the motion and force to open and close the mould and to hold the mould closed with force during injection. When the mould is closed in horizontal direction, the clamp is called as a horizontal clamp. When closed in vertical direction, the clamp is called as a vertical clamp. There are wide variety of clamping mechanism available today, but, majority of the machines are either hydraulic or toggle operated. The choice between toggle and hydraulic clamp is not that critical and is often determined by individual preferences.

Fig 1



Moving platen

It is a member of the clamping unit which is moved towards a stationary member. The moving half of the mould is clamped to this moving platen. This member usually includes the ejector knockout holes and mould mounting pattern of threaded holes or "T" slots.

Stationary platen

It is the fixed member of the clamping unit on which the stationary section of the mould is clamped. This member usually includes a mould mounting pattern of threaded holes or "T" slots.

2.1.1.3 Tie rods or tie bars

These are the guides for the movable platen. These also serve as a tension member for the clamp during injection.

Ejector (Knockout)

It is a provision in the clamping unit that actuates a mechanism within the mould to eject the moulded parts from the mould. The ejection actuating force may be applied hydraulically or pneumatically by cylinders attached to the moving platen, or mechanically by the opening stroke of the moving platen.

Full hydraulic clamp

It is a clamping unit actuated by a hydraulic cylinder that is directly connected to moving platen. Direct fluid pressure is used to open and close the mould, and to provide the clamping force to hold the mould closed during injection. Hydraulic clamps are generally recommended for applications that require a clamping tonnage more than 500 tonnes.

The advantages of hydraulic clamps are

1 Easy to adjust the clamping force.

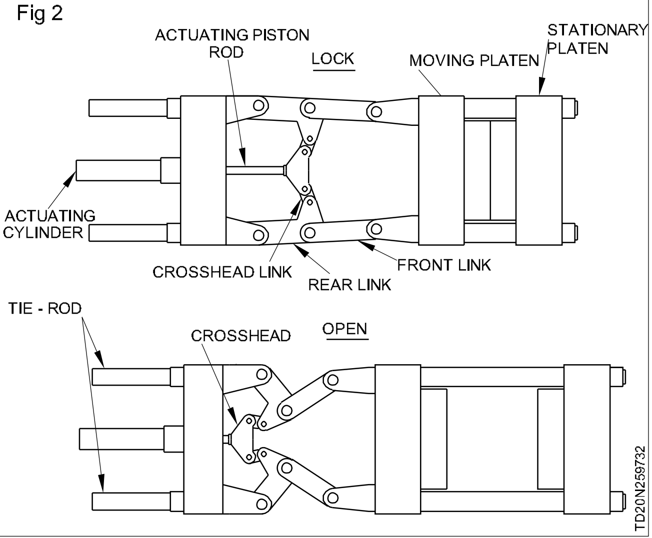
2 Gives direct read out of clamping force.

3 Clamping force remains constant through out the operation.

Toggle clamp (Fig 2)

It is a clamping unit with a toggle mechanism directly connected to the moving platen. A hydraulic cylinder, or some mechanical force device, is connected to the toggle system to exert the opening and closing force and hold the mould close during injection. The clamping force to hold the mould closed during injection is provided by the mechanical advantage of the toggle. Toggle mechanisms are preferred when the cycle time is less than 6 seconds.

Fig 2



The advantages of toggle clamps are:

1 Very compact and less cost for a clamp tonnage.

2 Linkage automatically allows clamp slow down obviating additional controls.

Slow mould open

A provision in the machine design enables slow platen movement for an adjustable distance during the initial opening of the mould.

Clamp close slow down

A provision in the machine design to slow down the moving platen for an adjustable distance before the mould faces come in contact.

Clamp open slow down

A provision in the machine design to slow down the moving platen for an adjustable distance before it reaches its maximum open position. This sequence is often employed to reduce the effect of knockout impact when mechanical knockouts are used. It is sometimes referred to as the ejector or clamp open cushion.

Clamp close stroke interruption

The clamp closing stroke is stopped to allow an auxiliary operation before completion of the closing stroke.

Clamp open stroke interruption

The clamp opening stroke is stopped to allow an auxiliary operation before completion of the opening stroke.

Clamp close preposition

A provision in the machine circuit to allow the clamp to open fully and then close to a pre-determined position. It is, generally, used to allow the ejector knockout mechanism to retract so that inserts can be placed in the mould.

Low pressure mould closing

A provision in the machine intended to lower the clamp closing force during the clamp closing cycle. The lower clamp forces minimize the danger of mould damage caused by parts caught between the mould faces. Provisions are, also, provided for parting mould faces at a timed interval in case of an obstruction.

Injection system terminology

Injection piasticizing unit

It is the portion of an injection moulding machine which converts a plastic material from a solid phase to a homogeneous semi liquid phase by raising its temperature. This unit maintains the material at a present temperature and forces it through the injection unit nozzle into a mould

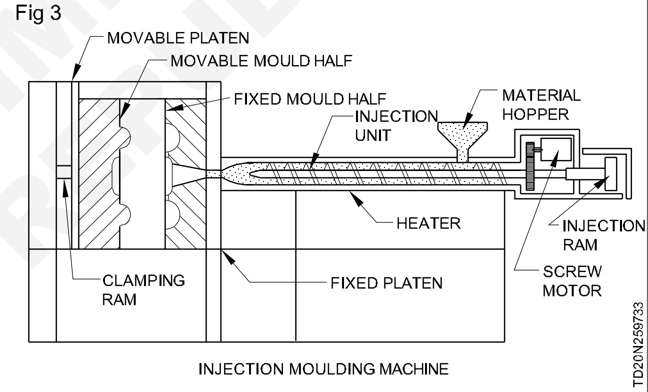
Plunger unit

It is a combination injection and plasticizing device in which a heating chamber is mounted between the plunger and the mould. This chamber heats the plastic material by conduction. The plunger on each stroke pushes unmelted plastic material into the chamber, which in turn forces plastic melt at the front of the chamber out through the nozzle.

Injection moulding machine (Fig 3)

An injection moulding machine is a machine for the converting, processing, and forming of raw plastic material of powder, pellets, or regrind into a part of desired shape and configuration.

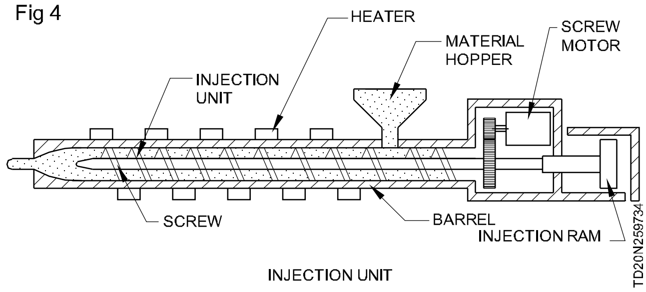
Fig 3



Injection unit: (Fig 4)

An injection unit is part of the machine which forces the material in to the mould.

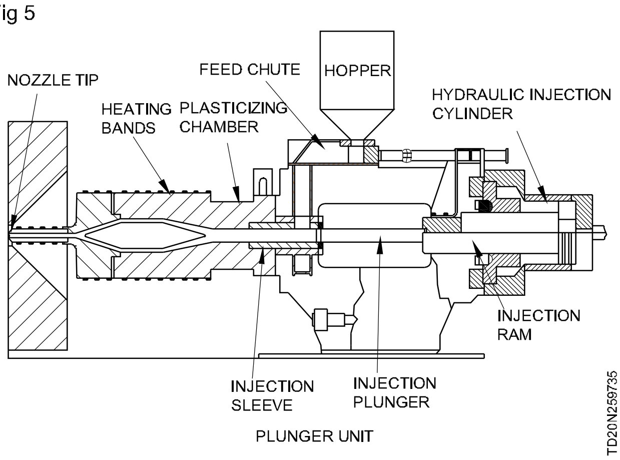
Fig 4



Two stage plunger unit (Fig 5)

It is an injection and plasticizing unit in which the plasticizing is performed in a separate unit. The latter consist of a chamber to heat the plastic material by conduction and a plunger to push unmelted plastic material into a second stage injection unit. This injection unit serves as a combination holding, measuring, and injection chamber. During the injection cycle the shooting plunger forces the plastic melt from the injection chamber out through nozzle.

Fig 5



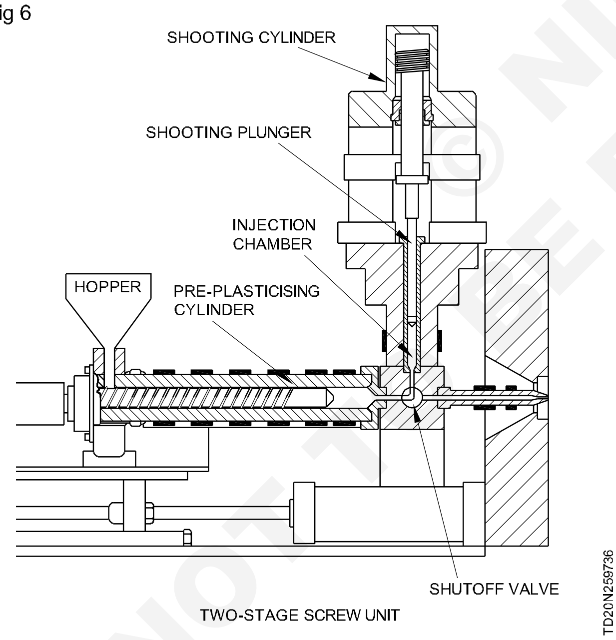
PLUNGER UNIT

Two stage screw unit (Fig 6)

It is an injection and plasticizing unit in which the plasticizing is performed in separate units that consist of a screw extrusion device to plasticize the material and force it into a second stage injection unit.

This injection unit serves as a combination holding, measuring, and injection chamber. During the injection cycle, a plunger forces the plastic melt from the injection chamber out through nozzle.

Fig 6

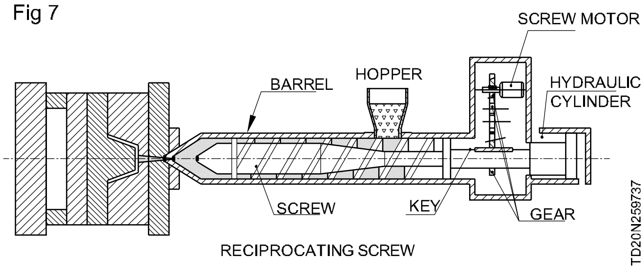


TWO-STAGE SCREW UNIT

Reciprocating screw (Fig 7)

It is a combination injection and plasticizing unit in which an extrusion device with a reciprocating screw is used for plasticizing the material. Injection of material into a mould can take place by direct extrusion into the mould, or by using the reciprocating screw as an injection plunger, or by a combination of the two. When the screw serves as an injection plunger, this unit acts as a holding, measuring, and injection chamber

Fig 7



RECIPROCATING SCREW

Adjustable injection rate

It is the ability to adjust the injection rate in step-less control between the maximum and minimum injection rate.

Prepack or stuffing

It is a method that can be used to increase the volumetric output per shot of the injection plunger unit by prepacking or stuffing (i.e.) by adding material into the heating cylinder by means of multiple strokes of the injection plunger. (Applies only to plunger unit type injection machines)

Plunger prepositon

It is the position of the injection plunger by either limit switches or pressure switches, so that total travel during injection is reduced. The primary purpose of injection preposition is to reduce overall time by eliminating unnecessary plunger travel time during injection. (Applies only to plunger unit type injection machines)

Dual injection pressure

It is the ability to select two separate injection pressure for timed intervals during the injection cycle.

Nozzle retraction stroke

It is the maximum stroke of the hydraulic cylinder, used to separate the injection rate unit from the sprue bushing of the mould for cleaning and/or purging purpose.

Injection system specification

Injection capacity or shot capacity

Injection capacity or shot capacity is the maximum swept volume (or trapped volume in a plunger unit) in cm3 that can be displaced by a single stroke of the injection plunger or screw assuming no leakage and excluding the use of a rotating screw to displace additional volume. It is based on the volume of material that is available for injection in one shot. It is given by the cross section area of the screw or plunger multiplied by the length of injection stroke. The shot capacity is usually given in terms of the weight of the polystyrene material that will occupy the volume prior to injection. For other materials, it must be calculated, based on the specific volume of the molten plastic. Melt temperature and back pressure affect the specific volume of the melt.

Thermoset injection capacity

Injection capacity can be measured in cm3 of swept volume, but, if there is no return valve on the thermo set screws, this cannot be used to convert to true shot-weight because some material flow back over the screw during injection happens. The amount of back flow is dependent on variables in both the machine and moulding material; hence, actual injection capacity cannot be absolutely defined.

Plasticizing capacity

It is the maximum quantity of a specified plastic material that can be raised to a uniform and mouldable temperature in a unit of time. This is, usually expressed in kilograms per hour as available from a plunger unit. In this case, screw unit plasticizing (plasticating) capacity is generally expressed in Kg per hour as calculated from the recovery rate.

Injection pressure

It is defined as the maximum theoretical pressure of the injection plunger or screw against the material expressed in kg/cm2 (assuming no loss of pressure due to frictional drag of the plunger or screw) at maximum force acting on the injection piston.

Maximum injection rate

The maximum calculated rate of displacement of the injection plunger or screw, expressed in cm3per second computed at maximum injection pressure as specified.

Minimum injection rate

The minimum calculated rate of screw, expressed in cubic centimeter per second computed at maximum injection pressure as specified.

Clamping system specification

Clamping force (Tons)

The clamp tonnage is the locking force in tons which is applied to the mould to keep it closed during the injection and cooling of the mould. It is the maximum force applied to the mould when it is closed for injection.

Clamp opening force (Tons)

It is the maximum force that a machine will exert to initiate the opening of the mould.

Clamp stroke:(Max) Millimeters

It is the maximum distance between the opening and closing mechanism can traverse a platen this may usually be adjusted to shorter travel to meet mould or moulding requirements.

Open daylight (Max): Millimeters

It is the maximum distance that can be obtained between the stationary platen and the moving platen when the actuating mechanism is fully retracted with or without ejector box and/or spacers.

Closed daylight or minimum mould thickness

It is the maximum distance between the stationary platen and the moving platen when the actuating mechanism is fully extended with or without ejector box and/or spacers. Minimum mould thickness will vary depending upon the size and kind of ejector boxes and/or spacers used.

Maximum closed daylight: Millimeters

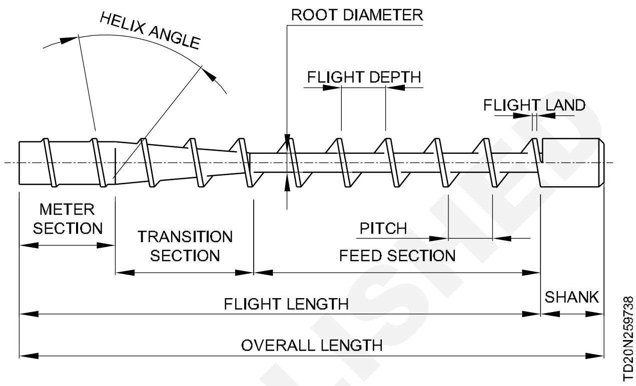
It is the maximum distance between the stationary platen and the moving platen when the actuating mechanism is fully extended without ejector box and/or spacers.

Minimum closed daylight: Millimeters

It is the minimum distance between the stationary platen and the moving platen when the actuating mechanism is fully extended with standard ejector box and/or spacers.

Screw terminology (Fig 8)

Fig 8



Plasticising screw

It is a helically flighted shaft when rotated within the barrel mechanically works and advances the material being processed.

Screw flight

It is the helical metal thread of the screw.

Screw root

It is the continuous central shaft, usually cylindrical or conical in shape.

Flight land

It is the surface at the radial extremity of the flight constituting the periphery or outside diameter of the screw.

Screw shank

It is the rear protruding portion of the screw to which the driving force is applied.

Feed section of screw

It is the portion of the screw that picks-up the material at the feed opening (throat) plus an additional portion downstream. Many screws have an initial constant lead and depth section.

Transition section of screw

It is the portion of a screw between the feed section and metering section in which the flight depth decrease in the direction of discharge.

Metering section of screw

It is a relatively shallow portion of the screw at the discharge end with a constant depth and lead.

Screw diameter

It is the diameter developed by the rotating flight land about the screw axis.

Helix angle

It is the angle of the flight at its periphery relative to a plane perpendicular to the screw axis .The location of measurement should be specified.

Light lead

It is the distance in an axial direction from the center of a flight at its outside diameter to the center of the same flight one turn away .The location of measurement should be specified.

Light pitch

It is the distance in an axial direction from the center of a flight at its periphery to the center of the next flight .In a single flighted screw, "pitch" and "lead" will be the same but they will be different in a multiple flighted screw . The location of measurement should be specified.

Flight depth

It is the distance in a radial direction from the periphery of a flight to the root. The location of measurement should be specified.

Constant lead screw, uniform pitch screw

A screw with a flight of constant helix angle.

Constant taper screw

A screw of constant lead and uniform increasing root diameter over the full flighted length.

Decreasing lead screw

A screw in which the lead decreases over the full flighted length (usually of constant depth).

Screw channel

It is with the screw in the barrel, the space bounded by the surface of flights, the root of the screw, and the bore of the barrel .This is the space through which the stock is conveyed and pumped.

Screw channel depth

It is the distance in a radial direction from the bore of the barrel to the root .The location of measurement should be specified.

Compression ratio

It is the factor obtained by dividing the developed volume of the screw channel at the feed opening by the developed volume of the last full flight prior to discharge. (Typical values range from 2 to 4, also expressed as a ratio 2:1 or 4:1.)

Screw speed

Number of revolution of the screw per minute is called Screw speed.

Plasticizing unit specification

L/D ratio

It is the common denominator used for the comparison of all

Barrel length to diameter ration- L/D

The distance from the forward edge of the feed opening to the forward end of the barrel bore divided by the bore diameter and expressed as a ratio where in the diameter is reduced to one, such as 20/1.

Screw length to diameter ratio-L/D

The distance from the forward edge of the feed opening to the forward end of the screw flight divided by the diameter of the screw expressed as a ratio. It is not based on the full flighted length of the screw.

Construction of screw (Fig 9)

The main elements of a typical reciprocating screw unit are a barrel, a motor used to rotate the screw and an injection ram and cylinder used to give axial movement of the screw with in the barrel.

For moulding thermoplastic materials, the barrel is usually fitted with electrical heater bands on the out side diameter and thermocouples are used to control the temperature of the melt.

Electric or hydraulic motors are generally used for screw rotation either directly or through a transmission system. Hydraulic motors have the advantage of supplying constant torque over wide range of speed where as electrical motors are more energy efficient.

By supplying pressurized hydraulic fluid to the cylinder, necessary pressure can be developed on the screw in order to inject the material into the cavity. The cross sectional area of the ram at the receiving area of the pressure is much larger than the cross section area of the barrel. So, a greater pressure is transferred on the melt than what is there in the hydraulic cylinder.

The material from the hopper falls at the rear end of the barrel by gravitational force or transferred by some other feeding device.

For less viscous and thermally stable materials like PP, a non return valve is attached to the front of the screw to avoid the back flow of the material. How-ever, for high viscous and thermally degradable material like PVC a smear head is often substituted for non-return valve. But, with this arrangement, some back flow may occur which may be minimized by reduction of flow area.

Sequence of operations

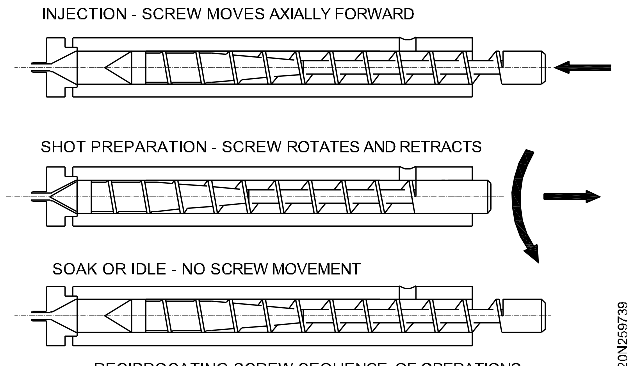
The Sequence of operation for a reciprocating screw is

1 **Injection:** Screw moves axially forward

2 **Short preparation:** Screw rotates anti-clockwise and retracts

3 **Soak or idle time:** There is no screw movement

Fig 9



RECIPROCATING SCREW SEQUENCE OF OPERATIONS

At the starting of the moulding cycle, reciprocating screw occupies a retracted position in the barrel and a charge of molten plastic occupies the region of the barrel bore between the front of the screw and nozzle. When the mould halves have been closed and clamp pressure applied, hydraulic fluid is supplied to the injection cylinder causing the injection ram and the screw to move forward, thereby, injecting material through the nozzle and into the cavity. This pressure is known as injection pressure. When the cavity has filled, a certain percentage of injection pressure is maintained in the injection cylinder until the molten material in the gates solidifies. This pressure is known as "holding pressure". During this period, additional material is, also, pushed into the cavity known as packing the cavity in order to compensate for the shrinkage to certain extent.

A new charge of polymer melt can be prepared for injection into the mould for the next shot. Screw rotation begins in the anticlockwise direction, and material is carried forward along the screw. During its passage along the screw, the material is melted and mixed, and it is discharged from the forward end of the screw. Pressure generated in the discharged melt is transmitted by screw to the injection ram, which displaces hydraulic fluid from the injection cylinder and allows the screw to retract in the barrel.

By controlling the free flow of fluid from the injection cylinder, the pressure acting on the molten material can be changed. This procedure is termed application of "back pressure". When the required volume of the melt has been discharged the screw rotation stops. The process of short preparation is termed "screw back".

Once the new charge of plastic melt for the next shot has been prepared and when the solidified part in the mould cavity has been ejected and the mould closed, the next moulding cycle can begin. In general, there can be a period of time between the end of screw rotation and the start of injection. Any such delay is termed as "soak" or "idle time".

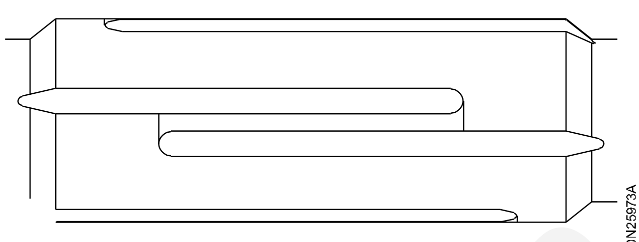
Types of screw designs (Fig 10)

In order to overcome many of the performance limitation of the conventional screws a number of specialized designs are available with emphasis on plasticizing process.

Local mixing screws

Local mixing sections are typically short devices designed to generate high rates of melting or mixing at the delivery end of the screw. The choice of the device is normally depends upon whether melting or mixing should be given importance.

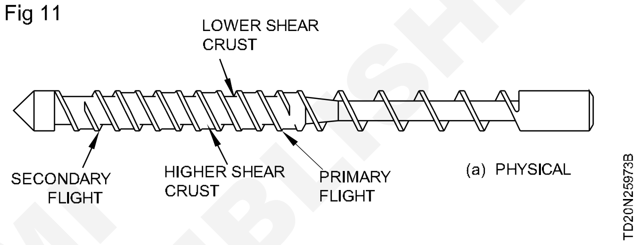
Fig 10



UNION CARBIDE LOCAL MIXING SECTION

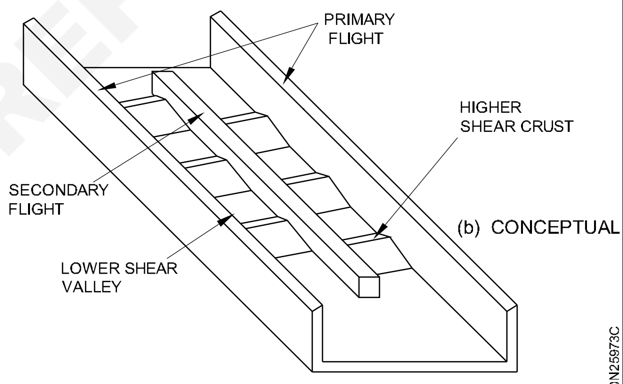
Extended mixing screws (Fig 11)

Fig 11



HPM double wave extended mixing screw (Fig 12)

Fig 12



HPM DOUBLE WAVE EXTENDED MIXING SCREW

Extended mixing screws have got more length for the mixing section. One method is using double wave flute. In this method two long melt channels are made in parallel with each other with periodically varying depths. The shallower depth provides high shear rate with periodic changes in pressure. By arranging channel flutes out of phase with each other material from on flute can be periodically changed to other flute.

In most moulding applications extended mixing screws are used to accelerate the last stage melting process and also to increase mixing.

Barrier screws (Fig 13 &14)

Fig 13

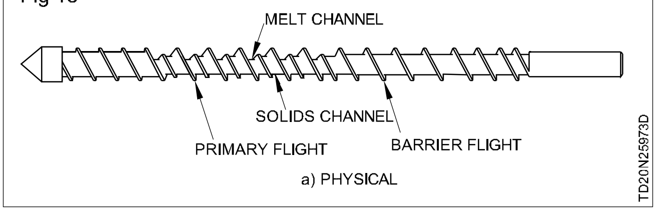
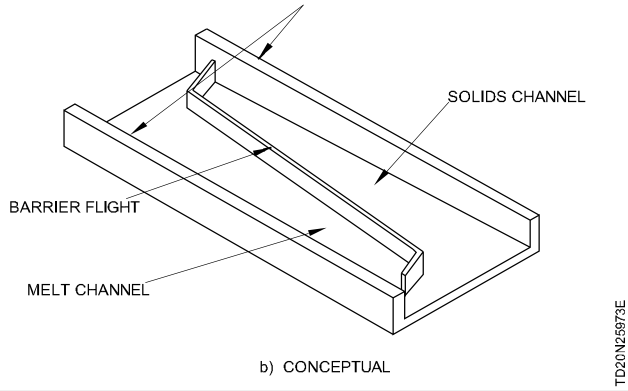


Fig 14

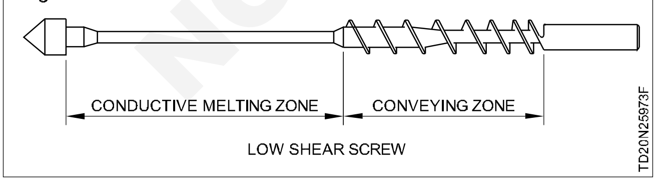


Barrier screws are specially designed for controlling the melting process. This control is achieved through the use of twin channels in the central melting section separated by an under cut barrier flight. One channel is referred to as the solid channel and the other as the melt channel. Solid material is trapped in the solid channel until melting occurs. Molten material is conveyed from the top of the bed and is collected in to the melt channel. The gap between the barrier flight and the barrel is critical as it should not allow any solid particle to go out of the trap. Through the length of melting section the volume of the solid channel progressively reduce and the volume of the melt channel progressively increases. When all the materials have been melted the solid channel is ended and the melt channel becomes the metering section.

Barrier screws are extensively used in extrusion process in order to improve the stability of the melting process. The use of barrier screw in injection moulding is very rare.

Low-shear-screw (Fig 15)

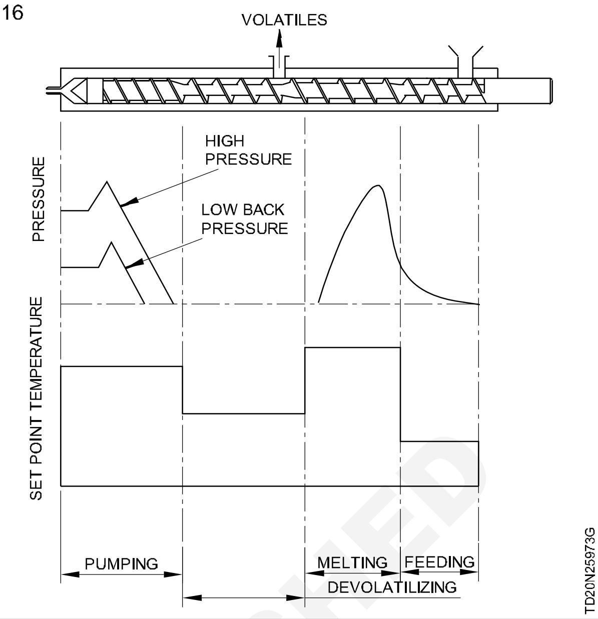
Fig 15



It is a specialized screw design with a flighted cross section, only, in the material conveying zone. There is no flight in the melting zone. Therefore, the melting is mainly due to the heat from the heater bands. These types of screws are required when an additional mixing may damage the property and in homogeneous structure of the material. Mainly, fiber reinforced materials use these type of screw.

Vented barrel injection moulding (Fig 16)

Fig 16



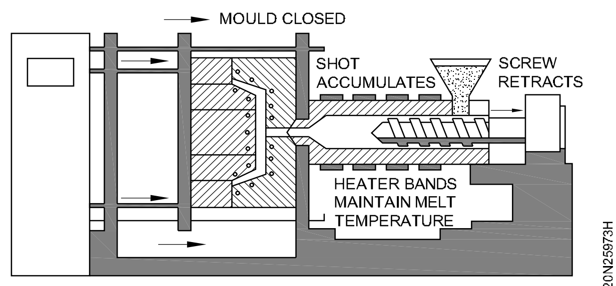
Generally, materials that are moisture sensitive will have to be dried before moulding. Vented barrel injection moulding machines are capable of removing the moisture during the processing of the material. Vaporized water and other volatiles can be removed by using a vented barrel moulding machine. Another method used to remove the moisture is by using pre dryer. But, in most cases, the more practical approach is the use of vented barrel.

This type of drying requires a two stage screw and a barrel incorporating a vent port as shown in the above figure. Molten material leaving the first stage of the screw enters a zone with a large cross section and with a greater flight depth. So, the area is not fully filled with the material and hence, the pressure also, drops equal to atmospheric pressure. The volatiles are released in to the air. At the end of the second stage, the material is again compressed to generate pressure necessary for it to flow through the non-return valve.

The sequence includes the following

- Plasticizing the resin (Fig 17)

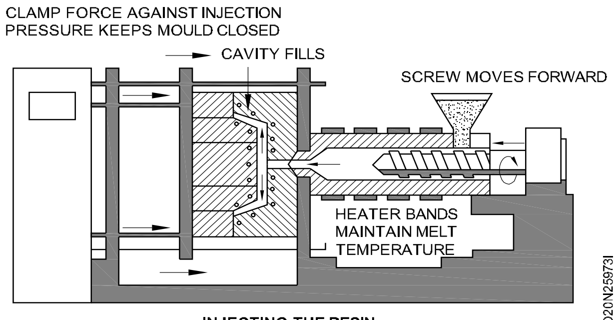
Fig 17



PLASTICIZING THE RESIN

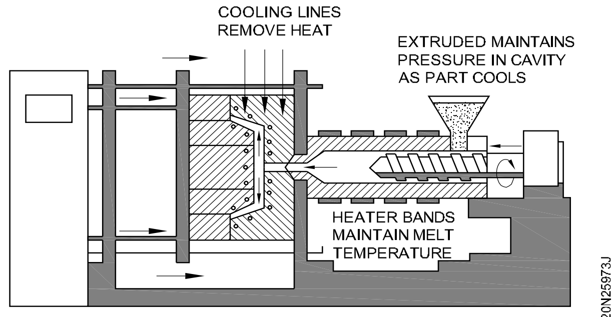
- Injecting the resin (Fig 18)
- Cooling the part (Fig 19)
- Ejecting the part (Fig 20)

Fig 18



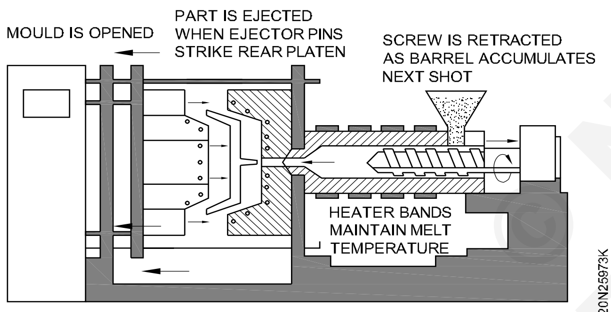
INJECTING THE RESIN

Fig 19



COOLING THE PART

Fig 20



EJECTING THE PART

Sequences in the moulding operation

1 Closing the gates triggers the first limit switch which brings the clamp - ram into forward position to close the mould fast.

2 During the penultimate stage of closing the second limit switch is activated which slows down the closing.

3 The third limit switch is activated during the final clamping to generate the necessary clamping pressure.

4 When sufficient pressure is developed behind the clamp a pressure switch is activated. This switch opens the nozzle valve and also activate injection high pressure timer in order to move the plunger forward.

5 When the injection high pressure times out an injection overall timer is activated. This timer gives the necessary holding pressure.

6 When the injection overall timer times out the melt decompression timer starts.

7 When the decompression timer times out the nozzle valve closes and the screw starts turning in the anticlockwise direction for conveying the material for the next shot. The clamp high pressure also comes down to low clamping.

8 When the screw rotates in the anticlockwise direction the material moves forward and reaches at the end of the screw and pushes the screw back. The backward movement of the screw is stopped by activating a limit switch.

9 The overall timer times out and the slow opening of the mould (opening of the clamp) begins.

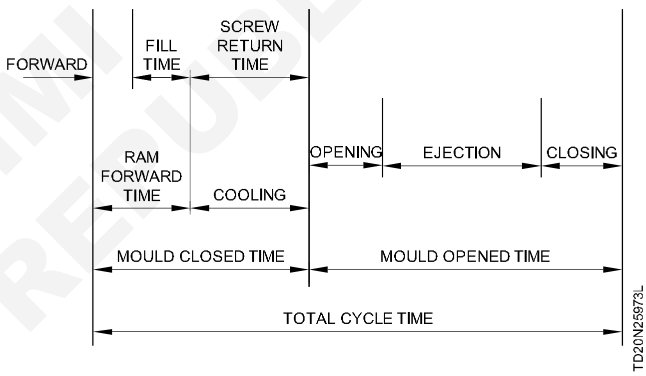
10 The slow opening of the clamp activates another limit switch which causes the clamp to open fast. Then another limit switch is activated to slow down the opening. The slow opening continues till it is stopped by the final limit which is set for the maximum opening.

11 A clamp open timer is also provided which either sets a time for the removal of the part or for automatic continual of the operation.

A mould cycle for a typical screw injection machine is Fig 21. Mould cycle is made up of a number of elements, any one of which can be shortened or accelerated, thus condensing the entire cycle.

It shows the machine condition needed for production of satisfactory parts in minimum time

Fig 21



Preparation of Mould Diagram

The mould diagram shows the machine conditions needed for the production of satisfactory parts in minimum time. Maximum and minimum presses for various cylinder temperatures are determined. The mould temperature and the other elements of the mould cycle are kept constant. Starting at allow temperature and low pressure a series of short shots will be obtained. By increasing the pressure, several satisfactory shots will be produced. At some point as the pressure continued to be increased for each shot, the mould will begin to flash or stick. This then becomes one point on the flash or stick line.

A second series of points are generated at a higher temperature. This procedure is continued with successive temperature increase until a temperature is reached at which polymer deterioration starts. By connecting all the last short shot points, a curve is drawn which indicates the minimum pressure temperature combination at which satisfactory parts can be obtained. The curve connecting all the flash stick lines indicates the maximum pressure temperature combination which can be used to produce satisfactory parts.

The area between the two curves is the mould area. Theoretically combination of pressure or temperature within this area should produce satisfactory components. But in actual conditions, some of the filled parts may show streaking, bubbles or other defects. If a record is kept of the conditions of moulding each part, it is found that some defects are consistently present when parts moulded in the particular area of the diagram. Other areas will produce consistently good parts. The optimum area will be the one characterized by the highest temperature and pressure necessary to produce parts which are completely satisfactory. The use of highest temperatures and pressures will result in a minimum fill time. The object of determining the minimum cycle time is to produce parts with maximum efficiency. The mould diagram will establish the optimum pressure temperature combinations.

To determine the minimum cycle time, the process is started at a given mould temperature with the highest pressure and cylinder temperature which will give satisfactory parts. Series of shots (at a constant pressure and temperature) with decreasing ram forward line is tried. Below a certain point, no satisfactory parts can be made. During this test, the cooling time should remain constant. After establishing the minimum ram forward time, the same procedure is repeated by decreasing the cooling rate. The ram forward time is kept at the minimum. This will establish a minimum moulding cycle for any given cylinder and mould temperature. The procedure should be repeated with either higher or lower mould temperature until an optimum is reached.

The establishment of an optimum fills time and a cooling time effects the total time the plastic remains in the cylinder. The net effect should be a lowering of the time in the cylinder so that possibly a higher cylinder temperature may be feasible. Another series of test shots should be made at various increased cylinder temperature establishing a minimum cycle for each cylinder temperature. A graph of the minimum cycle time verses the cylinder temperature is shown in figure. The minimum cycle time diagram will be very useful when establishing conditions for a new mould or new resin.

The mould cycle time can be again reduced by the following procedure. The fill time can be reduced to a minimum by correct choice of gate size and location. The second element, ram forward line depends on the time required for the gate to solidify. This time depends on the parts thickness, the gate size and the runner size. The mould should have the smallest gate possible. The screw retract time should be slightly less than the cooling time. This can be achieved by the increasing the screw speed or to reduce the back pressure. A machine with larger shot capacity can also be tried. The mould open time can be reduced by keeping the minimum distance for the mould to open just enough so that the parts can be ejected or removed from the core.

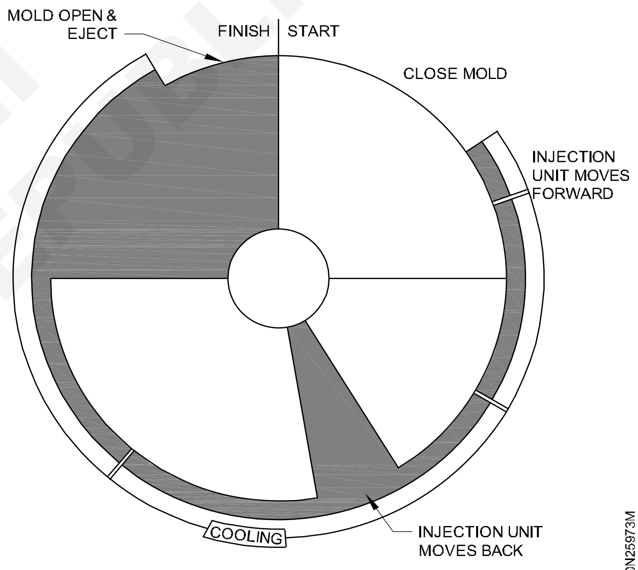
The two disadvantages of reducing the mould cycle time is,

1 The total shrinkage of the part will be more if a longer cycle is used.

2 The component will have higher moulding stresses.

3 Injection moulding cycle is shown in Fig 22.

Fig 22



INJECTION MOLDING CYCLE

Selection number of cavities

Objectives: At the end of this lesson you shall be able to
- **list the factors to be considered for the selection of number of cavities**
- **state the advantage and disadvantage of cavities moulds**
- **calculate the number of cavities.**

Introduction

Once the part has been made suitable for moulding, the next step is deciding on the number of cavities. The various factors to be considered are:

1 Production volume and delivery time.

2 Cost per part

3 Type of gate

4 Shape and size of the part.

5 Machine capacity.

6 Dimensional tolerance.

A proper selection can be made only after comparing the advantages and disadvantages of single and multi cavity moulds.

Advantages and disadvantages of single cavity mould

Advantages

1 It is easy to manufacture and cost is also less.

2 Parts are always identical.

3 Process controlling is easy.

4 It gives better control in design.

Disadvantages

1 It is not cost effective in high volume production.

2 Under utilization of machine in the case of small components.

3 Long delivery time.

4 Production stops when one cavity is damaged.

Advantages and disadvantages of multi-cavity mould

Advantages

1 High volume production.

2 Faster delivery time.

3 Full utilization of machine.

4 Production can continue even if one cavity is spoiled.

Disadvantages

1 Very expensive.

2 Difficult to manufacture.

3 Non-uniform cooling.

4 Non-identical parts.

5 Less flexibility in design.

Calculation of number of cavities

Usually the number of cavities is decided by the customer. But, from the moulder's approach, the number of cavities can be found out as given below.

A Based on Shot Weight

Generally shot weight (S) is taken as 85% of the shot capacity of the machine. If W is the weight of the moulding, then, the number of cavities is:

$N_c = S / W$

Problem

Calculate maximum number of cavities possible on a SP30 machine for a part having 10g weight. Shot capacity of the machine is 36g.

Effective shot weight = 36x0.85 = 30.6

Maximum number of cavities = 30.6/10 = 3.06

Since no fraction is possible the maximum number of cavities can be three.

B Based on Plasticizing Capacity

If P is the plasticizing capacity of the machine, Sm, is the number of shots per minute and

W is the weight of the component, then, the number of cavities is:

$N_c = P / S_{m} \times W$

Generally 85%of the plasticizing capacity is used for calculation.

Problem 1

Calculate the number of cavities possible for the above part (10g) based on plasticizing capacity. Plasticizing capacity for SP30 is 17kg/h. and cycle time achievable is 6 seconds.

Plasticizing capacity = 17kg/h = $\frac{17 \times 1000}{60}$ g/min

= 283.33g/min

Effective plasticizing capacity= 283.33 x0.85 = 240.83

Number of shots/min = 60/6 = 10

Material required / minimum for a single cavity mould = 10x10 = 100g/min

Therefore, the number of cavities possible = 240/100 = 2.4

Since no fraction is possible number of cavities possible is, only, two.

Problem 2

Calculate the number cavities possible for the above problem if the cycle time is changed to 15seconds.

Number of shots/min = 60/15 = 4

Material required / minimum for a single cavity mould = 4x10 = 40g/min

Therefore, the number of cavities possible = 240/40 = 6 cavities.

C Based on Clamping Force

The clamping force required to keep the mould closed during injection must exceed the clamp opening force. Generally cavity pressure is taken as 25-45% of the injection pressure. The clamping pressure required is given by the product of the cavity pressure and the total projected area of the component with feed system.

Clamping Force = Total Projected Area x Cavity Pressure

Problem

Calculate the number of cavities possible on a SP30 machine if the component diameter is 50mm. Injection pressure required is 1200kg/cm^2.

Projected area = 3.14 x 2.5^2 = 19.634cm^2

Cavity pressure = 1200 x 0.45 = 540kg/ cm^2

Clamping force required for single cavity = 540 x 19.634 = 10.602t

Considering 30% safety factor the force required becomes 10.602t x 0.3 =3.1806t

Then, add 10.602 + 3.1806 = (13.7826 or) 13.783.

Clamping capacity of the machine: 30t

Therefore, the number of cavities = 30 / 13.783 = 2.17

Since, no fraction is possible, the number of cavities is only two.

By carefully controlling the cavity pressure, it is possible to use a machine with a lower clamping force. In other words with the same clamping force more parts can be produced at one time, using a multi cavity mould.

The calculation by normal method is not accurate because the cavity pressure is always assumed. Cavity pressure can be accurately measured by placing pressure transducers or estimated using flow modeling techniques.

The actual pressure depends on the following factors:

1 Resistance in the runners and gates.

2 Temperature of plastics.

3 Viscosity of plastics.

4 Distance from the cavity to the gate.

5 The flow rate of the resin.

Advantages of measuring mould cavity pressure using pressure transducers are:

1 **Faster mould optimization of initial machine parameters:** Recording the mould cavity pressure shortens startup time as the effect of a machine adjustment can be known immediately.

2 **Faster start up time with predetermined machine parameters:** Moulding conditions are now defined by the established temperature and pressure profiles. Since trial and error method is not required the start up time is reduced.

3 **Optimized setting result in shorter production cycles:** The cavity pressure profile gives a good indication of happenings inside the cavity. A better revealing of the injection pressure allows more accurate adjustment of the machine, resulting in increased productivity through shorter cycle time and reduced energy consumption.

4 **Less tool modification:** The decision to rework a tool can be made on the basis of the dependable measurement and not a guess work. Mould modification in some instances can be avoided by using pressure controlled injection.

5 **Lower clamping force:** By carefully controlling the mould cavity pressure, it is possible to use a machine with a lower claming tonnage.

6 **Fewer mould release problems:** Excessive cavity pressure generally cause ejection problems. By carefully controlling the injection pressure it can be minimized.

7 **Improved dimensional control:** The dimensional correctness of the part directly depends on the cavity pressure. Controlling the pressure in cavity results in better dimensional repeatability.

8 **Uniform part weight:** The cavity pressure has a direct influence on the weight of the moulded part. With a tighter cavity pressure control very close tolerance on part weight can be achieved. Some cases, it results in considerable material savings.

9 **Mould protection:** Controlled cavity pressure results in a lower mould stress, less distortion, reduced wear and longer tool life.

10 **Consistent part quality:** Internal stress caused by the difference in set point temperature and pressure can degrade the part quality. This can be eliminated by early detection and taking corrective measures.

11 **Less rework:** Excessive pressure can result in flash and burr on part. This also can be avoided by timely intervention.

12 **More economical production with fewer and less skilled personnel:** Maximum production with few unskilled operators can be achieved by eliminating all possible problems by constantly monitoring the cavity pressure.

Other factors affecting the number of cavities are:

1 Location of mould cavities.

2 Lay out of the mould as decided by the product design.

3 Quantity requirements in a family mould.

4 Component assembly or packing requirement.

5 Indexing requirement for multi-colour and multi-material components.

Strength of the cavities

The maximum deflection commonly allowed is 0.125 to 0.25 mm depending up on the size of the tool. Of this, 0.10 to 0.20 mm may be because of the clearance between the insert and pocket and elongation of the bolster. For calculation purpose a maximum deflection of 0.025 - 0.05 mm is generally taken. The approximate thickness required for the side walls may be calculated from the following formula:

$$T = \sqrt[3]{\frac{C.P.D.^4}{E.Y}}$$

Where,

T = Thickness of cavity wall (cm)

C = constant.

P = maximum cavity pressure (kg/cm^2)

D = depth of cavity wall.

E = Modulus of elasticity of steel (2.1 x 10^6 kg/cm^2)

Y = the maximum allowable deflection of cavity wall.

Value of constant (C)

The value of "C" depends on length (L) of the cavity wall to the depth (D) of the pocket. (Table 1)

Table 1 Value of "C"

L/D	C
1.0	0.044
1.5	0.084
2.0	0.111
3.0	0.134
4.0	0.140
5.0	0.142

Problem

Calculate the minimum wall thickness required if the cavity depth is 100mm and cavity length is 300mm? Cavity pressure 500kg/cm2. Maximum deflection allowed is0.0.03mm.

We have, $T = \sqrt[3]{\frac{C.P.D.^4}{E.Y}}$

L / D = 300/100 = 3

Therefore, C= 0.134

$$T = \sqrt[3]{\frac{0.134x500x10^4}{2.1x10^6.x0.003}}$$

= 47.30mm

Table 2

Recommended materials for various mould parts

Mould elements	International standards					Hardness (HRC)
	ASSAB/TM	AISI	BS	DIN	JIS	
Cores, Cavities, Inserts, Pins, etc (Short run	Impax supreme	P20	EN325	40Cr Mn Mo7	SNCM	32-34
(Medium run)	Stavax ESR,8407	420PQ,H13	EN24	X42 Cr 13		40-45
(Long run)	Stavax ESR ,8407	420PQ,H13	EN24		SKD61	48-52
(Best Polish)	Stavax ESR Polmax DievacTM*	420SS		X42 Cr 13	SUS420	49-52
(Long Cores)	ASP 23,8407	T15,M2, H13		S-6-5-2	SKH 51	61-63
(High corrosion resistance)	Stavax Ramax	420ss		X42Cr13	SUS420	48-52
(Least cycle time-best heat transfer)	Alumec,Mouldmax AMPCOTM*	Be Cu B25 Alum 6061-T6				36-41
Wear Plates	Sevrker21	A2,D2,M4	EN31,BD2	X100Cr MoV51	SKD11	52-55
Wedge Locks	Sevrker21	A2,D2,M4	EN31	X100Cr MoV51		52-55
Guide Rail	Sevrker21,DF2	A2,D2,O1	EN31,BA2, BA3	106WCr6		52-55
Lock Ring	Sevrker21	A2,H13	EN31,BH13	X40Cr MoV51		56-59
Slides		A2,H13	EN31,BH13	X40Cr MoV51		49-51
Stripper Rings		A2,S7	EN31	60MnSiCr4		56-58
Sleeves		D2	EN47	50CrV4		58-60
Mould Plates /Blocks	Holdax, Ramax, Thyroplast2316TM*	4140, P20, 420F,1040	EN8D, 708M40	Ck45	SCM440	26- 34 40-42
Ejector Pin		H13/std				68-72 (Nitrided)
Leader Pin		P5,P6/std	EN32B&C			59-61 (case-Hard)
Guide-Bush		P5,P6/std	EN32B&C			59-61 (case-Hard)
Locator Ring		1015,1020	EN1A	C15,C20		
Cams	ASP 23	M4	EN31		SKH51	62-64

Under cut and splits

Objectives: At the end of this lesson you shall be able to
- **state what is under cut**
- **brief two types of undercut**
- **list the components of under cut**
- **explain the method of operations**
- **describe the locking method of splits**
- **brief the safety arrangement of split.**

Introduction

A moulding which has a recess or projection which prevents the normal in line ejection of the component is termed as an undercut moulding. An undercut moulding requires the removal of that part of the impression which forms the undercut prior to ejection.

External undercut components

Any recess or projection on the outside surface of the component which prevents its removal from the cavity is termed an external undercut.

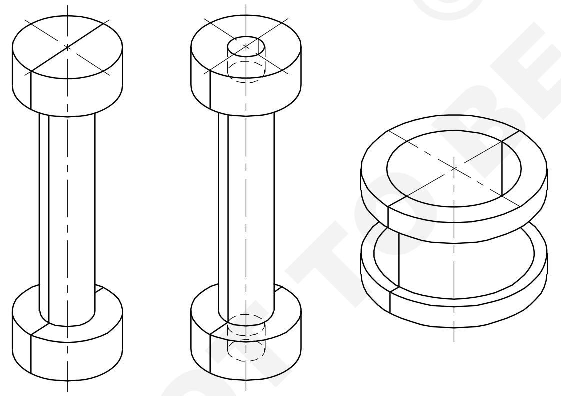
There are two types of undercut:

1 Local undercut (Fig 1)

2 Continuous undercut (Fig 2)

The recess or projection occurs in one place, only, is known as local undercut. (Fig 1)

Fig 1

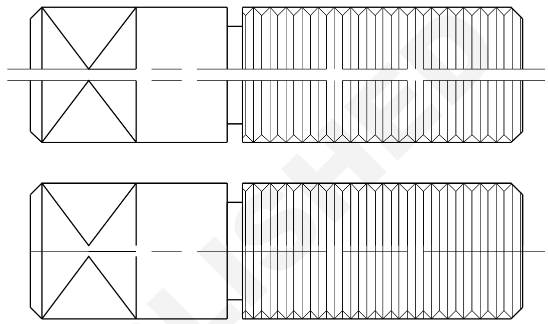


LOCAL UNDERCUTS

The recess or projection around the outer surface of the component which is continuous is known as continuous undercut. (Fig 2)

In either case, it is necessary to split the cavity insert into two parts and open these at right angles to the line of draw, to relieve the undercut before the mouldings are removed. Since, the cavity is in two pieces, a parting line will be seen on the finished product:

Fig 2



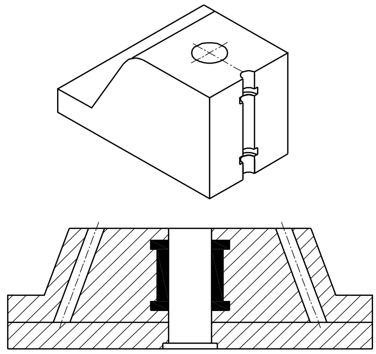
CONTINUOUS UNDERCUTS

The parting line can be positioned on any centre line for a symmetrical component. For unsymmetrical components, there is, only, one possible position. For example, the parting line for pen cap can only occur on the centre of the position. An incorrect parting line will restrict the free opening movement of the split, which will result in scored, cracked components. The joint line should be as invisible as possible in the moulding.

Splits (Fig 3)

Cavities made of different moving blocks are known as splits. They are used to mould a component with an external undercut.

Fig 3



SPLIT CAVITIES

The cavity form is machined into different cavity blocks. The cavity blocks are moved away as the mould is being opened, so that, the undercut is freed from cavity surface. When the mould is opened, the component is on the core which allows normal ejection. There are two basic designs where the splits are retained on the mould plate and actuated automatically - sliding splits and angled - lift splits. In both, there are moving parts and it is necessary to

1 Guide the splits in desired direction

2 Actuate the splits and

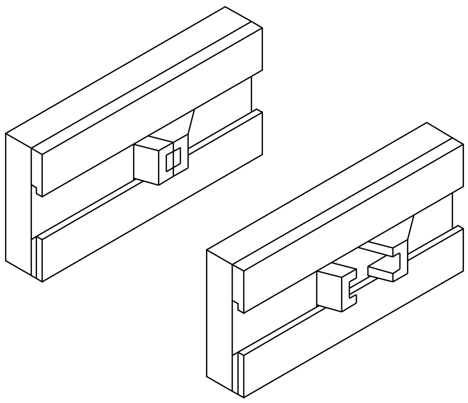
3 Securely lock the splits in position, before, the material is injected into the mould.

Actuation of splits

Sliding splits (Fig 4)

The splits are mounted in guides on a flat mould plate and are actuated by mechanical, hydraulic or pneumatic means. The splits are positively locked in their closed position by heels. Sliding splits can be mounted on either the moving or the fixed mould plate.

Fig 4



FINGER CAM ACTUATED

TD20N259814

Guiding and retention of splits

There are three main factors

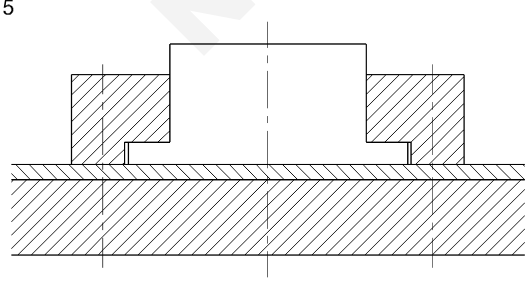
1 Side movement must ensure that the split halves always come together in the same place.

2 All the parts of the guiding system must be of adequate strength to support the weight of the splits and to withstand the force applied to the splits by the operating mechanism.

3 The two halves must have a smooth, unrestricted movement.

The guiding function is normally accomplished by providing a T-shaped slot on the mould plate or using two shouldered Gibs . (Fig 5). The starting position (retention) is by means of spring, ball catch or other standard items. (Fig 6)

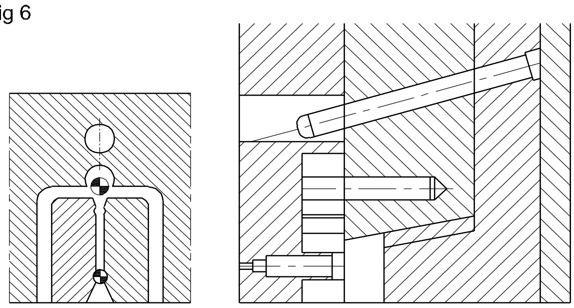
Fig 5



SLIDES AND GIBS

TD20N259815

Fig 6



SLIDE RETENTION METHODS

TD20N259816

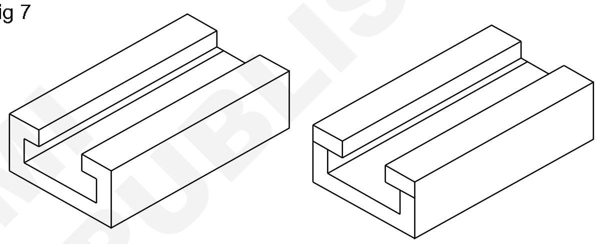
Mould plate designs for slidig splits

Slide movement is accomplished by the following designs.

"T" slot design (Fig 7)

The figure "a" shows the solid design. This single part construction is ideal for small moulds where the mould plate is hardened. The figure "b" shows a mould plate with slot milled in it. Two flat steel strips are clamped on either side to get the required "T" shape

Fig 7



T-SLOT (INTEGRAL)

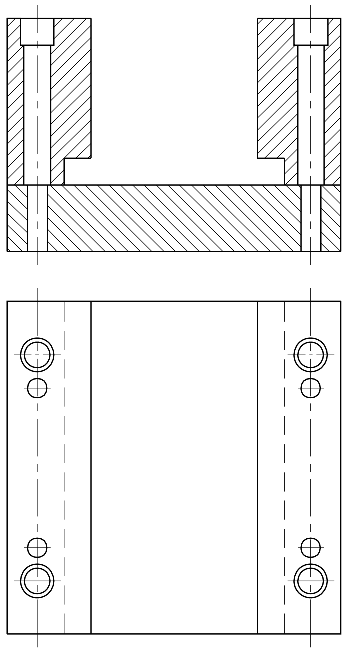
T-SLOT (WITH STRIP)

TD20N259817

Gib design (Fig 8)

In this design, a flat mould plate is selected. A shoulder is provided on the slider. The shoulder slides on an L-shaped rail fixed on the flat plate. In this case, the mould plate can be soft. In addition to rails, a wear plate is placed below the slide.

Fig 8



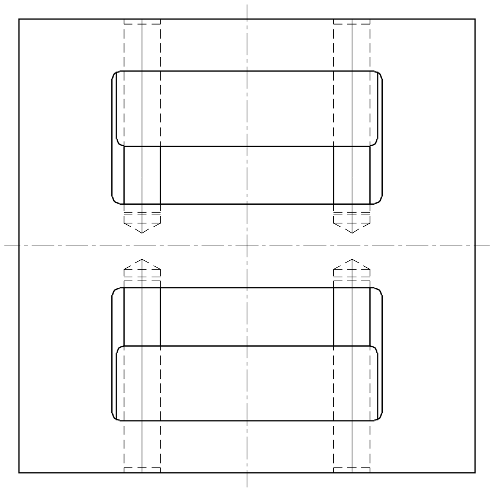
GIB DESIGN

TD20N259818

Guide pin design (Fig 9)

In this case, a blind pocket is milled in the plate. The slide is guided on two hardened and ground pins with in the pocket. The slide can be without shoulder. The plate need not be hard, but, a hardened wear plate should be given below the slide.

Fig 9

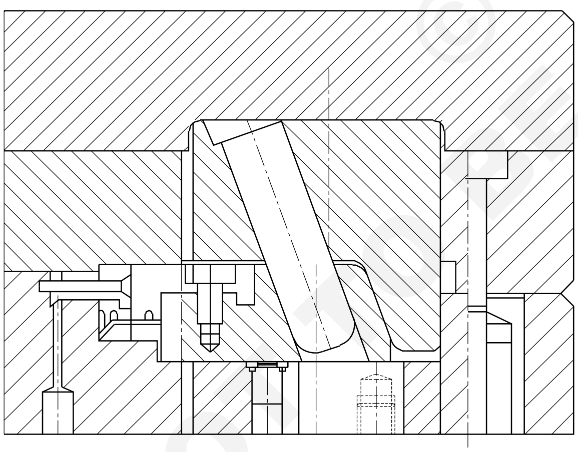


GUIDE PIN DESIGN

Sliding unit (Fig 10)

Standard sliding units are also available from manufactures like DME, HASCO etc. These integrated units incorporate sliding as well as retention mechanisms which can be easily attached to the mould plate.

Fig 10



Methods of operation

The splits are actuated by various types of cams like finger cam, dog-leg cam and cam track method. As the mould is opened, the cams, attached to the fixed half move the splits to slide across the moving plate and when the mould closes, the splits are progressively closed.

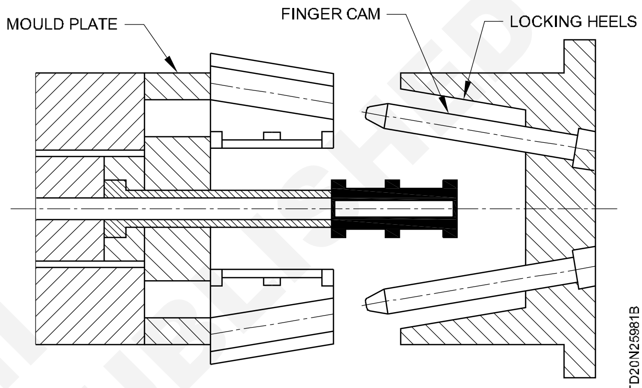
Another method of actuating the splits is, by the use of compression springs. But, as these can be used only to open the splits, the locking heels are used to close the splits. The splits can also be actuated hydraulically.

Finger cam actuation (Fig 11)

Finger cams are hardened pillars mounted at an angle in the fixed mould plate. The splits have corresponding angled circular holes to accommodate these finger cams.

As the mould opens, a finger cam forces the splits to move outwards. Split movements stops as the contact with the finger cam is lost. Further movement of moving half causes the ejector system to operate. On closing, the finger cams re-enter the holes in the splits and forces the split to move inwards. The final closing is done by the locking heels. The traverse by each split across the face of the mould plate is determined by the length and angle of the finger cam.

Fig 11



Movements of slide and finger cam

The split movement must be kept to a minimum. The clearance "C" has two purposes.

1 It ensures that the force which is applied to the split during the injection is not transferred to the relatively weak cam.

2 It permits the mould to open a small amount before the splits are actuated. In certain cases this movement can be used to withdraw the core from the moulding.

The amount of delay movement D can be calculated by the following equation.

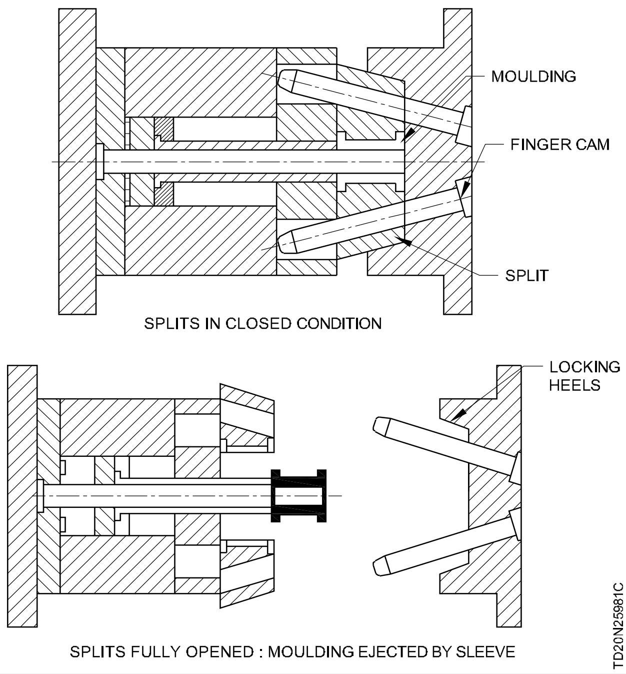
An angle of 10°-25° is given for finger cam. Larger angle is given for longer slide movement with a shorter cam pin. The diameter increases with the length of pin. The lead angle at the front is normally (+5°). One or two finger cams are used to operate each split depending on the split size.

Split design

An angular hole at 25 degree from vertical is made in the slide. A side clearance of 0.5mm, front clearance of 0.3mm and a back clearance of 0.8mm are normally given by boring or milling. For practical purpose the hole can be drilled 1.1mm bigger than the cam diameter after shifting the centre 0.25 mm towards the back clearance. This will give a clearance value approximately to 0.3mm at front side and 0.8mm at backside.

Finger cam actuated mould [Sleeve ejection] (Fig 12)

Fig 12



Wedge angle

Wedge angle is kept 2-3° more than the cam angle. The difference in angle is required for two reasons:

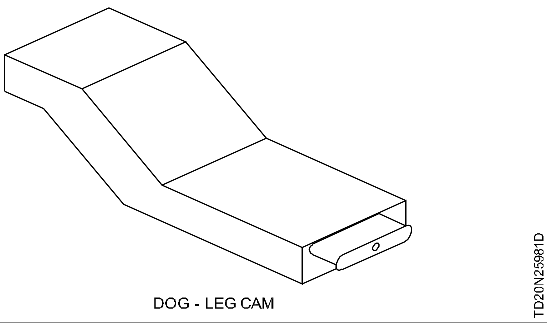
1 Maintain a gap between the moving slide and locking heel surface as the mould opens to avoid cross locking.

2 It ensures the final closing is done by the strong heel, not the weaker cam pin.

Dog leg cam actuation (Fig 13)

Dog leg cam is used where a greater split delay is required. When there is a stationary core pin in the fixed half of the mould the component may stick on the cavity side. This can be avoided by having the part encased with in the splits till the part is stripped off from the stationary core pin. In order to delay the split movement a straight portion is given on the cam. The splits do not immediately begin to open when the mould halves are opened because of the straight portion of the dog leg cams. During this time the part comes out of the stationary core pin. Further movement of the moving mould causes the movement of the splits by the angular portion of the dog leg cam. Thus the part is released from the undercut. The reverse action occurs when the mould is closed.

Fig 13

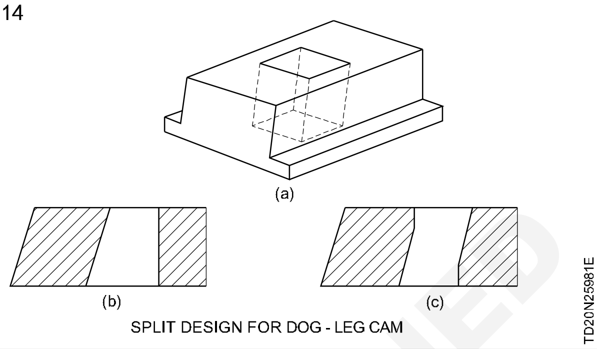


The dog leg cam has a rectangular cross section. The angle may vary from 10° to 25°. The lead in can be at tapered or radius.

Split design for dog leg cam (Fig 14)

The hole in the slide can be angular through out, one side angle and other side straight or combination of taper and straight on either side as shown below.

Fig 14



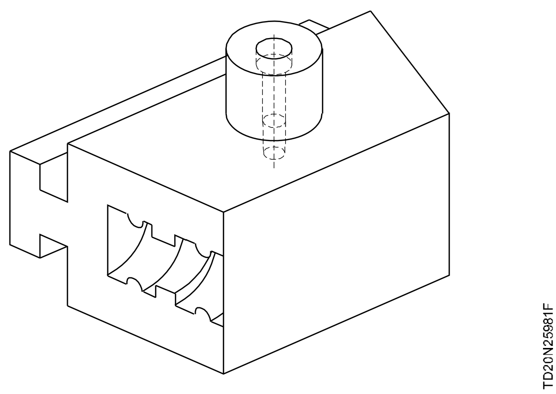
Cam track actuation

Cam track is, also, used for delayed slide movement. A cam track is machined in a steel plate. It is fixed on the fixed half of the mould. A freely rotating boss is fixed on either side of the slide. As the mould open and close the boss slides on the cam track, which results in corresponding slides movement. A wedge block is needed for final closing.

Split design (Fig 15)

In this design, no shoulder can be given on the slide as it will obstruct the cam plate movement. A freely rotating sleeve is fixed on either side with a distance screw in order to avoid friction and to reduce wear and tear.

Fig 15

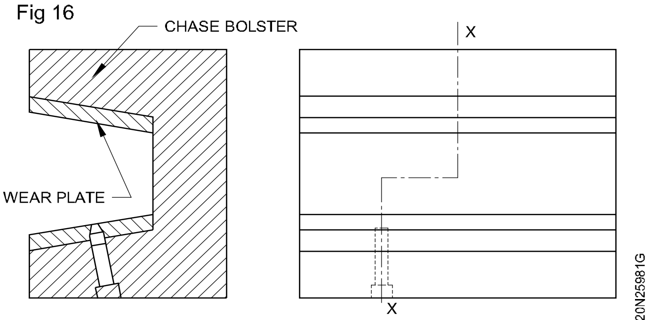


Split locking method (Fig 16)

It is essential to hold the splits rigidly, during the actual injection phase, as the high pressures developed within the impression will tend to force them apart. Sliding splits can be conveniently locked in position by the use of a chase bolster.

Each split will have a sloping or angled face accurately matching the angled face of chase- bolster. Two basic designs of chase-bolster are (1) open channel type and (2) enclosed channel type.

Fig 16

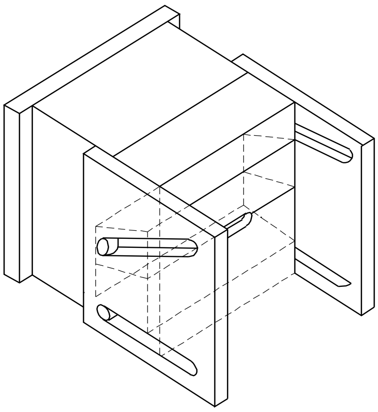


SPLIT LOCKING METHOD

Open channel chase-bolster (Fig 17)

Simplest of the designs is by machining a channel with angled sides across the width of a plate. The projections formed are called locking heels. Wear plates are used to resist wear. Open-channel chase-bolster can be made as an assembly by attaching individual heel block to a plate.

Fig 17

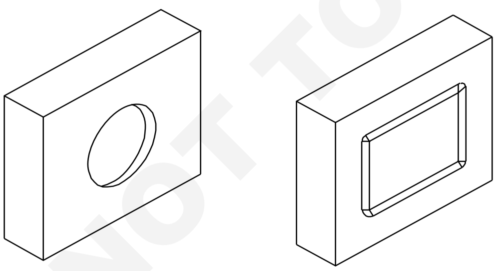


CAM PLATES FITTED TO OPEN CHANNEL CHASE BOLSTER

Enclosed chase bolster V (Fig 18)

An enclosed chase-bolster is preferred for deep splits as it results in a more rigid structure. The chase-bolster is made by machining a pocket with tapered circular or rectangular form. The taper is kept 2-3º more than the finger cam angle.

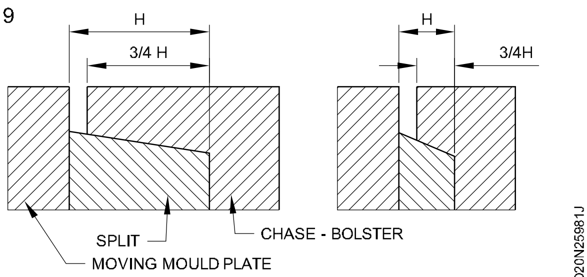
Fig 18



ENCLOSED CHASE BOLSTER

Depth of locking heel in chase bolsters (Fig 19)

Fig 19



Splits safety arrangement

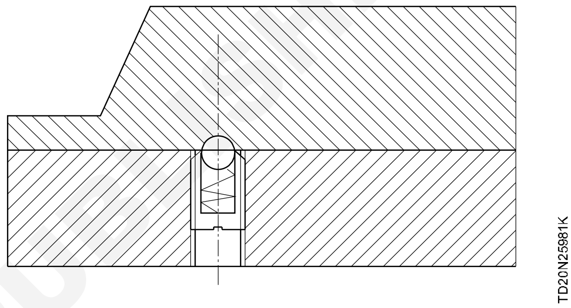
It is necessary to provide certain safety features in moulds with cam method of actuation. When the mould is fully open, the splits are not in contact with the cams.

There fore the splits may be moved out of alignment by shock, vibration or even gravity. To arrest this movement by gravity the splits should operate horizontally with respect to the machine. To prevent the movement of the splits by shock or vibration, the following methods can be used:

Ball catch method (Fig 20)

A spring loaded plunger is fitted below the surface of the split. When the split is opened the plunger is engaged in small conical depression, which retains the split nominally in that position. The distance between the plunger and the depression is equal to the movement of the split.

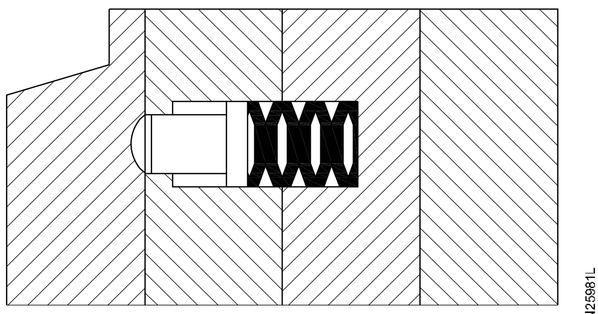
Fig 20



Spring loaded pin method (Fig 21)

The splits are spring loaded so that after they are actuated they remain in the open position. A pin fitted to the underside of the split is free to move in a recess in the mould plate. A spring is fitted between the pin and the end of the slot.

Fig 21

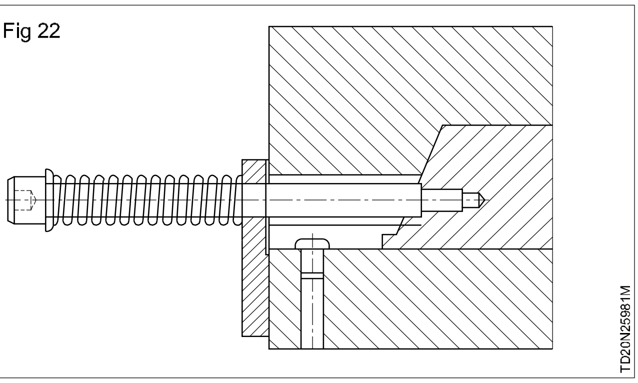


SPRING LOADED PLUNGER

Spring with pin or plate stop (Fig 22)

In this case the splits are spring loaded so that after they are actuated they remain in the open position with the help of a stop pin or plate. As we use bigger diameter spring slightly heavier slide can be used. Since the spring is in line with the split movement the compression load has to be taken by the cam.

Fig 22



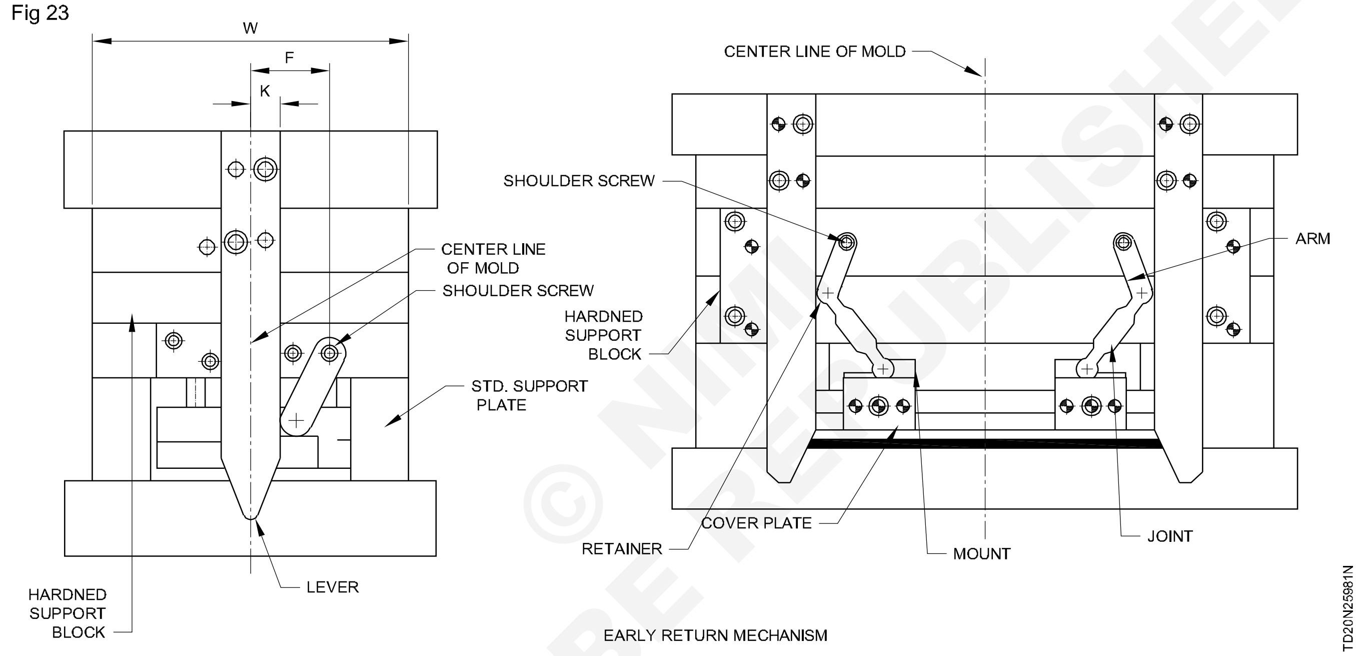
Early return mechanism (Fig 23)

When there are ejector pins or sleeves directly below the slider path the ejector system has to be taken fully back before the slider is brought in to avoid collision. The ejector system can be positively returned early using the mechanisms shown above. When the mould closes the knock out bar fixed on the fixed half hits the retainer bar which moves the ejector system back. The knock out bar moves past the retainer bar in to the clearance provided on the ejector plate and base plate when the ejector system has been fully returned to its initial position. The slide crosses the pin or sleeve if any below it only when they have been moved out off collision.

Position sensors

Some times position sensors are placed to ensure that the slide has been returned to initial position. Sensor also may be placed below the ejector system to ensure its position especially when hydraulic knock out systems are employed.

Fig 23



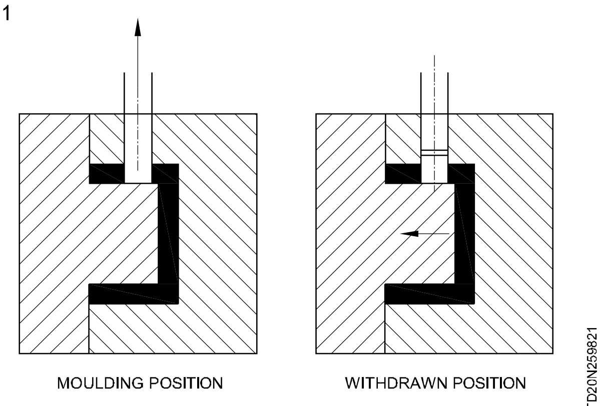
Side cores & side cavities

Objectives: At the end of this lesson you shall be able to
- **state what is side core and side cavity**
- **explain the side holes, recess or slots in moulding**
- **enumerate the design requirement of internal side core /side cavity.**

Introduction

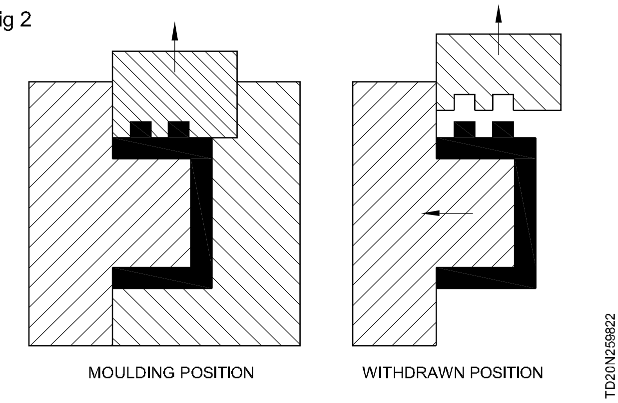
A side core is a local core which is normally mounted at right angles to the mould axis for forming a hole or recess in the side face of a moulding. This side core prevents the in-line removal of the moulding and some means must be provided for withdrawing the side core prior to ejection. Fig 1 which shows the side core in the forward and the withdrawn action.

Fig 1



The side cavity performs a similar function to the side core, in that it permits the moulding of components which are not in line of draw. This element provides for components with a projection or projections on one or more of their side faces. Fig 2 which shows the side cavity in the forward and the withdrawn action.

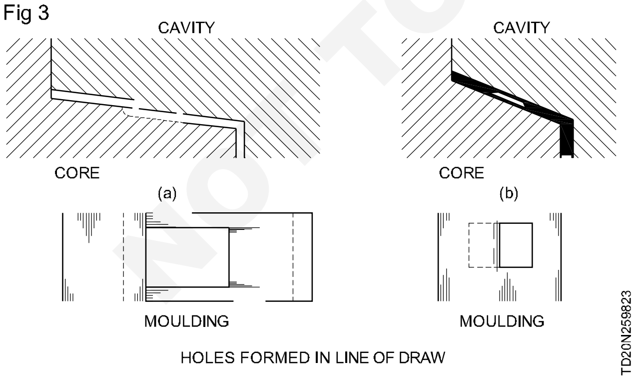
Fig 2



Mouldings representing side holes, recess or slots (Fig 3)

In general, any component which has a local recess, hole or slot which is not in line of draw will necessitate the incorporation of a side core in the mould design. However there are a few exceptions. For example, a hole in the side face of a component could be moulded in the line of draw by smart component design. The hole is formed by a part of the core abutting on to the sloping face of the cavity. To achieve this condition the component must be designed with a definite step as shown in Fig 3a. When a step is not permissible an alternative design may be adopted as in the below Fig 3b. In this the side wall of the component is caused to slope at an obtuse angle with respect to the base. This permits the hole to be formed by a projection from the core which abutts on to the cavity as shown. Please note that the top face of the projection must be such that it does not create an undercut. Moulded holes of this type are very simple and if designer agrees this can de adopted.

Fig 3



A hole can also be incorporated into the side face of the moulding by a subsequent machining operation. This method should always be considered by the mould designer before proceeding with a side core design as it has the following advantages.

1 The mould is simple and therefore relatively cheap.

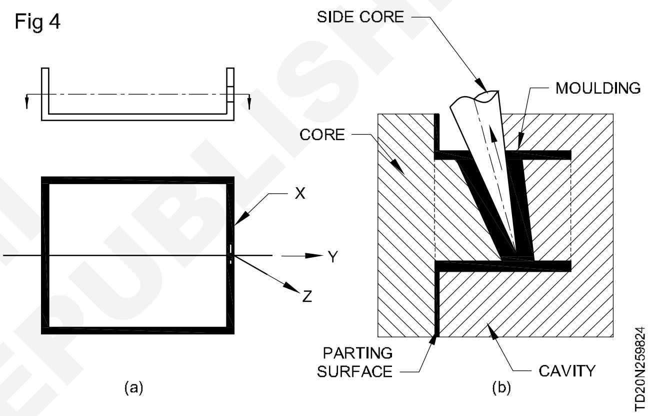
2 Ease of operation of the mould creates less likelihood of production problems.

3 Whereas a side core breaks up the normal flow of material entering the impression and there is a probability of flow lines developing, with this method it is impossible for this to happen.

The general form of the component may make use of the split design more appropriate than the side core design. However, note that if the component necessitates a slender core, the side core design can be used with advantage.

If the component has a hole that is not parallel to the parting surface as shown in the below figure, the side core must be withdrawn at a suitable angle to the parting surface of the mould (Fig 4a). Curved holes can be moulded, providing the component design allows the curved core to be withdrawn on a radiused path. (Fig 4b)

Fig 4



Now consider the case where it is preferable to use the side core technique for what would normally be considered a split mould component. The component which is shown in Fig 5 is a circular section which is having a slender hole at the middle. It incorporates a peripheral undercut on the exterior as shown. By adopting the side core design, the outside shape of the moulding is formed in the two mould plates, the component being moulded at right angles to the mould's axis. (Fig 6)

Fig 5

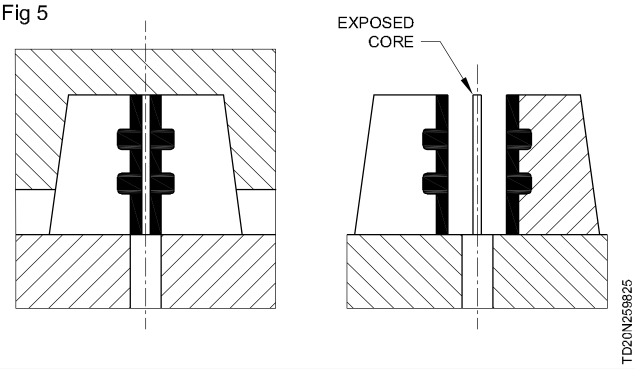
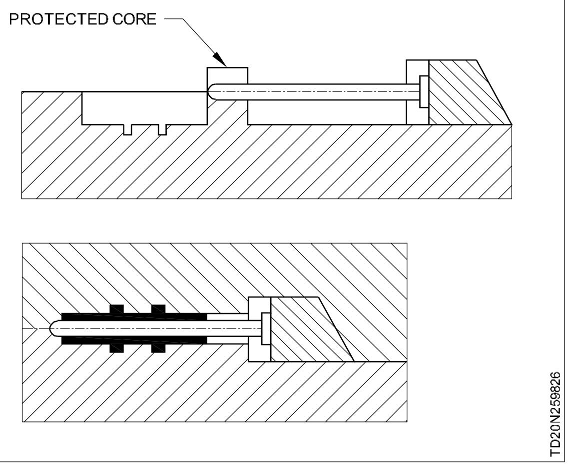


Fig 6



The advantages are

1 The long slender core is guided at all times in one of the mould plates. This protects the core against accidental damage.

2 As the component is adjacent to the parting surface, a great saving in mould height can be achieved as compared with split design. Reduction in mould thickness often allows the component to be moulded in a smaller injection moulding machine.

The design requirement for side cores / cavities

1 Arrangements must be made for guiding the assembly

2 Means must be incorporated for actuating the assembly

3 The assembly must be securely locked when in the moulding position.

Mould designs incorporating side cores and side cavities for components having holes or projections vary considerably according to the position, number and form of the restriction. The design can be placed in one of the following methods:

1 Design for mouldings with peripheral undercuts, having a slender center hole.

2 Design for moulding with holes or slots in one or more sides.

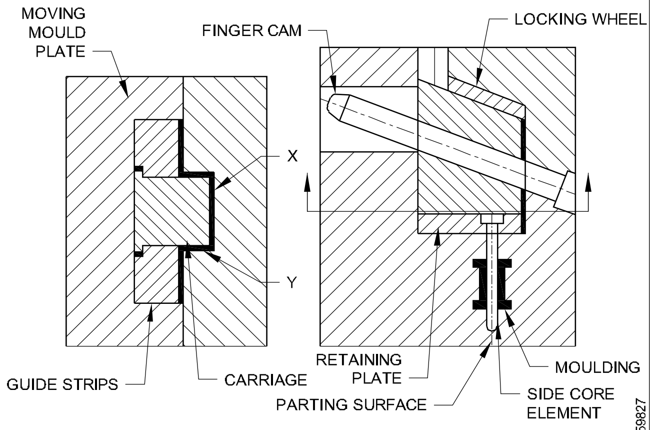
3 Design for mouldings having recess or projections in one or more sides.

4 Designs for mouldings with curved holes.

Internal side cores / cavities (Fig 7)

The side core assembly in similar to splits and methods are adopted for guiding, locking and operating it. The carriage is mounted in guides which are securely attached to the moving mould plate. The side core element is secured by the retaining plate to the carriage. The actuation of the carriage is by means of a finger cam, and is locked in the forward position by a locking heel.

Fig 7



INTERNAL SIDE CORE ASSEMBLY LONGITUDINAL AND TRANSVERSE CROSS-SECTION THROUGH PART OF MOULD

a **Internal side core or side cavity assembly details:** The assembly consists of a carriage and either a side core element or a side cavity element. These elements are secured to the carriage either directly or by means of a retaining plate. The carriage is provided by a pocket machined in both mould plates. The major part of the carriage is accommodated below the parting surface in a pocket. The width of this pocket is sufficient to accommodate the gibs. The portion of the carriage which projects above the parting surface must be accommodated in a pocket in the fixed mould plate. One face of this pocket is angled to form the locking heel.

b **Guiding arrangements:** The guiding arrangements for the internal side core or side cavity is similar to that of guiding splits. The assembly is relatively small unit.

c **Method of actuation:** The internal side core assembly can be actuated by means of a finger cam. Other method like dog leg cam actuation and spring actuation are also commonly used.

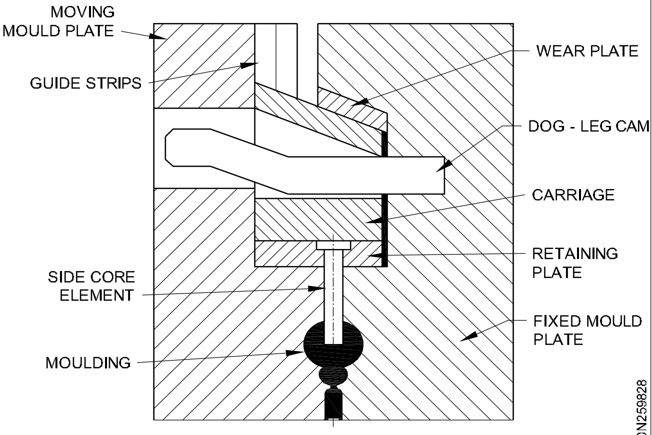
The finger cam: Method of actuation is used when a short delay is required. The amount of clearance between the cam and the cam hole determines the actual delay period. The finger cam must be of sufficient length to withdraw the side core from the moulding completely

The dog - leg cam: (Fig 8) Method of actuation is specified if a longer delay period is required. It is desired to withdraw the moulding completely from the fixed half before retracting the side core. The ensures that the moulding remains in the moving half in readiness for ejection

T**he spring loaded system: (Fig 9)** This is used for mouldings with shallow undercuts or projections. The figure shows a section through a side core assembly. The component has a small indentation in one side formed by a side core mounted in a standard carriage assembly. For spring actuation, a stud is attached to the bottom of the carriage as shown in figure, and this is accommodated in a slot machined in the mould plate.

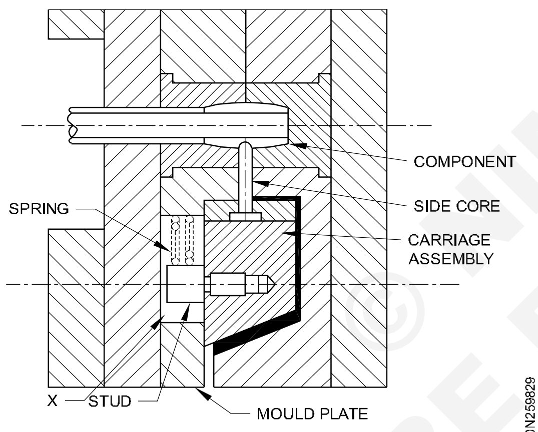
A spring or springs are fitted in the slot and cause the side core assembly to withdraw immediately the mould opens. Note that the locking heel is used to progressively return the assembly when the mould is being closed.

Fig 8



DOG - LEG CAM METHOD OF ACTUATING SIDE CORE ASSEMBLY

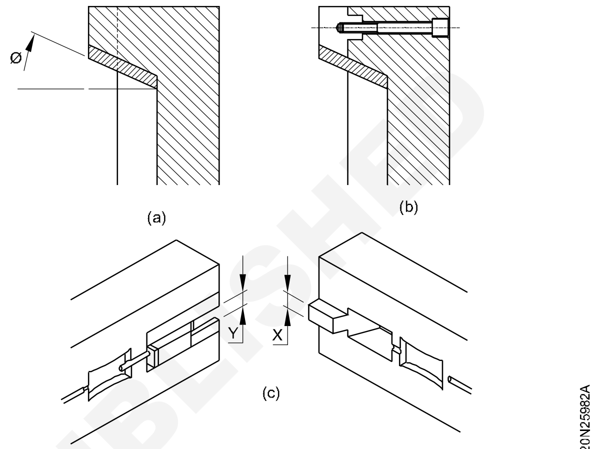
Fig 9



SPRING - LOADED SYSTEM FOR ACTUATING SIDE CORE ASSEMBLY

d Locking the carriage assembly (Fig 10): The final closing movement and lock of the carriage assembly are by means of a locking heel. The angle specified for locking heel, and the corresponding angle of the carriage are normally 150 and 400. When the finger cams are used to actuate the carriage, the angle must always be greater than the operating angle. To provide wear resisting surface for the locking heel, and to provide an adjustment for wear, wear plates are fitted on the angled surface. The locking heel normally protrudes above the parting surface and two alternative designs.

Fig 10



LOCKING HEEL

Moulding internal undercuts

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

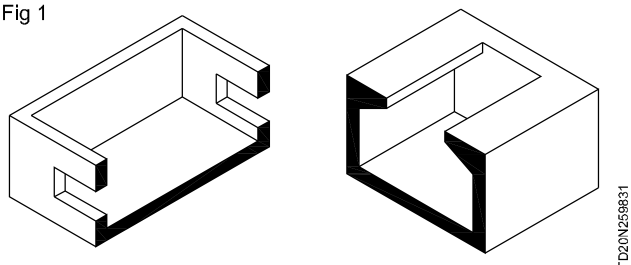
- **describe the different methods used in moulding internal under cuts**
- **explain the uses of the following in moulding internal undercuts**
- **form pins**
- **split cores**
- **side cores**
- **exlain the method of stripping internal undercuts.**

Moulding internal undercuts: Any restriction which prevents a moulding from being extracted from the core in line of draw is termed as internal undercut. The method adopted for relieving internal undercuts depends upon the shape and position of the restriction.

If the undercut is local the restriction may be incorporated in a form pin.

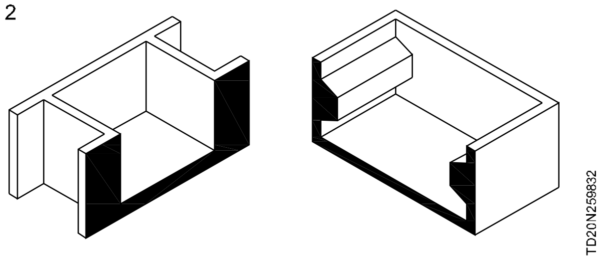
In some cases the under cut extends completely along the wall of the component. Such components are moulded by incorporating a core which is of two parts. The moving part is termed as split core. (Fig 1)

Fig 1



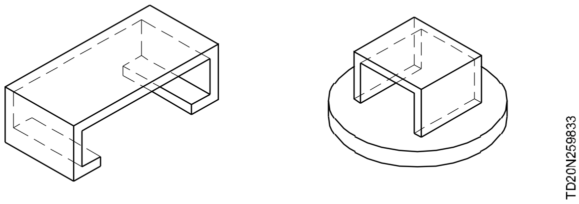
This design can be used for components which have undercuts on opposite faces. (Fig 2)

Fig 2



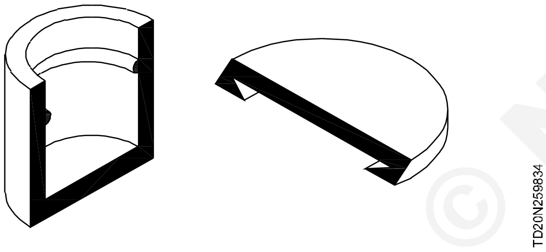
The conventional side core design can be used for certain internal undercut components. But this method is restricted to those components which have an open side. (Fig 3)

Fig 3



Certain type of components can be stripped from the core. In such cases the cost of the mould will be less. (Fig 4)

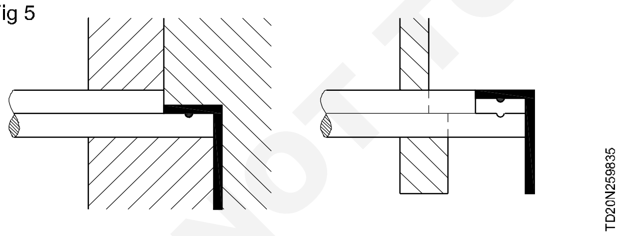
Fig 4



Form pin : The under cut form is provided on a form pin. The form pin can have a straight action or it can have an angled action.

Straight action form pin (Fig 5) : This design is used for components with an undercut on one internal wall only.

Fig 5



Form pin angled action : The working face of the pin is made to move inwards relative to the core during ejection. (Fig 6)

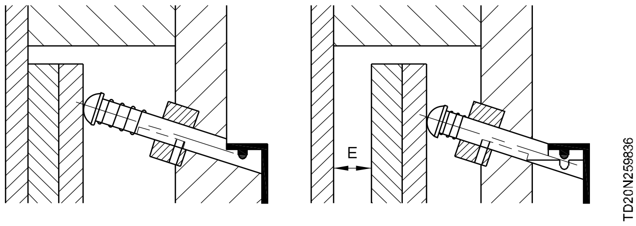
This design is used for components which incorporate internal undercuts on one or more walls.

The amount of withdrawal obtained may be computed from the following relationship.

$M = E \tan \theta$

Where M = the withdrawing movement

Fig 6



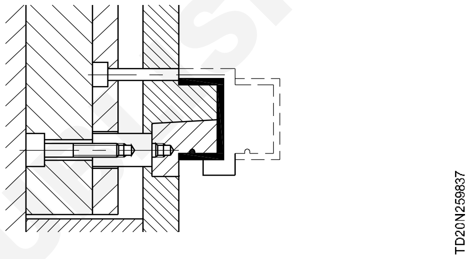
E = ejection movement

q = fitting angle of form pin.

Split cores: A split core design is used for components which have internal undercuts that cannot be incorporated on a form pin. The split core can be moved forward for ejection either in a straight plane or an angled plane.

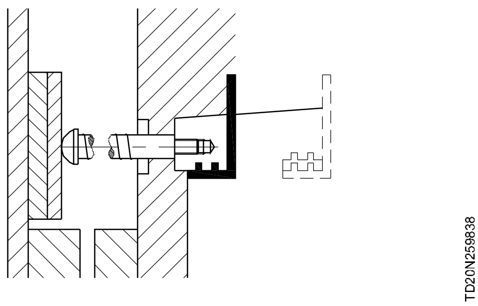
Split core straight action: This design is used for components with an external undercut on one wall only. (Fig 7)

Fig 7



Split core angled action: In this design the core is made to move outwards during the ejection. This withdraws the restriction and allows the moulding to be extracted in line of draw. It is used on components with undercuts on opposite faces. (Fig 8)

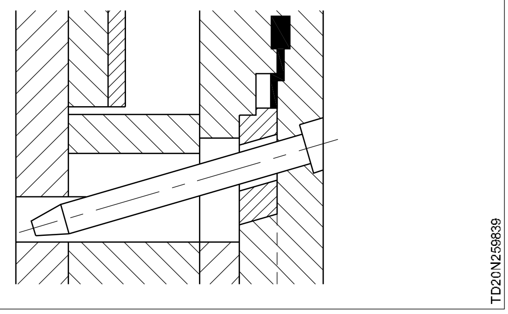
Fig 8



Side cores: Some components with internal undercuts can be moulded using a conventional side core design. (Fig 9) However this method is limited to components which have atleast one open side. This permits the side core to be withdrawn through this side to relieve the undercut and allow the moulding to be ejected in line of draw.

Stripping internal undercuts: A simple and effective method for stripping an internal undercut is to strip the moulding off the core.

Fig 9

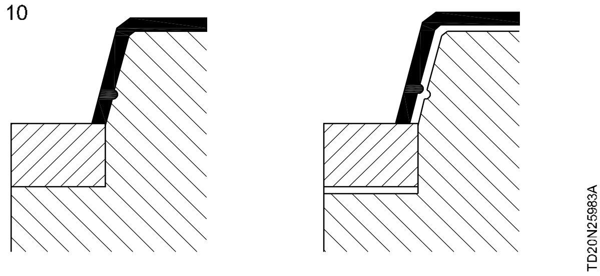


The stripping depends on

- the shape of the undercut
- the elasticity of the material
- whether the external form permits expansion during ejection.

The stripper plate design is the most common method used for stripping this type of components (Fig 10). Valve ejection and sleeve ejection can also be used. The component should be free to expand during ejection.

Fig 10



Moulds for threaded components

Objectives: At the end of this lesson you shall be able to
- **explain the factor to be considered while designing moulds for components with threads**
- **describe the methods of moulding threaded components**
- **explain the different methods employed in the removal of internally threaded components**
- **explain the different methods employed in the removal of externally threaded components.**

Moulds for threaded components

A thread is a form of "undercut". This increases the intricacy of the mould design. The extent of complication varies and it is dependent on the following factors:

Type of thread	- internal, external, continuous or discontinuous.
Method	- thread form moulded or incorporated by the use of a metal insert.
Type of production	- manual, semiautomatic or fully automatic.
Other considerations	- whether the thread form allow for stripping.

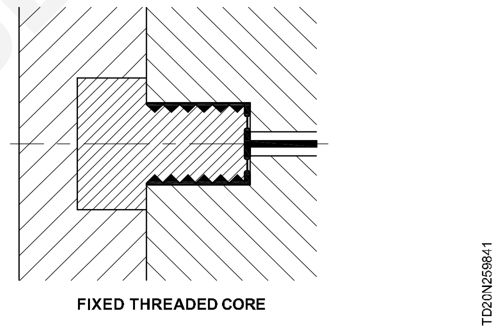
The moulded component which incorporates the thread may be classified as external or internal depending upon the type of thread. Small diameter internal threads are incorporated by moulding in metal insert which has the required thread profile in it. This design eliminates the necessity for long threaded slender cores which are very susceptible to damage.

Moulds for internally threaded components

Fixed threaded core design: Fig 1 shows a section through a simple mould of this type. The threaded form is incorporated in a fixed (non-rotating) core attached to the moving mould plate.

When the mould is opened the moulding remains on the core. It is unscrewed by the operator.

Fig 1



FIXED THREADED CORE

The advantages with this system are

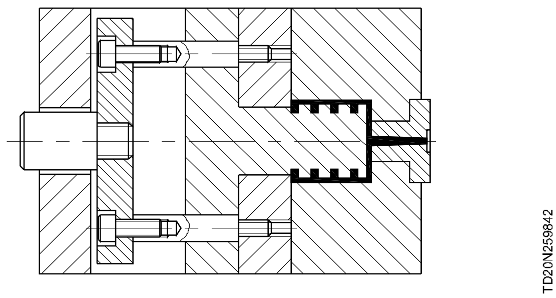
- Mould cost is considerably less.
- As there are no moving parts in the mould, the servicing cost is kept to a minimum.

The major disadvantage with this system is that for multi impression moulds, considerable time is consumed in manual unscrewing. This will increase the mould cycle time.

Stripping internal threads: The internally threaded components can be stripped from the core if

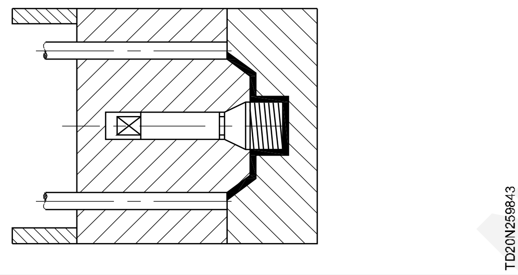
- the thread form is roll thread
- the plastic material has sufficient elasticity during the ejection phase. (Fig 2)

Fig 2



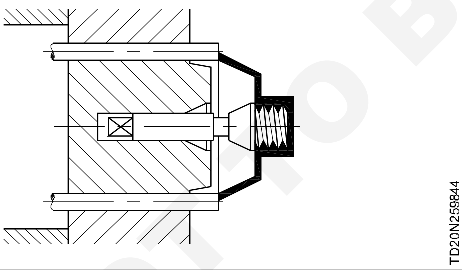
Loose threaded core : In the mould the threaded hole in the moulding is formed by the loose threaded core. The loose core is accommodated in a pocket machined in the main core. When the mould is opened the moulding is ejected by an ejector pin system. The loose threaded core is ejected with the moulding. The moulding is unscrewed from the loose core. Two sets of loose cores are used during the production. At the end of the first moulding cycle, the second set of cores are inserted into the mould and the next cycle is commenced. During this cycle the first set of loose cores removed from the first batch of mouldings are made ready for the insertion. (Fig 3)

Fig 3



When a threaded hole is incorporated with a boss, on the interface of the component, the boss will restrict the ejection of the moulding from the core. This restriction will cause distortion of the component. (Fig 4)

Fig 4



Unscrewing moulds: Manual unscrewing of individual components from the mould will be un economical in mass production because of the larger time consumed by the process. If the component design does not permit stripping of the threads, automatic unscrewing should be provided (Fig 5). To provide rotary motion an unscrewing unit is fitted behind the moving mould in place of the conventional ejector unit. In case when a positive ejection is also required an ejector system may also be incorporated (Fig 6).

There are three possible designs:

- the threaded core is merely rotated to remove the moulding
- in addition to being rotated it is also withdrawn
- the cavity can be rotated (in case of external threads).

Fig 5

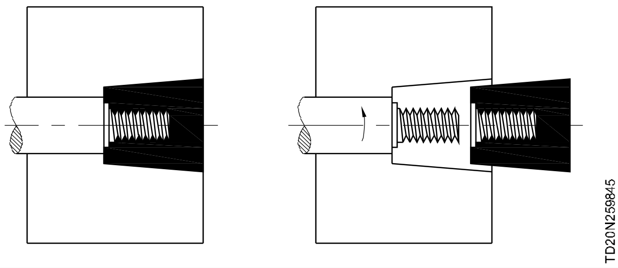
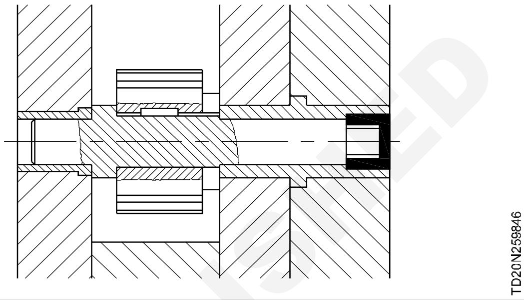


Fig 6



Various power systems are available to actuate the unscrewing mould.

Manual

Mechanical

Hydraulic or pneumatic

Electric.

There are alternative methods of linking the power sources to the mould.

Gear train

Chain and sprocket

Rack and pinion

Worm and worm wheel.

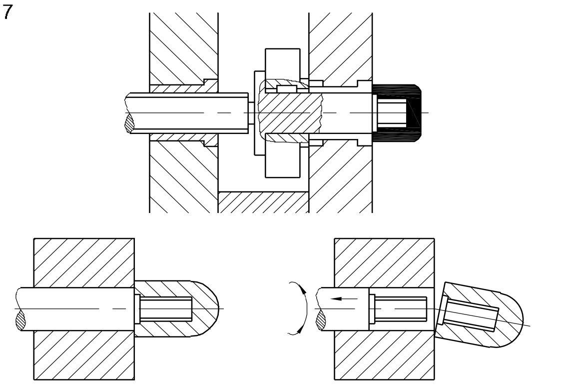
Axially fixed rotating core: This design is used for components whose external form permits the cavity to be located in the same mould half as the threaded core. When the threaded core is rotated in an axially fixed position with respect to the cavity the moulding is progressively ejected, provided the cavity is maintained in a stationary position.

The external shape of the moulding should be such that it cannot rotate with the core during ejection phase. Smooth cylindrical components are unsuitable for this technique because the moulding will rotate with the core once the initial cavity - moulding adhesion is broken.

Withdrawing rotating core: The threaded core is unscrewed from the moulding by progressively withdrawing the core. (Fig 7)

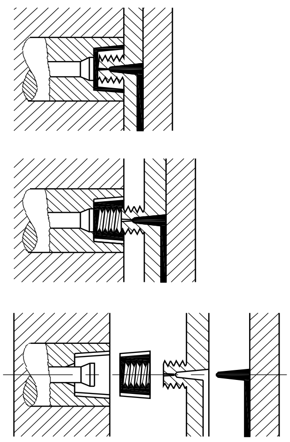
Rotating cavity (Fig 8): When gate marks are not permissible on the external surface of the moulding, rotating cavity design is used. The feeding is done through the centre of the core. Pin gate can be adopted. The gate is mounted on the injection half.

Fig 7



WITHDRAWING TYPE OF ROTATING CORE

Fig 8

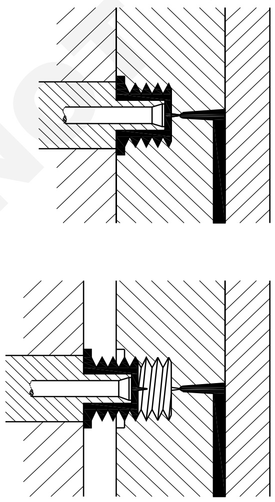


The rate at which the moulding is unscrewed must be synchronised with opening movement of the mould. If the mould opens too fast the moulding will be pulled from the cavity and the rotation of the cavity will be pointless. If the mould is opened too slowly the rotating cavities will tend to jack open the mould.

Moulds for externally threaded components

Fixed threaded cavity design: In the simplest form the threaded portion is machined directly into the cavity insert.When the mould is opened the moulding can be unscrewed from the mould plate. (Fig 9)

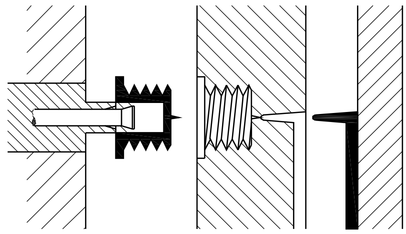
Fig 9



Automatic unscrewing method for externally threaded part.

Automatic unscrewing: The design is same as the rotating cavity design. However the cavity is fixed and the core is rotated to provide the unscrewing action. (Fig 10)

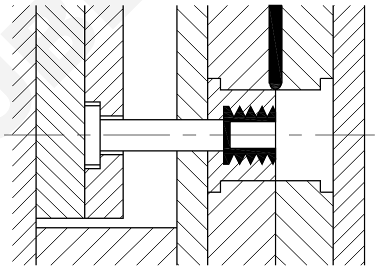
Fig 10



Stripping external threads: External threaded components can be stripped from a cavity provided

- the component has a roll thread form
- the thread depth is relatively shallow in relation to the diameter
- the moulding material is sufficiently elastic to return to its original shape after being deformed. (Fig 11)

Fig 11

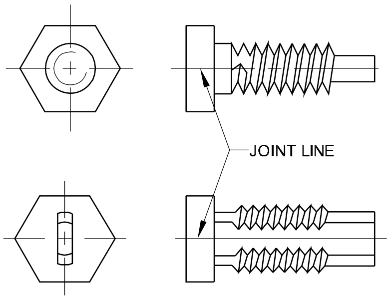


STRIPPING EXTERNAL THREADS

With conventional threads of `V' or square form this design cannot be employed.

Threaded splits: The principle of operation is same as that for ordinary split mould design. But in this case the undercut is a thread form and the joint line occurs along the thread. The split movement can be provided by dog leg cams. (Fig 12)

Fig 12



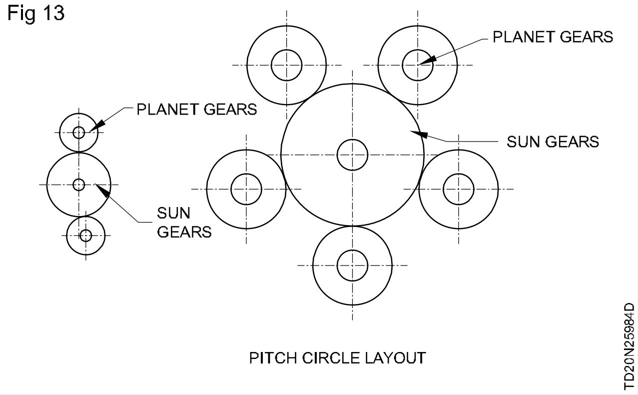
ON EXTERNALLY THREADED COMPONENTS MOULDED IN SPLITS

Layout of impressions

The layout mainly of three types

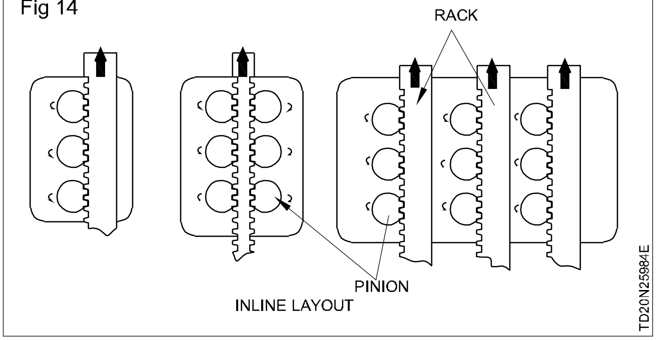
A Pitch circle layout (Fig 13)

Fig 13



B Linear (Fig 14)

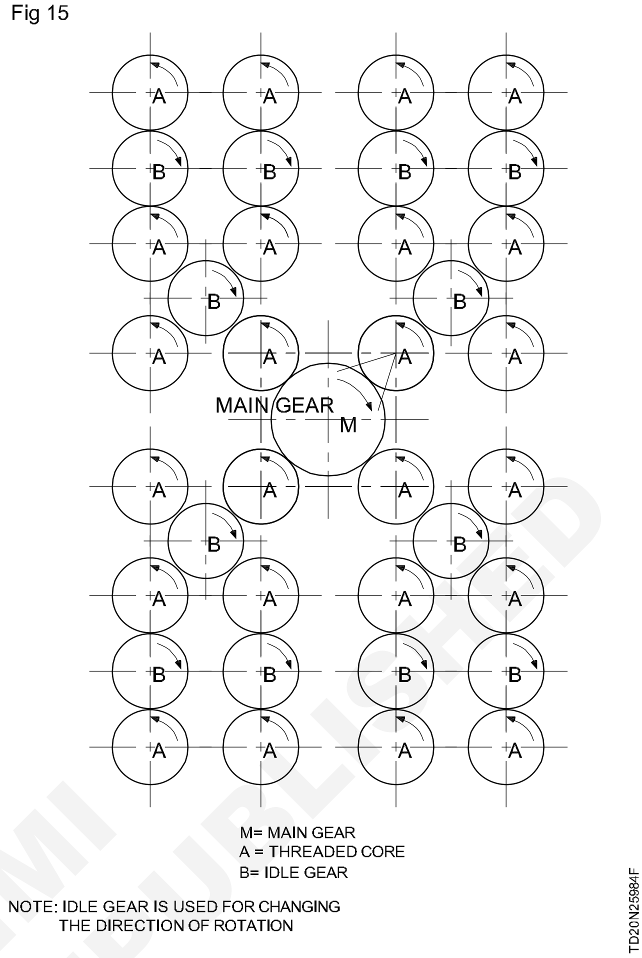
Fig 14



C Combination of linear and radial

24 cavity layout for rotating cores mould (Fig 15)

Fig 15



Moulding of thermo set material

Objectives: At the end of this lesson you shall be able to
- **brief the processing methods of thermo-set materials**
- **explain the compression moulding methods and its related terms**
- **list out the advantages and disadvantages of semi positive mould over fully positive mould**
- **enumurate the process and types of transfer moulds and advantages and disadvantages**
- **brief the process of injection moulding of thermoset and its advantages and disadvantages**
- **list out the defects of compression and transfer moulding.**

Introduction

Thermo-set moulding is one of the oldest methods of the manufacturing plastic components. Thermo-set materials are characterized by their excellent heat resistance, electrical resistance and their strength and rigidity.

The various Thermo-set components are electrical switches, plugs, holders, house hold items like cooker handles, vessels, plates, electronic components like IC, connectors, etc.

Processing methods

Thermo-setting resins are usually moulded in hydraulic presses. The pressure is, normally, up to 210kg/cm2 .The raw materials are in the form of powder, flakes, sheet or pellets. The mould is heated and the material softens and flows under the action of heat and pressure. The curing time ranges from 15 seconds to 20 minutes. During moulding, the material undergoes a chemical change; hence, reprocessing is not possible. The volatile gases produced during the chemical change should be allowed to escape. So, it is necessary to open the mould slightly during moulding. The mould is, then, immediately closed for the final curing. This opening and closing of the mould is known as breathing. Generally, the raw material is preheated up to 100°C around half an hour.

Thermo-set parts are generally manufactured by the following processes

1 Compression moulding.

2 Transfer moulding.

3 Injection moulding.

Compression moulding

Compression moulding is a method of heating and curing the material inside a mould. The material can be in powder, flakes or pellet form. It is kept in a hot cavity and the mould is closed to allow for the material to flow and fill the cavity. The material solidifies and set inside the cavity. The cavity is given enough space above the part level in order to accommodate the high volume of the powder form of material. So, the size of the cavity depends on the bulk factor of the material.

Definition

Bulk factor is defined as the ratio of the volume of the loose powder to the volume of the moulded part

Bulk factor can vary as low as 1 to as high as 18.so the cavity would have to be much deeper to contain material of the high bulk factor.

To minimize the effect of varying bulk factor, with respect to the size of mould cavity, preforms are used.

Preforms are small pellets or tablets of moulding material that are formed to shape in a special perform mould at room temperature, so no curing takes place, only densification.

Other advantages of preforms are:

a Preforms heat faster, thus reducing the mould cycle.

b The mould charges are of uniform weight.

c Mould costs less because of smaller cavity size.

d Cavity fills faster.

The bulk factor is given by the formula:

$$\text{Bulk factor} = \frac{\text{Volume of the powder}}{\text{Volume of the moulded part}}$$

Pellet

The volume of the pellet is approximately half of the powder. You have to consider, whether, the saving in heating power would be higher than the cost of pelleting.

The heating temperature is generally between 130ºC - 350ºC. The curing time is 1-2 minutes for large components and, few seconds for small components. The mould is opened one or two times, before curing, in order, to allow for the gaseous by-products to escape. Then, the mould is finally closed to give a void free component.

Compression moulding methods

Compression moulds can be classified according to different characteristics.

Depending upon the type of operation, it can be classified as:

1 Hand compression moulds.

2 Semi- automatic compression moulds.

3 Automatic compression moulds.

Depending upon the flash in moulding, they can be classified as:

1 Open flash mould

2 Horizontal flash mould

3 Vertical flash mould

4 Inclined flash mould

Depending upon the Land provided around the cavity, they can be classified as:

1 Internally landed

2 Externally landed

Depending upon the number of cavities, they can be classified as:

1 Single cavity mould

2 Multiple cavities mould

Depending upon the pressure applied on the component, they can be classified as:

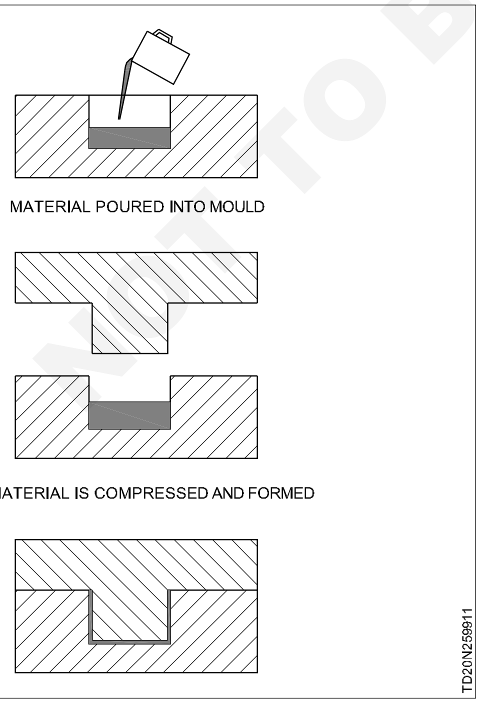
1 Positive mould

2 Semi-Positive mould

Hand compression mould (Fig 1)

Hand moulds, are, usually small in size and do not weigh more than about 12 kg for ease of handling .The material is put in to the cavities and two halves of the moulds are assembled and placed between the platen of the press. After the press has been closed and the parts moulded, the mould is opened and removed from the press and the parts ejected from the mould on a conveniently located bench. The process is, then, repeated. Hand mould production is limited to small parts, short runs, or prototype work. Hand moulds being removed from the press, facilitates the loading of inserts.

Fig 1



Semi-automatic moulds

The term semi-automatic is used to cover moulds from which the moulded articles are ejected by movements of mould parts actuated by the press. After ejection, the press operator, then, places the moulded parts in a suitable container or on a bench for flash removal etc.The operator cleans the mould of the excess material and the process is repeated. Almost, limitless varieties of parts can be produced by semi-automatic compression moulds. There are different types of semi-automatic compression moulds depending upon the direction flash line and its thickness.

They are:

1 Semi-automatic open flash mould.

2 Semi-automatic fully Positive mould.

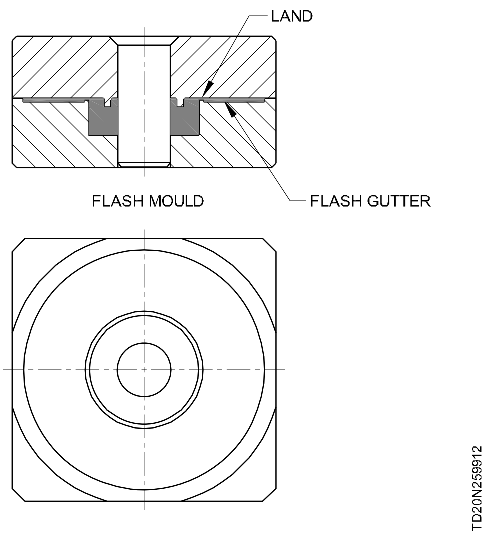
3 Semi-automatic landed positive mould.

4 Semi-automatic semi-positive mould

Semi-automatic open flash type mould (Fig 2)

The first mould constructed was of flash mould. The open flash mould is the simplest type of mould. In this type of mould, excess material is allowed to flow over the lands, which are about 2-3 mm wide out around the cavity. Clearance between 0.5 mm to 1 mm is provided between the punch and the cavity, to permit the out flow of the excess material. The moulding is never subjected to high compression in this mould. Hence, the components produced are not very dense. This type of mould is, mainly, used for thin and shallow components. For manufacturing buttons, dinner ware plates and saucer, flash type moulds are preferred. These types of moulds may be used to produce large parts from Rubber, DMC and SMC mounding compounds. The mould is very cheap.

Fig 2



The flash type mould is, not generally, used with coarse filled material or pieces which required high density. The disadvantages are

1 There is no positive location between mould halves. Location will have to be provided in the form of substantial alignment dowels.

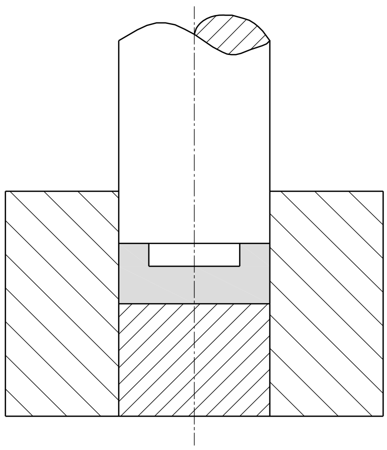
2 More wastage of materials.

3 The cavity has to be large enough to contain the full charge of material. If, the powder is used, waste will be more due to powder being forced out of the cavity during closing of the mould.

To reduce the material lost, the original charge has to be correspondingly increased, otherwise, a porous moulding results, also, the closing speed of the mould has to be carefully controlled .The higher closing speed which is used , the more material will be lost, waste is, therefore, excessive in this type of mould.

Semi-automatic fully positive mould (Fig 3)

Fig 3



FULLY POSITIVE MOULD

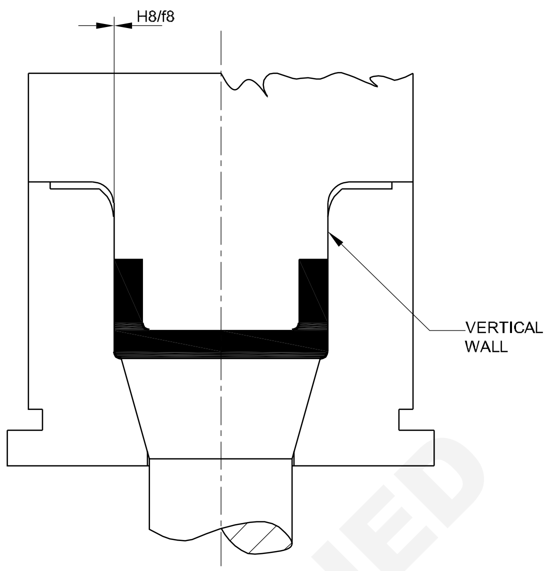
The fully positive mould is used primarily with material containing coarse fillers.

In a positive mould, there is very limited means for the excess material to escape. The flash between the top and bottom force can be produced in the direction of pressure, normal to it or at an angle to it. The flash produced in the direction of the pressing the loading chamber section is the same as the moulding contour. The flash thickness varies with the accuracy of the fitting of the top force and the loading chamber. In a positive mould, almost, the full force is applied on material and very little material is allowed to escape. The clearance between plunger and cavity varies between 0.05 mm to 0.13mm per side depending upon the size of the moulding and plastic material to be moulded. The component is very dense, flash free, but, not dimensionally accurate. This type of mould is used for large deep draw parts where maximum density is required. If very bulky long fabric or glass filled compounds are specified adequate loading space can be provided. Such moulds are usually limited to single cavity production where each charge of material must be weighed to assure depth of height and density.

Semi-automatic landed positive mould (Fig 4)

Multi-cavity moulds may use the landed positive design. The telescoping length is, also, deeper in this case. The thickness of the part is determined by the external land area.

Fig 4



SEMI - AUTOMATIC MOULD
(POSITIVE MOULD WITH OUTER LANDING FACE)

Landed positive feature will cause some trouble, if care is not taken at all times to remove flash from the land surfaces. Continued operation with flash left on the land surfaces will probably results in damage or breakage of the lands. For this reason, it is advisable to make provision for additional pressure pads out side the cavity.

Semi automatic semi-positive mould

A positive mould cannot be used in every application. When the component has radius around the top edge a flange like razor edge appears on the punch in a positive design. This must be eliminated by providing a straight portion around the radius. In both, positive and semi-positive moulds in corporate a powder well above the cavity surface, the differences being that, in the semi positive type, there is a taper (0.25 - 1°) on the side wall of the powder well and, on the positive type walls are parallel. As the two halves of the mould begin to close, mould acts much like a flash mould as the excess material is allowed to escape. As the plunger, telescopes into the cavity full pressure is exerted on the material and produces piece parts of maximum density. There is a little clearance between side walls of the cavity, which results, in a very thin flash being formed.

It has the advantages of both flash and positive type moulds. Semi-positive mould are further classified into three according to the direction of flash line.

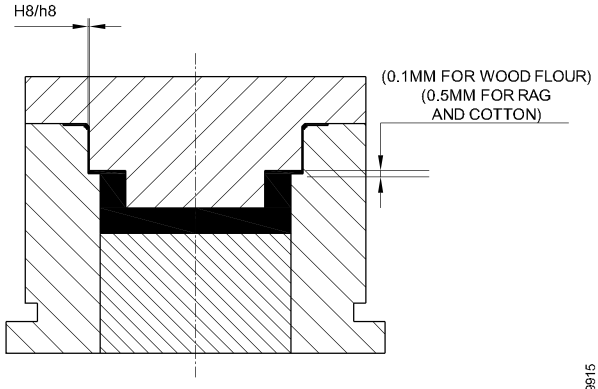
They are:

1 Horizontal flash mould (Fig 5)

2 Vertical flash mould (Fig 6)

3 Inclined flash mould (Fig 7)

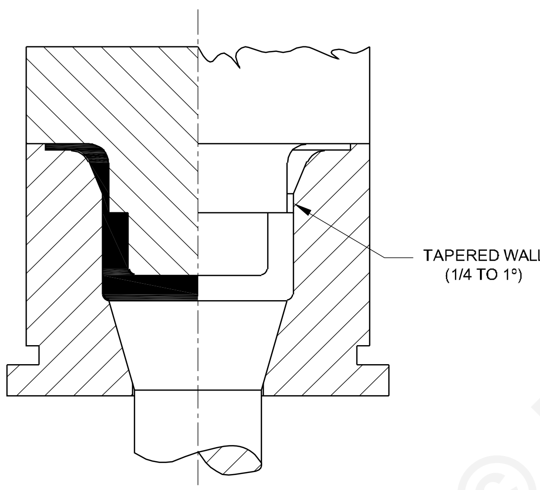
Fig 5



SEMI-POSITIVE MOULD
HORIZONTAL FLASH TYPE

TD20N259915

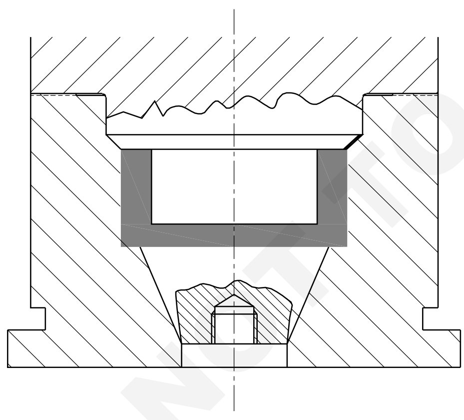
Fig 6



SEMI-POSITIVE MOULD
VERTICAL FLASH TYPE

TD20N259916

Fig 7



SEMI-POSITIVE MOULD
INCLINED FLASH TYPE

TD20N259917

Advantages and disadvantages of Semi-positive mould over Fully-Positive mould

Advantages

1 This mould has the ability to extract out the gases and excess moulding material easily.

2 Some, thermo-set material is moulded easily in a taper powder-wall than a vertical sided powder-wall.

3 Urea formaldehyde can be easily moulded in such type of mould. Fully positive mould is recommended for Phenol formaldehyde.

Disadvantages

1 This mould shows lack of proper alignment between top and bottom force.

Automatic compression mould

In an automatic compression moulding operation the closing of the mould, ejection of the part and removal of the part from the mould are made automatic. Sometimes, the loading of the material also, can be made automatic.

Compression mould can be made for the following applications.

1 Compression mould with linear or rotating table: When this type of mould is used for bigger components, the bottom half is moved or turned out from under the punch, and thus, the component can be is easily removed. The component may be pulled of the core by compressed air.

2 Compression mould with ejection system functioning in two or three steps: When the component is provided with a jacket (side walls) it can be safely removed from the mould, only, if one of the side wall (out side surface) is removed freed from the other(inside wall).

3 Compression mould for internally threaded component: The threaded component is moved out of the core by means of ratchet and wheel or hydraulically actuated racks. Collapsible core is used, if the thread is interrupted.

4 Compression mould for externally threaded component: External threads may be made using wedge inserts or cam operated splits.

5 Compression mould for under cuts: Undercut moulding can be made with mechanically or hydraulically actuated slides.

Transfer moulding

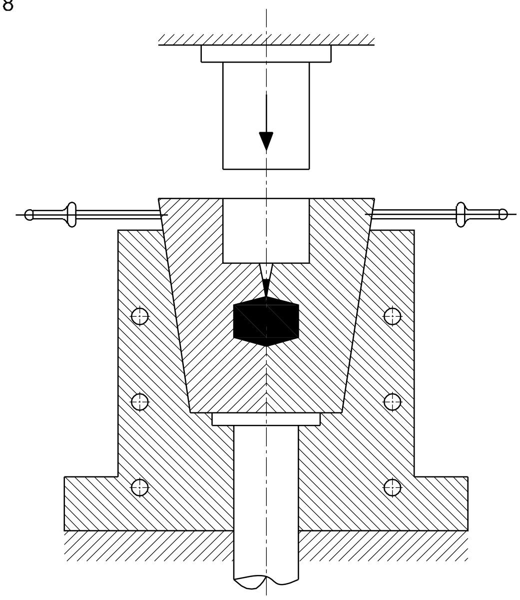
In transfer moulding, the material is transferred from pot through runner and gate into the closed impression. The material placed in the hot pot is pressed by a plunger. According to the plunger construction, transfer moulding is classified into two. If, the plunger is part of the mould, then, it is called as pot transfer mould and if, it is a part of the machine, then, it is known as plunger transfer mould.

The material required in transfer moulding, is more, as there is loss of material after each moulding. Some materials, remain in the channel and at the bottom of the pot. Parts produced by transfer moulding exhibit good electrical insulation due to uniform curing. Material with good flow characteristics only can be transfer moulded.

Transfer mould, also, should be heated, but, the plunger temperature should not exceed 80ºc. Big mould needs cooling.

Pot transfer moulding (Fig 8)

Fig 8



POT TRANSFER MOULD

TD20N259918

In this process the material flows from a pot into the closed cavity at high velocity. The kinetic energy of the high velocity flow is converted into heat which in turn heats up the material very much and uniformly during the flow. There fore only a short time is required for the completion of the thermosetting reaction. In most of the cases the material is preheated so that the cycle time is reduced further.

Pot transfer moulding is done in a three plate mould. The plunger is fixed on the upper part of the mould. The core and cavity plates are closed first. The operator places the preheated material into the pot. Then the plunger plate is actuated which forces the liquid material into the impression.

The diameter of the pot can be roughly calculated using the formula:

D = G1/3

Where, D = Diameter of pot

G = weight of the component

Plunger transfer moulding (Fig 9&10)

In plunger transfer moulding the plunger is not part of the mould but part of the machine. The plunger can be made to operate from either top or bottom of the machine. The press platen holds the mould halves together.

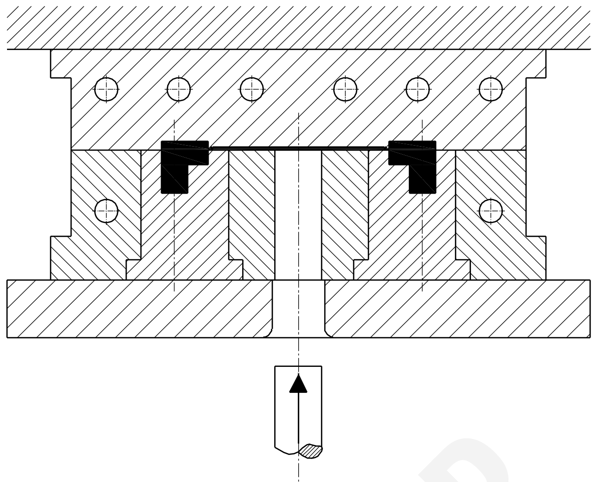
Advantages and disadvantates of transfer moulding

Advantages

1 Uniform heating. (Material flowing through a relatively small diameter is heated more uniformly than the material poured in the compression moulds).

2 Faster moulding cycle possible. (due to less curing time required)

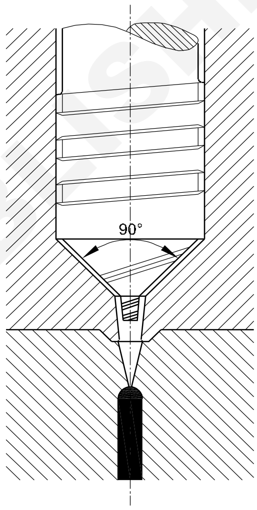
Fig 9



PLUNGER TRANSFER MOULD

TD20N259919

Fig 10



CONICAL PLUNGER OF MOLD WITH NON-OPENING TRANSFER POT

TD20N25991A

3 Closer tolerance can be achieved.(since, the material injected close to the cavity less flash is produced)

4 Slender Insert moulding possible.(Less chance of deflecting the inserts)

5 Encapsulation possible. (delicate parts can be easily covered)

6 Different components possible in one mould. (same curing time)

7 Thin-walled components can be made.(burning of material due to excessive curing period can be avoided)

8 Material volume need not be exact. (remaining material removed as biscuit)

9 Thick-walled components can be most favorably made. (Less chance of uncured inclusions)

10 Good electrical insulation. (due to uniform curing)

Disadvantages

1 High mould cost in certain cases.

2 More material wastage.

3 Not all qualities of materials are fit for transfer moulding.

4 Properties, not uniform, in certain cases. (Moulding will be slightly laminated in flow direction).

Injection moulding of thermoset

Although, not widely practiced injection moulding, also, dominates high volume

Thermo-set part production. Materials like epoxy, alkyds, di-allyls and DMC can be successfully injection moulded.

The processing methods are quite different from thermoplastic injection moulding. The injection cylinder is normally cooled instead of heated. Materials like PF, UF, MF, PUR and silicon can also, be moulded, if, there is provision for cooling the hopper and the screw. The heating is, only, by friction or heated by circulating fluid between the electrical heaters and the chamber. Many machines are, now, available which may be changed from Thermo-plastic to Thermo-set moulding and vice versa, by changing, the nozzle cap and screw. A plunger or low shear screw injects the material into the mould. The mould is heated for curing.

Advantages and disadvantages of injection moulding of thermoset

Advantages

1 Flash free components.

2 Thick and void free sections.

3 Very fast.

Disadvantages

1 Machines are very expensive.

2 Fillers can not be used.

3 Some resins do not cross link to the same degree.

Processing parameters

Material	Temperature°c			Cure time	Injection Pressure(MPa)
PF	65-90	85-120	165-195	15-90	85-240
UF	65-95	85-110	135-165	15-90	100-240
MF	85-105	95-120	145-185	15-90	100-240
DMC	30-60	40-80	140-160	20-90	25-85
ALKYD	90-110	100-120	160-180	20-90	-

Compression / transfer moulding defects

The defects, in both compression moulding and transfer moulding, are almost, same and given in as common.

Problem	Moulding condition	Moulding compound	Mould
1 Sticking to surface of mould	- Temperature too low - Cure time too short	Material too fresh (contain moisture)	- Mould stained - New mould not run-in with lubricant - Contaminated with previously run material
2 Sticking due to tightness in cavity	- Temperature too high - Pressure too high	- too stiff in flow	- Insufficient draft
3 Breakage on ejection	- Soft material - Temperature too low - Cure time is too short - Hard material - Mould temperature too high		- Insufficient draft - Uneven ejection - Undercuts
4 Ejector pins penetrating in the moulding	- Temperature too low- - Cure time is too short - Mould temperature too high	- Heat distortion	- Diameter of pins too small - pins not fully returned
5 Thick flash	- Temperature too high - Too much materials - Pressure too high - Press closure too slow	- Too stiff - Too fast curing	- Mould not shutting properly
6 Excessive flash tightness in cavity	- Too much material	- Too easy in flow	- Flash clearance too large

7 Scratches and scars	*	*	- Undercuts in mould - Rough surfaces - Draft angle too shallow
8 Short shot (moulding)	- Temperature too high - Insufficient charge weight - Pressure too low - Closing speed too fast - Fast curing	- Too stiff inflow	- Diameter of pins too small - pins not fully returned
9 Sink mark	- Insufficient charge weight - Pressure too low	- Too easy inflow	- Flash clearance too large
10 Porosity & Voids	- Insufficient charge weight - Air trapment - Closing speed too fast - Incorrect positions of charge	- Too easy flow	- Inadequate venting
11 Surface ripple	- Palletizing & preheating not proper	- Typical long fiber material	- Matt finish sometimes allows to disguise this defect.
12 Cracks	- Very rare		- Uneven ejection - Undercuts - Rough surface
13 Dull patches	- Insufficient pre heating		- Stained mould - Worn out chromium plating
14 Blisters	- temperature too low	- Too fresh - Too short curing - Poor pre heating - Temperature too high	- Inadequate venting
15 Knit lines	- Press close too slow - Preheat too high - mould temperature too high	- Too stiff inflow	- Insert or core pin too near to edge - Vent at end of flow
16 Poor gloss	- Temperature too low - Cure time short		- Mould stained - Unpolished surface
17 White marks	- high temperature - Pre heat too high - Burnt material		
18 Distorted parts	- Uneven mould temp. - Uneven pre heat - insufficient cure time	- flow too easy - shrinkage too high	- Uneven ejection - Variation in wall thickness high - Wrong gate position

Transfer Moulding defects

All the problems in the compression moulding, is also, applicable for Transfer moulding

It is, based on assumption, that, the mould in question has adequate runner and gating provision.

Incomplete filling (short shot), knit lines, burning blisters, can all result from excessive frictional heat development, when transferring the moulding compound through runner and gate will occur. A correct mould design is very important.

Mould heaters and thermocouples

Objectives: At the end of this lesson you shall be able to
- **brief the types of mould heaters**
- **state what is thermocouple and how it works.**

Injection moulding of plastics s a complex application that some times requires many kinds of heater.

General types of heater

Band heaters (Fig 1)

Coil and nozzle heaters (Fig 2)

Cartridge heates (Fig 3)

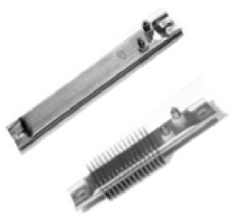
Strip heaters (Fig 4)

Fig 1

Fig 2

Fig 3

Fig 4



Thermocouple

A thermocouple is a device that is used to sense temperature in your process. It has two wires that are made of different metals they are jointed together at one end. In the type J thermocouple one wire (or leg) is made of iron, the other wire us made of constantan. You may hear of the referred to as "an Iron constantan thermocouple" or just type J.

There are several types of thermocouples used to sense temperature, but for our purpose, we will concentrate on the most common thermocouple type in the plastics market the Type J.

Type J thermocouples are used in extrusion, blow moulding and injection moulding. They are used because of the relatively low cost and acceptable accuracy in the 0-800°C temperature range.

When the thermocouple is connected to a temperature controller, it enables the temperature controller to measure the process temperature.

How does a thermocouple work

This is the short version of how a thermocouple works. As we know, a thermocouple is made up of two different metals, then joined at one end, this forms the hot junction, sometimes referred to as "the hot end".

When the temperature changes (increases for example) a small dc voltages is produced in proportion to the temperature because of the two dissimilar metals. If both wires were iron or constantan, there would not be a voltage produced, so you would not have a thermocouple.

Surface finish

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

- **state the importabts of mould polishing**
- **brief the different types appearance of mould finishing**
- **list the mechanical equipments and perishables used in mould polishing**
- **explain the finishing process of moulds**
- **enumerate the problems and solution in mould polishing**
- **explain the surface treatments of moulds.**

Mould polishing

Mould polishing is a very important step in producing a mould that works properly and a part that is functional and visually attractive. Proper finishing of the mould is important for the release of the part. Polishing insures that the part is not damaged during ejection. For example, scratches from rough surfaces or dents from ejector pin. It, also, prevents damage to the mould that could occur, if the mould closed on a part stuck in the mould. The finish of the moulding largely depends on the finish of the cavity and core.

Release

Release is the ability of the mould to eject the component. To accomplish this, all cutter marks must be removed from the moulding surface of the cavities and cores. Further, the finish should be vertical, from bottom of the cavity or core to the parting surface, to aid in the ease of ejection. This is called draw stone. This, also, will speed up the moulding cycle.

Appearance

There are different types of appearance required after finishing. They are:

a Rough finishing: There are mainly three types of rough finish. The first is called draw stone. This is the most basic of all finishes. It is strictly for release. The second is a sand-blast or glass-bead. This produces a matte finish, creating non-slippery, non-reflective surfaces. The third type is a textured finish. This is done to create a specific appearance, such as wood grain, leather or pebbly textures.

b Mirror finishing: A mirror finish is called diamond finish. There are three basic diamond finishes. The first, a rough diamond, is primarily used for release or as a preparation for a very fine texture. A diamond, the most often specified, is used to enhance the appearance of opaque parts and the ability to see through clear and translucent parts. A high shine or #1 diamond is necessary for optical or highly reflective parts.

c Texturing: To prepare a surface for texturing, you must know what the texture will be. This is necessary to prevent over or under polishing. Basically, the finer the texture, the finer the finish needs to be.

The product designer of a finished plastics product desires some sort of surface finish to the component. As a general rule, it is considered that all moulding surfaces of a mould must be providing a mirror polish. A mirror polished moulding surface helps the moulds in many ways as given below

- Easy flow of molten plastics material (melt) into the moulds
- Easy ejection of finished product from the mould
- Glazing attractive surface for a commercial product
- Good transparency for clear plastics materials
- High brilliance and reflect ability for optical and automobile lamp reflectors
- Rough surface of moulds would be liable to retain acids and leading to localized corrosion

Overview of the process

The polishing process involves a number of steps that must be performed in the proper order. This will ensure the best finish in the least amount of time. Remember, polishing is replacing one set of scratches with finer set of scratches, until, the desired finish is reached.

Polishing can be done by following method

- Lap first with rough grade lapping paste (emery paper) to remove machining marks
- After that medium and fine grade of lapping pastes (emery paper) are applied to give good finish
- Final polishing is done by chrome powder
- Diamond pastes of different grades are also used
- Amount of material removal is only a few hundredth
- Care must be taken to remove all machining marks initial stage itself to save time.

Standard specification of finish (Table 1)

Currently, there are three standards being used to define a finish. They are the "OLD" SPI which has six finishes, the "NEW" SPI which has 12 finishes and RMS values which is the measurement of the depth of the scratches.

Table 1

Old SPI	New SPI	RMS	Description
#1	A - 1	3 micron	Grade #3 Diamond buff
-	A - 2	6 micron	Grade #6 Diamond buff
#2	A - 3	15 micron	Grade #15 Diamond buff
	B - 1	20 micron	600 Grit emery
	B - 2	30 micron	400 Grit emery
	B - 3	40 micron	320 Grit emery
#3	C - 1	25 micron	600 Draw stone
	C - 2	35 micron	400 Draw stone
#4	C - 3	45 micron	320 draw stone
	D - 1	*	Dry Blast Glass Bead # 11
#5	D - 2	*	Dry Blast # 240 Aluminum Oxide
#6	D - 3	*	Dry Blast #24 Aluminum Oxide

The mirror polish is achieved by adopting polishing process, followed by a proper procedure. The very first step is to remove the cutter marks left over by the machining operation over the moulding surfaces. This is done by using various types of hand tools like files, needle files and raffle files. After obtained surface finish corresponding to the finer grade of files, abrasive stones of various grades are used, which again followed by various grades of emery papers. It is a laborious and time consuming process which determines the final, required final dimensions of, mould cavity and core. Therefore, the polishing of moulds is to be carried out under the close supervision of skilled mould makers. The various sequences of polishing process are illustrated in the following figures.

To avoid manual strain and to speed up the polishing process, there are polishing machines available with lot of accessories packed in a polishing kit which is, also, explained with figures

Mechanical Equipment in Mould Polishing

Mechanical equipment (hand held) is important to the mould polisher for three reasons.

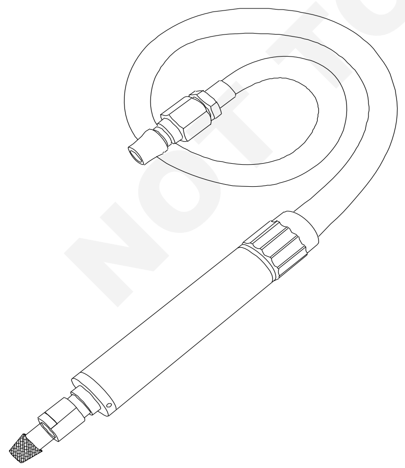
1 Mechanical equipment greatly reduces the amount of time required to reach the desired finish.

2 Mechanical equipment provides flexibility (many solutions to one problem) and versatility (ability to do more tasks) for the mould polisher.

3 Using hand held mechanical equipment enables the polisher to be more consistent in finish and in the time required from piece to piece and from job to job.

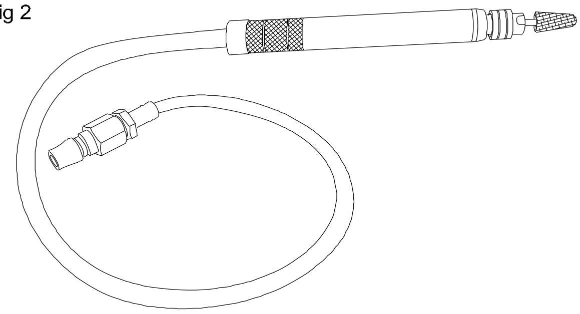
A Heavy Grinder (Die-Grinder): Die-grinders have a quarter inch collets. They are used to drive large mounted points, carbide burrs, abrasive discs, flap wheels, large brushes, and felt bobs. Die grinders are used to install large radii, remove deep cutter marks, and to diamond polish large surfaces. Heavy grinders, called die grinders, are available in air drive and electric with flexible shafts. (Fig 1)

Fig 1



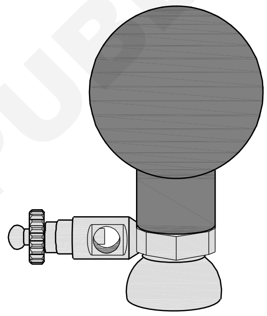
B Pencil Grinders (Light duty): Pencil grinders have interchangeable 1/8" and 3/32" collets. They are used to drive small mounted points, carbide burrs, abrasive discs, and diamond brushes and felt bobs. Pencil grinders are used to install small radii, removing deep cutter marks on small pieces or tight areas, and diamond finishing. Light duty grinders or pencil grinders are available in air drive and electric with flexible shafts. (Fig 2)

Fig 2



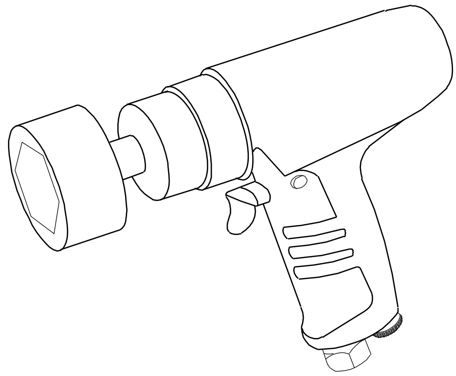
C Right Angled Head: Right angled head is available in large ¼" and small 1/8" collets. They are used with the same cutter and abrasives as die and pencil grinders. They are useful for getting into tight places like cavity side walls. They, also, afford the polisher more control when using abrasive discs. Right angled heads are available in air-drive and electric with flexible shafts. (Fig 3)

Fig 3



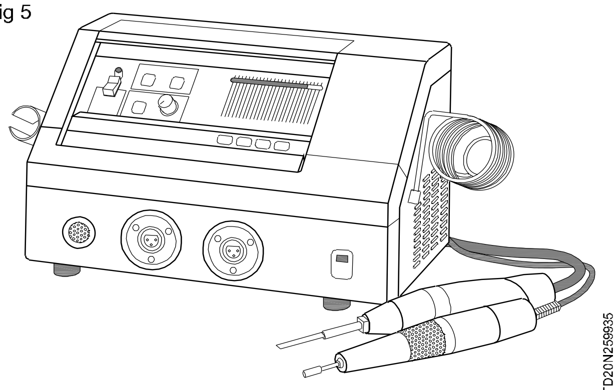
D 45 Degree Head: The 45 degree head is a variation of the right angle head. It is used with the same cutters and abrasives as the right-angled head. They are good for getting into tight spots and the bottom surfaces of deep cavities. 45 degree heads are available in air drive and electric with flexible shafts. (Fig 4)

Fig 4



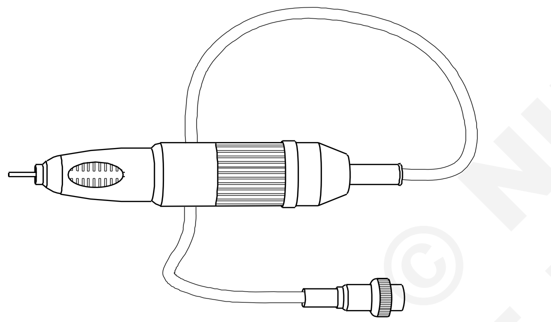
E Profiler: The profiler is a reciprocating hand piece. It reciprocates 2000 to 10000 times per minute. Stroke length is set on a cam inside the body. Profilers are the most frequently used piece of polishing equipment. They are used to push small files, polishing stones, wood and brass laps, and felt buffs. Profilers are available in air and electric with flexible shafts. (Fig 5)

Fig 5



F Ultrasonic Hand Piece: The true ultrasonic hand piece operates on a wave length. They are used with similar tools and abrasives as a profiler. Because, ultrasonic vibrates in a 4 to 30 micron range, their use is limited to small areas and intricate details. They are very efficient when used properly. Tool length is critical for proper operation. It is available in air and electric drive. It should be noted that the air drive is not a true ultrasonic tool. (Fig 6)

Fig 6

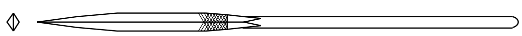


Mould Polishing Perishables

Perishables perform the actual cutting, smoothing, and shining of the mould. Mechanical equipment may make the perishables work faster, but, it is the perishables that are doing the polishing work. From the coarse files and discs through stones and emery, and finally, to the fine diamond compounds, all perishables work together to refine the moulding surface to conform to the customer's specifications.

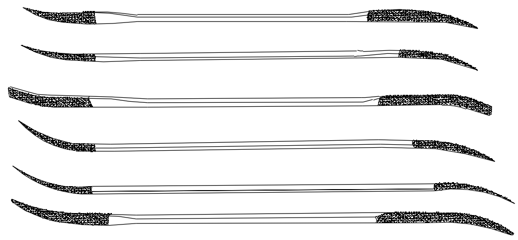
a Mill bastards: Large files, called mill bastards, are rarely used in polishing, but, they deserve a mention, because, on occasion, they are used during the benching process. Mill bastards came in four basic shapes. Flat, round, triangle and oval. Typically, 8 to 12 inches long, they have coarse teeth and can remove larger amount of steel. Because of this, caution should be used with these tools. They are used for flattening, initial smoothening or large radii and cleaning out corners. They are always pushed by hands. (Fig 7)

Fig 7



b Refflers: Small files, called refflers, are used much more frequently than larger files. Rifflers are used in the benching phase of polishing. They come in many shapes and configurations. (Fig 8)

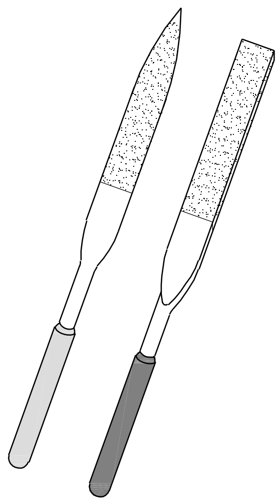
Fig 8



Typically, rifflers are six inch long with a small cutting surface on each end. They are used for initial smoothing in hard to reach and detail areas. Rifflers can be used by hand or with a profiler.

c Diamond files: The name diamond file refers to the cutting medium, not the shape. They are used in both benching and smoothening phases. Diamond files are 3" to 6" long with a cutting surface, if, 1" to 4". They come in four shapes flat, round, half-round and triangle. Diamond files are made by electroplating diamond particles to a file blank. They are excellent for use during an EDM process. Diamond files can also be used to install all radii. They can be used by hand removing cutter marks from thin ribs, especially, when the ribs have been burned, during EDM process. Diamond files can be used by hand, but, are most effective when driven by a profiler. (Fig 9)

Fig 9



d Grinding wheels: These are wheels mounted at the tip of a shank. Different shapes are available depends on the usage. These wheels are clamped on air grinders and can be used for removing material, during benching. Deep cutter marks can be removed by this process. (Fig 10)

e Flap wheels: Flap wheels are strips of emery with one end epoxy to an arbor. Flap wheels are 1" t0 4" in diameter. They are used with the die grinders. Because, they are resilient, flap wheels works well on flat and contoured surfaces. Flap wheels are not aggressive enough to remove cutter marks. They are used to take out disc and cartridge roll marks. Flap wheels work best, when only, light pressure is applied. (Fig 11)

Fig 10

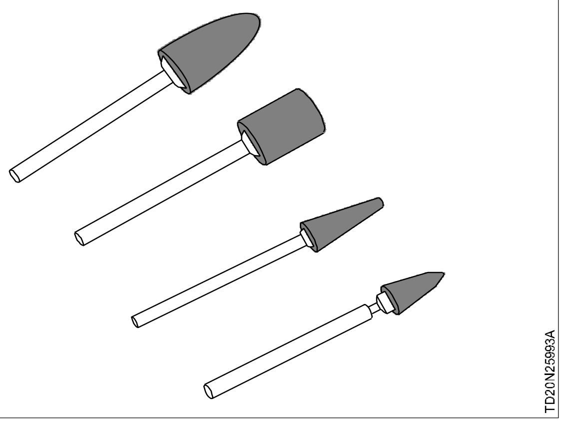
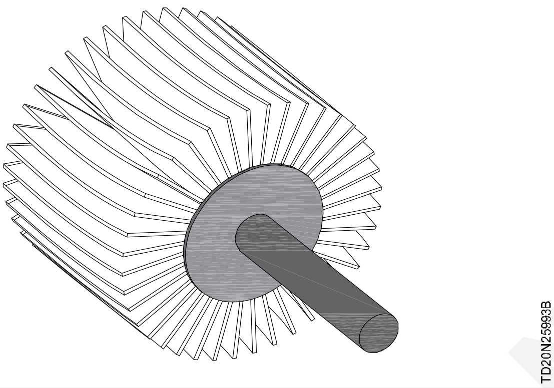
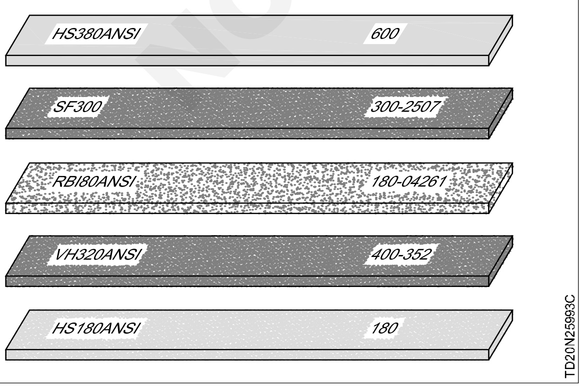


Fig 11



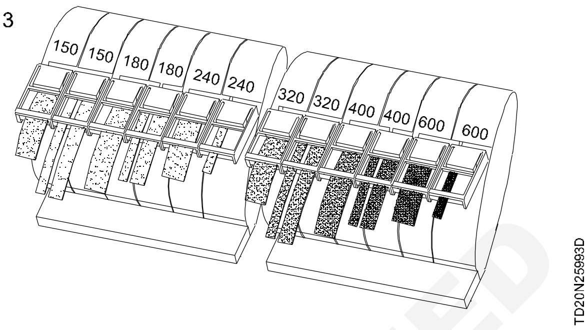
f Stones: Stones are manufactured "sticks". Aluminum oxide and silicon carbide are the abrasive particles used. They are used in the final finishing phase of polishing. Stones are square, rectangular or round. Sizes range from ¼" x 1/8" to 1 square inch, but, all are 6" long. Grits range from 120 very coarse to 1200 very fine. Coarse stones (120 - 400) are for removing cutter marks and benching marks. Finer stones are used to refine the finish and to prepare for diamond finishing. Stones are used with a profiler and by hand for a draw stone finish. Stones should be used with a lubricant such as mineral spirit to prevent "loading up." (Fig 12)

Fig 12



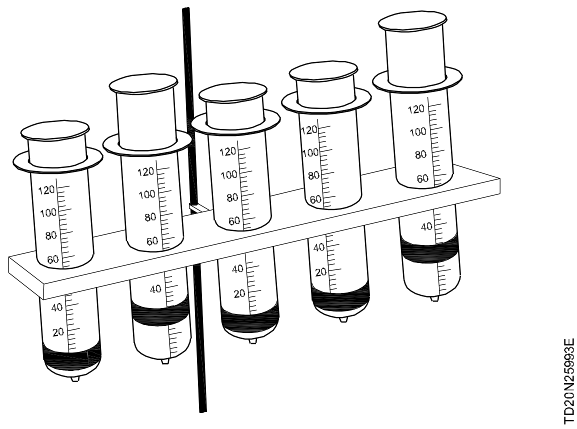
g Emery: Emery is a cloth with abrasive material glued to one side and is used in the final finishing phase. Emery comes 1" and 2" widths on rolls of varying lengths. Grits range from 60 to 600. Emery is pushed by hand or by wrapping it over a stick. When wrapped over a stick, a rubber pad should be glued to the contact surface. The emery should be used with lubricant to prevent loading and to produce a finer finish. Emery is used to smooth large surfaces and refine a stone finish prior to diamond finishing. (Fig 13)

Fig 13



h Diamond compound: Diamond compound is a mixture of diamond particles and a grease base. Diamond particles are graded for size by filtering through a mesh. They are used in the final finishing phase. It is available in grades from 90 - very coarse (80 - 100 microns), to 1/10 - the finest (0 - ¼ microns). The coarse grades are used for cutting and the finer ones are used for shining. This can be used with brass, wood, nylon and felt laps pushed by a profiler. Diamond compound is also, used with brushes and felt buffs driven by a die or pencil grinder. Diamond polishing removes stone and emery scratches and is the final step in producing a "mirror" finish. (Fig 14)

Fig 14



i Brushes: Brushes are made from animal hair, nylon and brass. Brushes are available from coarse to soft. They come in three shapes- cup, wheel and end. They are available in many sizes. Brushes are the most common way of driving diamond compound. Cup brushes are used to diamond corners and flat surfaces. Wheel brushes are used in inside diameters, side walls, parting lines, radii and round pins. End brushes are used on flat or lightly curved surfaces. Brushes are driven with die or pencil grinders, right angle and 45 degree angle heads. Light pressure and slow speeds of 2000 to 3000 RPM are recommended.

j **Buffs:** Buffs are available in many different sizes and shapes. Buffs are used to eliminate brush marks and to increase the shine. Buffs come in hard, medium and soft. The harder the buff, the more cut. The softer the buff, the more shine. Buffs are driven with die and pencil grinders, right angle and 45 degree angle head. Light pressure and a speed of 4000 to 6000 RPM are best.

Finishing process

Following, the basics is critical to achieve the best possible finish in the shortest possible time. By completing each step in the process, before going on to the next, a mould polisher will avoid doing double work and over-polishing. Understanding and following the proper finishing process will save time, money and avoids removing too much stock. The experienced mould polisher will follow the finishing sequence shown below to achieve the desired finish in the most efficient manner.

a Bench work

b Stone Finishing

c Rough Diamond Finishing

d Fine Diamond Finishing

e Buffing

a Bench work: Benching is the process of preparing the work piece for actual polishing. A bench work must be complete before you begin to polish. Benching can be as little as installing a radius or as involved as disking, grinding and blending converging surfaces and then smoothing with flap wheels or coarse emery. When extensive benching is required, deep cutter marks and stepped surfaces should be worked first, then, the radii should be installed or refined. A polisher will always work from details to open areas. Start benching with coarse perishables, progressing to finer.

b Stone Finishing: Stone finishing, also called stoning, refines all surfaces of the work piece. When each phase of stoning is completed, all surfaces should have the same uniform surface. Stone finishing is done with a profiler. To get the best possible finish in the least amount of time, stoning should always be done in the following order, starting with the deepest area and working out.

1 Stone all corners and detail such as radii

2 Stone all horizontal surfaces

3 Stone all vertical surfaces

All surfaces must be done before changing to a finer grit.

c Rough Diamond Finishing: Rough diamond removes the stone marks and begins the shining process. All brushes and laps used for diamonding should be dedicated to each grade of diamond compound. To begin rough diamonding, all surfaces should have at least # 600 stone finishes. A # 400 to 600 emery finish is the best. Diamond steps are the same as stoning. A mould polisher will diamond the deepest detail first and work out to larger surfaces. The largest possible brush is used. In all diamonding, the key is to use a minimum of pressure and a brush speed of 2000 to 3000 RPM.

d Fine Diamond Finishing: It is done in one to three steps. The number is determined by the final finish required. Fine diamonding techniques are the same as for rough diamonding. Start the fine diamond finishing with #6 grade diamond compound. If required, the second step is done with #3 grade diamond compound. For an optical quality finish, a third step is done using #1 grade diamond. Every time the grade of diamond compound is changed, it is critical that all surfaces are free from previous compound.

e Buffing: Buffing is the final step. It is done both mechanically and by hand. Buffing begins when you have completed your last diamonding step. When buffing, a mould polisher will start with the same grade of diamond as the last brushing steps. Techniques are same as diamonding -working from details to large surfaces using the large buff possible. Light pressure and a sped of 4000 to 6000 RPM will produce the best results. When mechanical buffing is complete, hand rubbing with soft felt or a cotton ball will remove any swirl marks left by the buffing process.

Chrome Plating

It is another method to for finishing the metal moulds other than polishing. The thickness of the plating should be grater than 0.025mm. The followings are the advantages of chrome plating,

1 Chrome plating has a low co-efficient of friction and thus facilitates

Problems in mould polishing and solutions

Problems	Cause	Solution
Orange peel	Too much pressure or speed during the diamond finishing steps	Re-stone until orange peel is removed and re-diamonded.
Pitting	Defective steel. Too much pressure or speed when diamonding. Did not remove all EDM marks when stoning. If pits have a comet-like tail then too much pressure was applied. If no tail exits, then steel may be defective or marks were not completely removed.	If from diamonding or defective steel, re-stone and diamond carefully. If from EDM, re-stone and diamond.

Dull shine	Contamination from stone grinder or diamond from previous operation left on surface. Contaminated brushes or buffs.	Clean all surfaces and re-diamond using new brushes and buffs.
Estimating surface condition	Underestimating or overestimating the surface condition will slow the polishing process	Decide beforehand how aggressive your initial abrasives should be. Adjust up or down to reach optimum speed.
EDM surface finish	Cutter marks left by an EDM, characterized by a pitted surface. This process leaves a hard scale on the surface	When ever possible, start an abrasive with loosely packed particles such as diamond files or discs. When approximately 50 to 75% of EDM is removed, switched to stones.
Lack of cleanliness	Improper care and storage of abrasives. Dirty work area. Not thoroughly cleaning surfaces when changing to a finer abrasive. Slow down job and produces in consistent finishes.	Store each grade of stone in its own container. Decide all diamonding brushes and buffs to only one grade of diamond. Store in covered containers. Keep work area clean. Never set diamonding supplies on bench. Meticulously clean pieces between each operation.
Lack of methodology	Slow down job and produce in consistent finishes	Always follow proper procedure.

The best solution to a polishing problem is, not to create a problem in the first place. A good mould polisher will always follow proper techniques and procedures as outlined in this session. Some problems, such as EDM burns can not be avoided, but, if handled properly they become easier to remedy. Estimating surface condition is the hardest problem to overcome. It is very difficult to estimate the depth of a cutter marks. Only, with experience can reasonable guesses be made.

Surface treatment method

Some plastic, notably poly-olefins and acetals, are high resistant to bonding other media to themselves. To overcome this, deficiency treatments like flame treatment and electronic treatment such as corona discharge, plasma discharge and chemical treatments are commonly, used.

Flame treatment consists of passing the object through an oxidizing gas flame. The flame causes an oxidation on the plastic surface and enables the adhesion of inks, enamels etc.

In corona discharge process, high voltage current is used for the oxidation of the surface. In plasma process, air at low pressure through an electric discharge is used to change the physical, chemical properties of plastics. In chemical treatment, products are dipped in acids resulting in an etched surface which makes it receptive to paint.

Types of surface treatment

Different types of surface treatment of plastics are screen process printing, hot stamping, heat transfer decoration, in-mould decoration, two-colour moulding, electroplating and vacuum metalizing.

Screen process printing

The process of printing through a screen consists of forcing ink or paint through the interstices of stenciled screen of the required image with a squeeze. The screen material is often nylon mesh or other synthetic material. Metallic screens such as stainless steels are also, in common, use.

Hot stamping

Hot stamping by the roll leaf process is a method of decorating plastic materials. The leaf or foil consists of a thermoplastic colour coat applied to an acetate or cellophane carrier film. When a heated die is pressed against the foil carrier film, the colour coat is released and adheres to the product placed beneath it.

Heat transfer decoration

It consists of a paper carrier, a fusible release coat and a thermo plastic ink image. The ink becomes sticky when heated by a hot platen. When it is pressed against the product the ink bonds to the surface. Some of the fusible release coat carries over with the ink, providing the image with a glossy protective coating.

In - mould decoration

In this method, the decoration becomes an integral part of the product making it most durable and permanent. In compression moulding, the mould is loaded with the resin in the usual manner and closed. After a partial cure, it is opened, and the cycle is proceeded, after keeping the foil consisting of a printed cellulose sheet and closing it again. The foil is fused to the product, during the moulding cycle and becomes a part of the moulding. For thermo plastic mouldings, the foil or overlay is placed in the cavity prior to the injections. The overlay is printed in a film of 0.05 - 0.10 mm thick of the same polymer of that of the product. The ink used to imprint the film must be heat resistant. The gate location in this case is very important to avoid shifting of the overlay during injection. Similar methods can be adopted, in decorating blow-moulded products also.

Two-colour moulding

Two-colour moulding is an injection process, where, two colours are successively moulded to create their portion of the finished parts. The shell is moulded first and then, it is transferred to another cavity, where, the second colour is injected to complete the product.

Electro plating of plastics

Plating is done on many plastics to provide a hard wear and corrosion - resistant surface. Physical properties such as tensile strength, heat deflection temperature etc. are also, enhanced in plated plastics. The surface to be plated is to be properly conditioned and sensitized. The proper choice of resin for products to be plated is also, very important. Many parts like knobs, light reflectors, name plates etc. are made out of plastics and plated to give metallic appearance with light weight and corrosion resistance. Plating is done on many plastics including Phenolic, urea, and acetal. ABS, Polycarbonate, etc.

Vacuum metalizing of plastics

It is process, whereby, a bright thin film of metal is deposited on the surface of a plastic moulded product or film while being subjected to a high vacuum. The deposited metal may be gold, silver or most generally, aluminum. Deposition is the result of vaporization of small clips of the metal to be deposited attached to an electrically heated filament. When electrical energy is applied to the filament, the clips melt and coat the filament. Increased energy caused vaporization of this coating, and plating of the product takes place. The deposition is compiled at high vacuum, in order of 0.5 micron. The thickness of the resulting coating is about 1 micron.

Multi day light mould

Objectives: At the end of this lesson you shall be able to
- **state what is day light and multi day light mould**
- **brief the simple under feed mould**
- **enumerate the under feed mould with reverse referred spure**
- **explain the working system of floating runner & cavity plate and other standard design**
- **list and brief the non standard latch/locks used in multi day light mould.**

Introduction

A mould with more than one opening is known as Multi-daylight mould. It is, also, some times called as Multi-part mould or Multi-plate mould. That is, a mould with single opening (single daylight) is called as Two plate mould or Two part mould and with two openings (double daylight) as Three-plate or Three part respectively.

A stripper plate mould is an example for two daylight mould. It has two openings namely, one between the cavity and stripper plate and the other opening between the core and the stripper plate.

The main purposes of multi-daylight moulds are:

1 Centre injection of components in multi cavity mould.

2 To provide more cavities in limited space.

3 To avoid off-setting of bigger component.

4 To provide more than one pin gate for large components.

Under-feed moulds

An under-feed mould is one, in which, feeding done from the bottom of the cavity. In a single cavity, under-feeding can be easily achieved with the main sprue gate. But, in a Multi-cavity mould, it can only be achieved with under-feed runner system.

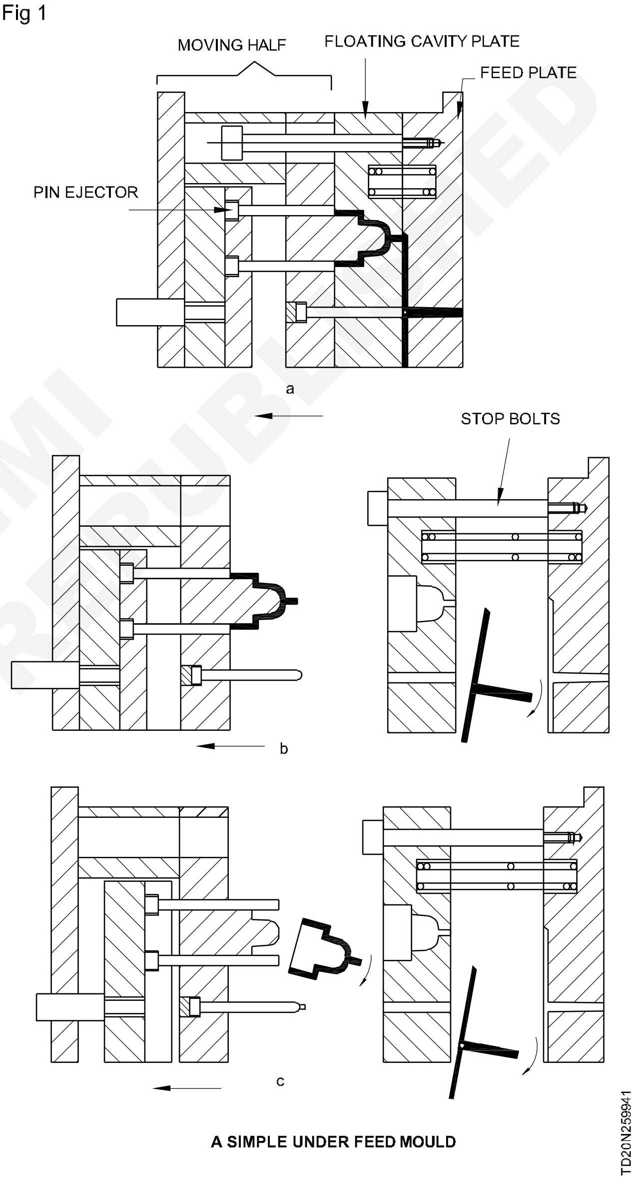
A simple under-feed mould

A simple under-feed mould contains a feed plate and the cavity is floating Fig 1a. The floating cavity plate may be solid or insert type. It may be guided on the main pillars, in which case, sufficient plate length should be taken Fig 1b. The feed plate contains the runners and the main sprue .The impressions are fed through a pin or sprue gate in the cavity plate. The gate becomes the part of the component and hence, has to be separated later. The runner should be made of a trapezoidal shape, as there is chance of sticking it on the cavity plate with round shape (Fig 1c). The disadvantage is that, the steel below the cavity can, only, be very limited.

Working principle

The working sequence of the operation is that, first, the mould is opened between the floating cavity plate and the feed plate by installing springs in between them. This makes sure that the sprue is pulled as the mould is opened. The sprue puller in the Core-half keeps the sprue butting on the cavity plate. The travel of the floating plate is stopped by the limit bolts attached to the fixed half. Further travel of the core-half moves the sprue puller pin out of the runner and the drops runner from the butting face of the floating plate. Once, opening between the core and cavity is enough for, dropping the parts the ejector system is actuated and the parts are ejected. When the mould is closed for the next shot, all the plates return to its closing position.

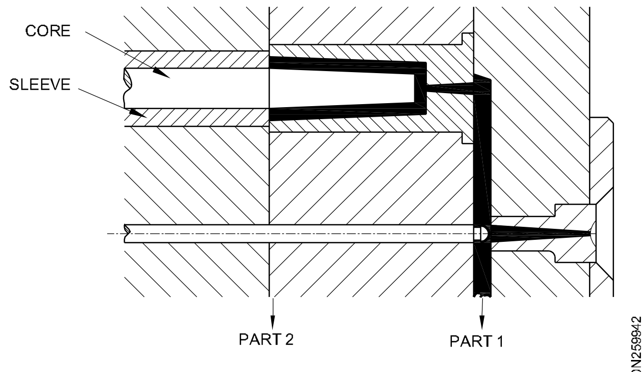
Fig 1



A SIMPLE UNDER FEED MOULD

Under-feed with reverse-tapered sprue (Fig 2)

Fig 2

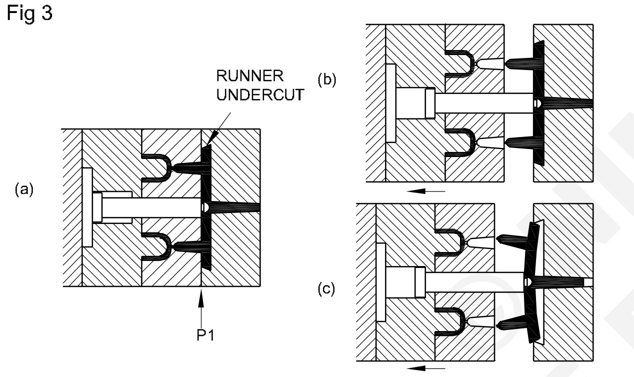


SECONDARY SPRUE GATE

This system enables pin gate to be used irrespective of thickness below the cavity bottom surface. The secondary sprue has a reverse-taper, till, the start of the pin gate. (Fig 3)

The disadvantage with this method is, that, feed system is not free to fall and hence, requires manual pick up.

Fig 3



DESIGN AND SEQUENCE OF UNDERCUT RUNNER SYSTEM,
DESIGN 1: (A) MOULD CLOSED;
(B) MOULD PARTLY OPENED, RUNNER RETAINED IN UNDERCUTS;
(C) FURTHER MOVEMENT AND THE SPRUE PULLER PULLS SPRUE FROM MOULD PLATE

An alternate design which incorporates delayed sprue-puller activation enables free fall of the feed system.

The following points must be born in mind, while, designing this system:

1 The gap between the floating plate and runner plate should be sufficient enough to allow the free fall of the runner system.

2 Consider the amount of deflection required, so that, the sprue fully comes out of the hole.

3 The diameter of the sprue puller must exceed the diameter of the width of the runner.

Floating runner plate (Fig 4)

This design ensures positive pulling runner and stripping of the runner system. This design requires additional elements and opening. So, we require a four- part mould (four-plate or triple daylight mould).

Working principle

In this, additional pullers are fixed on the fixed half. So, the runner system sticks on to the fixed half. A stripper plate is required to strip the runner from these secondary pullers. So, an additional plate is attached to the fixed plate which strips the runner from the puller. We call this plate as Runner stripper plate. In this mould, the runner is machined behind the cavity plate.

The first opening is between the cavity plate and the runner stripper plate. The runner is held by the sprue puller pins and the secondary sprue is separated from the component. As the mould opens further, the limit bolts pulls the stripper plate. The stripper plate movement is stopped by shoulder bolts. The runner system falls down. Then, the opening between the core and cavity begins. The cavity movement is stopped, when the gap is sufficient for part ejection. The part is, then, ejected. When the mould is closed again, for the next shot, all the plates return to its closed position.

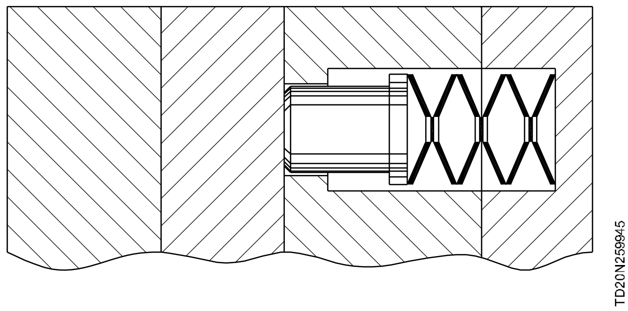
Working systems for floating cavity plate

We know that the opening of the multi-plate mould has to be in certain order. Now, how to lock certain floating plates and get controlled opening, only, between the required plates. Various methods are in use. We will see some of them.

Spring system (Fig 5)

The spring system in combination of stripper bolts gives a useful method of operating and controlling the movement of the floating plate in many applications. It is very simple to use, but, the main disadvantage is that, the floating plate may get stuck due to friction. This system is, mainly used in small and medium sized moulds for reasons of simplicity.

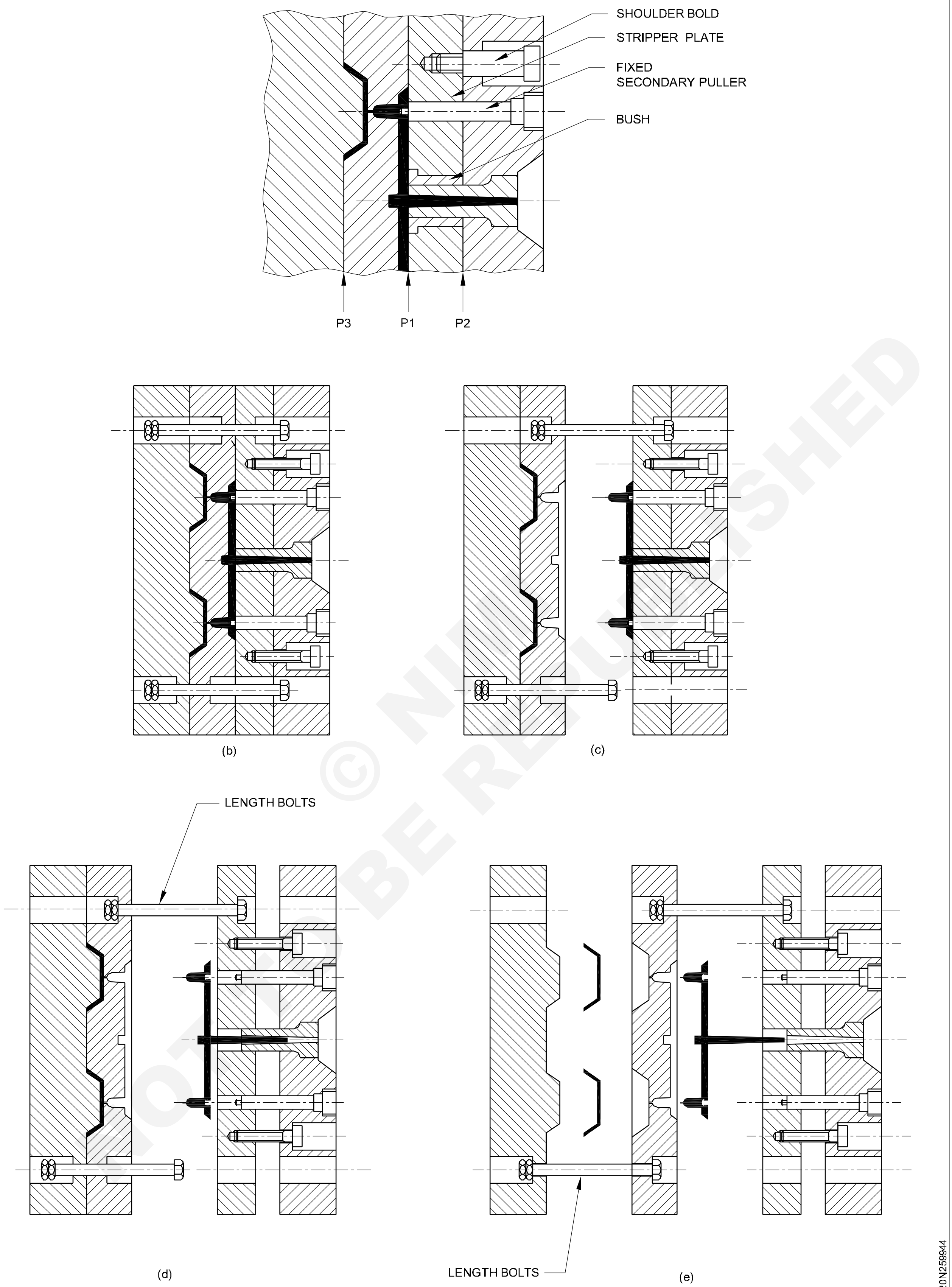
Fig 5



Ball catch system

This system consists of a number of spring loaded plungers mounted in the floating plate. These plungers are in contact with the grooved pillars which are fixed on the moving mould plate and extending into the hole of the floating plate, where, it is in contact with the plunger. The plunger is, always, in contact with the pillar in the functioning position. So, this frictional resistance holds the moving and the floating plate together, till, the floating plate has been pulled by the stripper bolts.

Fig 4


SHOULDER BOLD
STRIPPER PLATE
FIXED
SECONDARY PULLER
BUSH
P3
P1
P2
(b)
(c)
LENGTH BOLTS
(d)
LENGTH BOLTS
(e)
TD20N259944

Hasco puller system (Fig 6)

This system available from Hasco standards comprises of three parts,

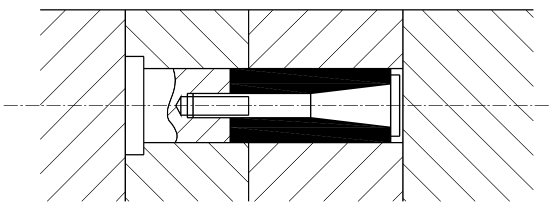
1 A shouldered insert with threaded hole

2 A Nylon sleeve.

3 A Taper screw.

The shouldered steel insert is fixed in the moving plate and the sleeve is attached to the insert with a taper shouldered screw. The sleeve is inside a hole, in the floating plate, as the mould is closed. The sleeve expands, as we tighten the screw. The sleeve diameter is adjusted, according to the frictional holding.

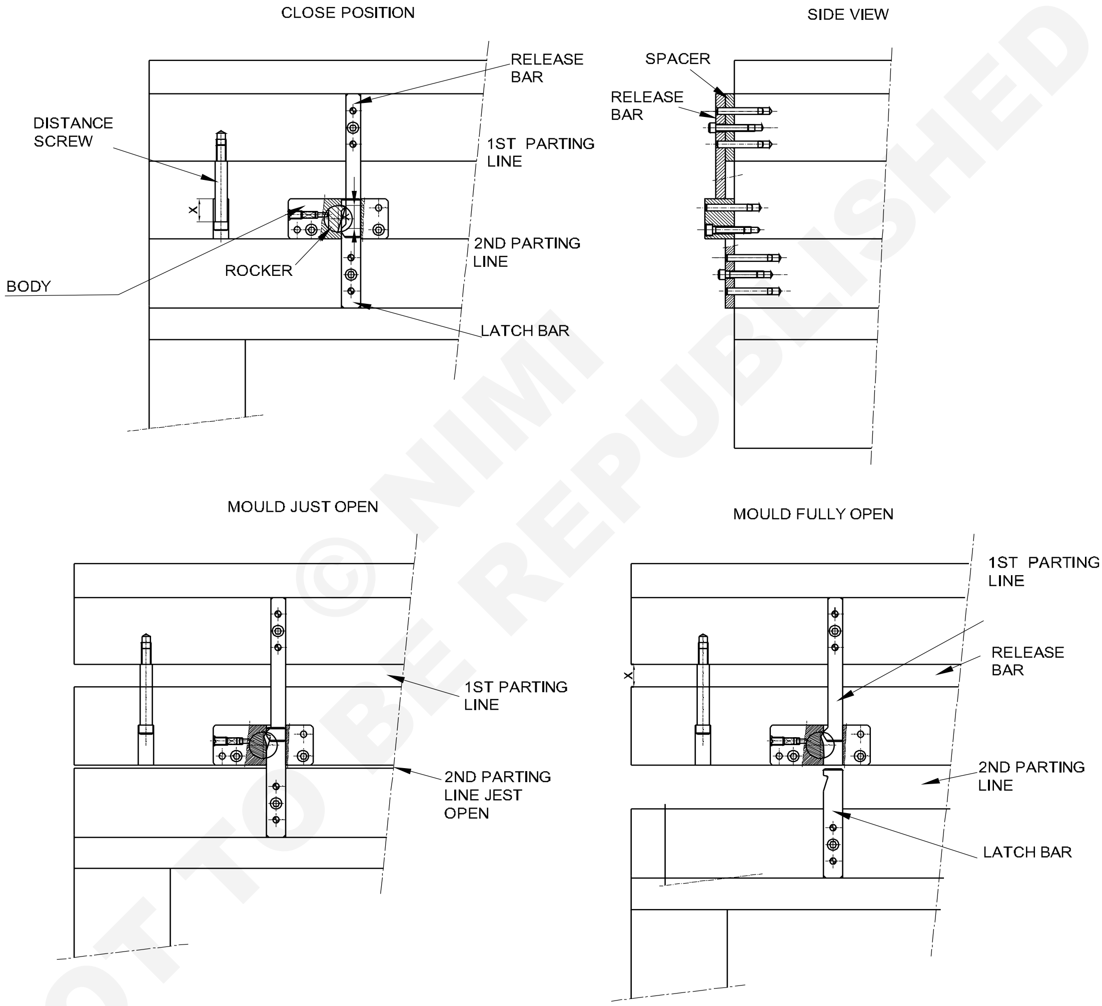
Fig 6



TD20N259946

Dme jiffy latch- lock system (Fig 7)

Fig 7



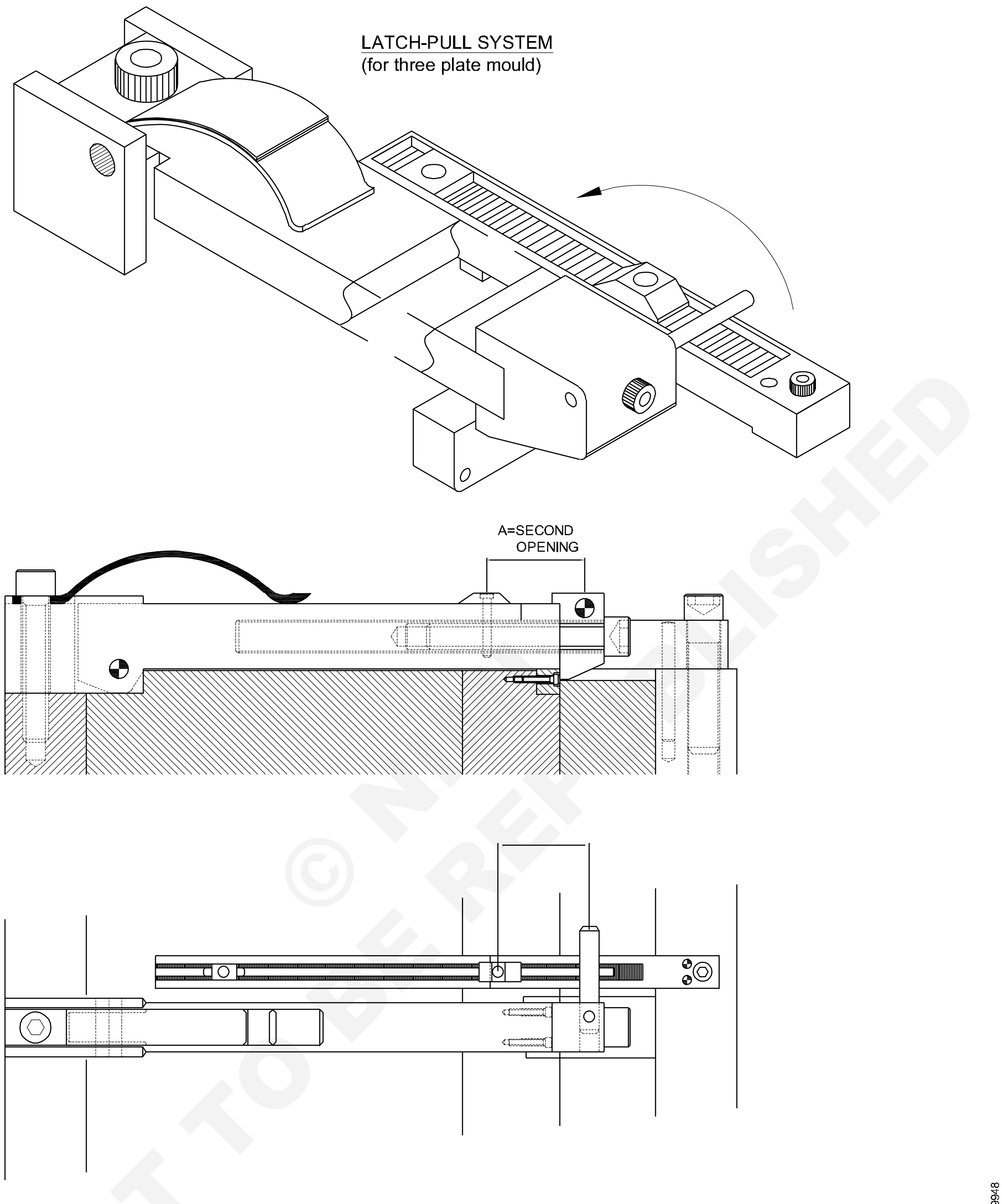
TD20N259947

For one mould, at least, two latch locks are required. These latches are mounted at outer surface of the mould. If, only two are used, they should be provided at the center of the mould. Body must be parallel screwed and doweled at the mould plate. The bars have to properly slide in to the body. Both latch lock must be accurately adjusted. Inaccuracies can lead in stripper plates and breaking of the bars. Latch bars and release bars must be preset, when the mould is closed. The mould is, then, opened and checked for motion sequence. This procedure is repeated, till, the latch lock works satisfactorily. Then, the latch bar and release bar are dowelled.

Alba latch pull system (Fig 8)

Alba latch pull system is very simple to use and it can be easily installed. These latches are mounted externally and the sliding cam plate provides easy stroke adjustment. The cam plate can be easily moved in a serrated swinging arm and clamped with an Allen screw. The cam plate is hardened up to 58 HRC. The hook is, also, a separate insert hardened up to 55 HRC, which also, incorporate a sliding pin of 6-12mm diameter. A leaf spring provides necessary return movement to the swinging arm. Since, the latch and the cam plate are separately attached to the mould, there is no possibility of over stroking the machine and causing damage.

Fig 8

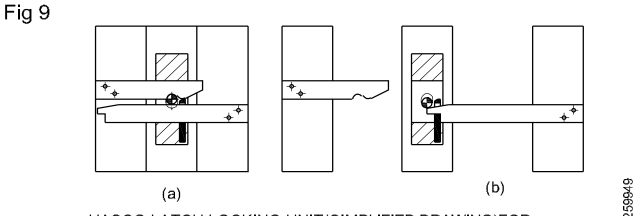


LATCH-PULL SYSTEM
(for three plate mould)
A=SECOND
OPENING
TD20N259948

Other standard designs

Hasco latch- lock system (Fig 9)

Fig 9



HASCO LATCH LOCKING UNIT(SIMPLIFIED DRAWING)FOR CONTROLLING FLOATING CAVITY PLATE ACTUATION.

Dms latch- lock system (Fig 10)

Italian hooks latch- lock system (Fig 11)

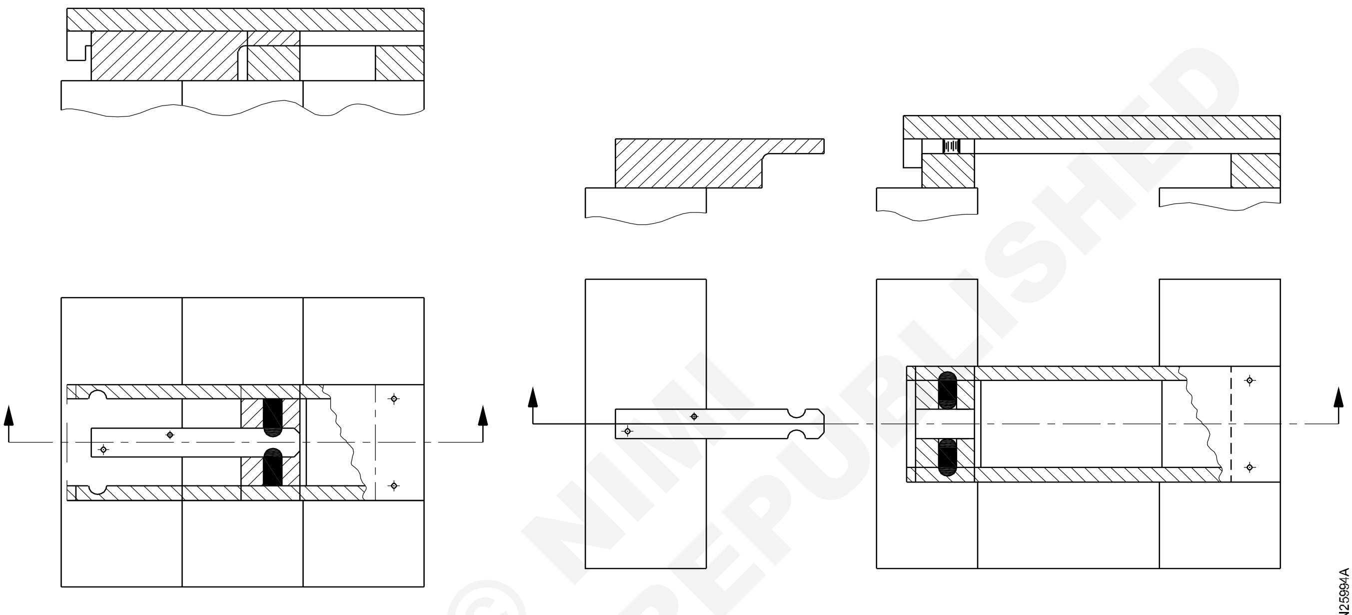
Non-standard latch locks (Fig 12)

Sample multi-daylight designs

Triple-daylight mould (four plate mould) (Fig 13)

Penta-light mould (six plate mould) (Fig 14)

Fig 10



LATCH LOCK SYSTEM USING UNDER FEED TYPE MOULD

Fig 11

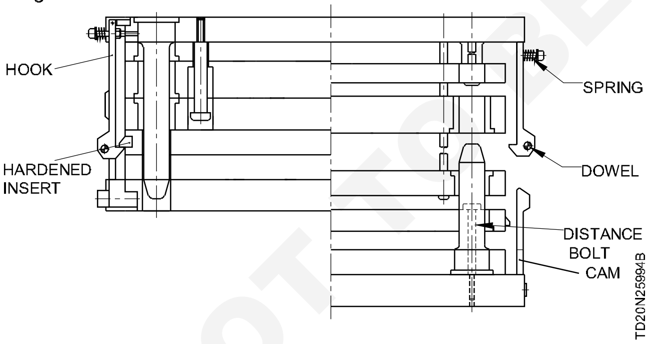
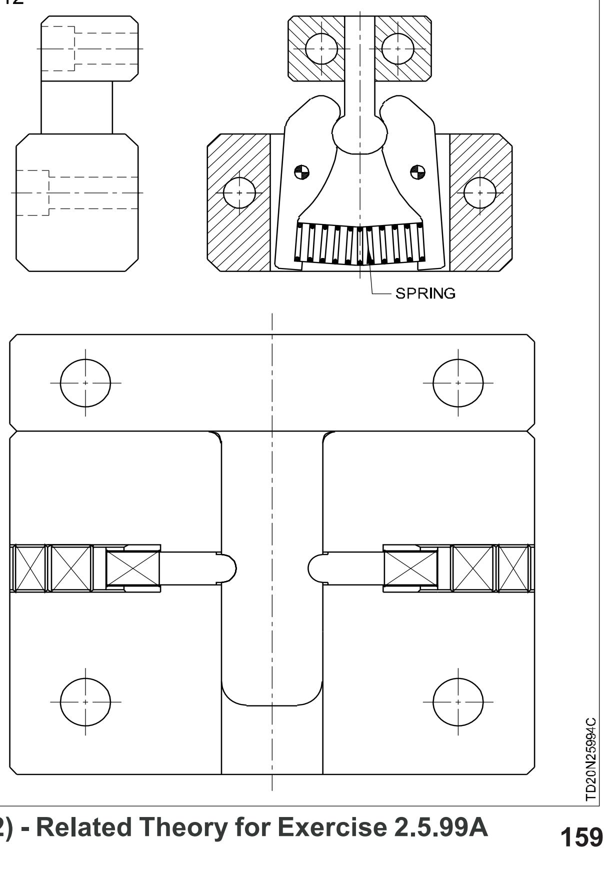


Fig 12



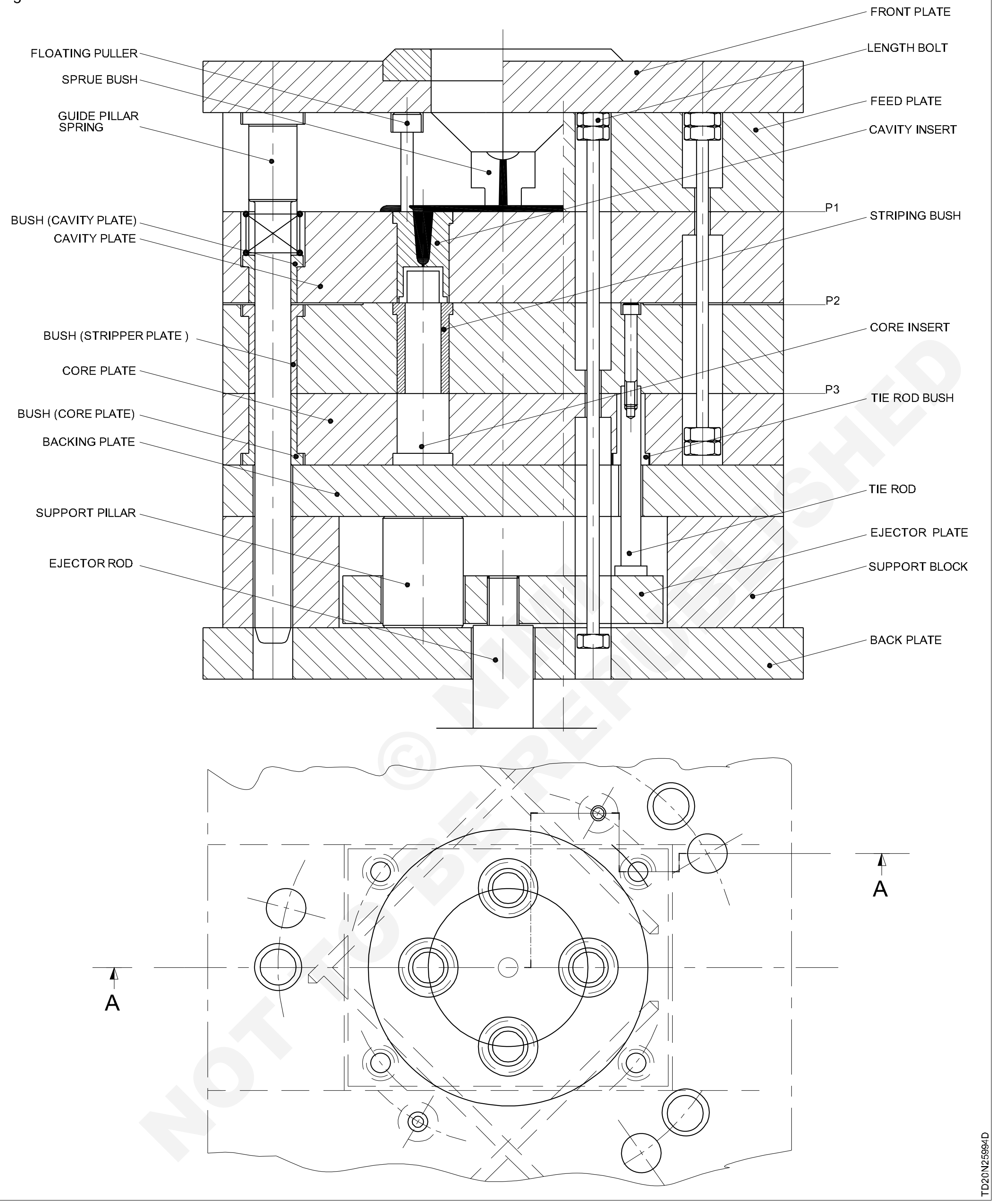
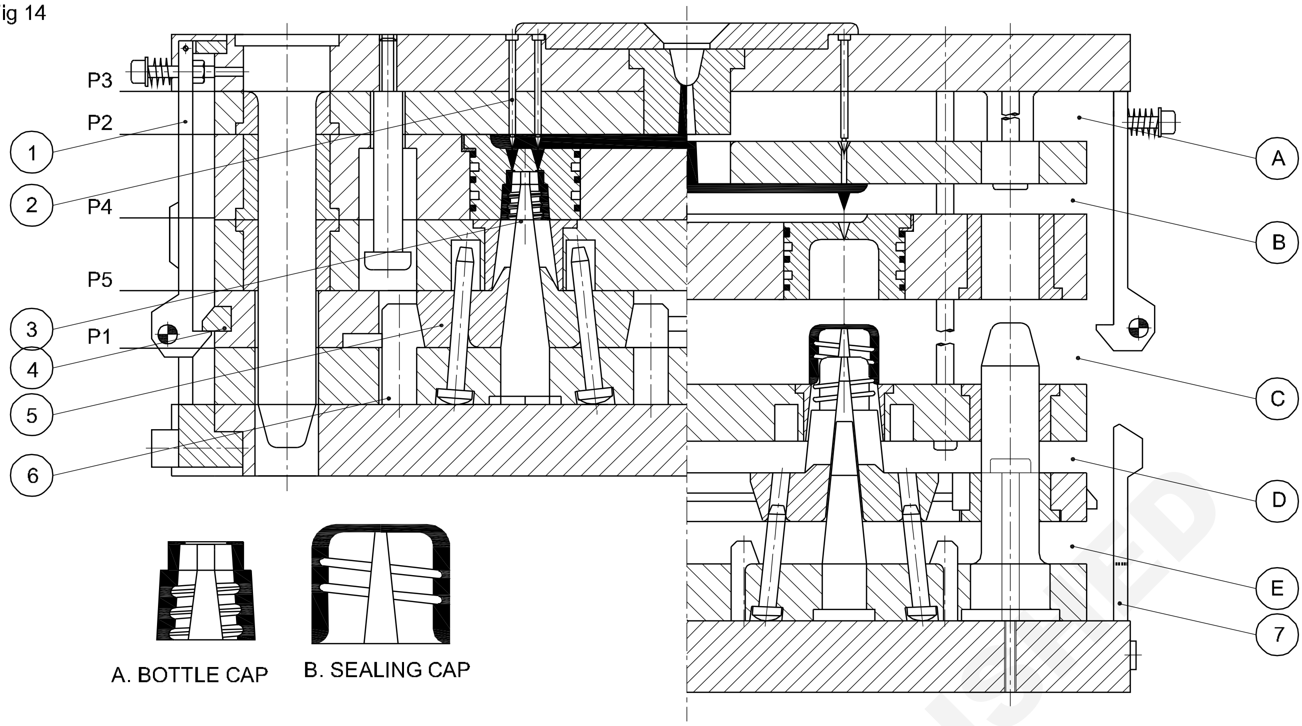

FRONT PLATE
LENGTH BOLT
FLOATING PULLER
SPRUE BUSH
FEED PLATE
GUIDE PILLAR
SPRING
CAVITY INSERT
P1
STRIPING BUSH
BUSH (CAVITY PLATE)
CAVITY PLATE
P2
CORE INSERT
BUSH (STRIPPER PLATE)
CORE PLATE
P3
TIE ROD BUSH
BUSH (CORE PLATE)
BACKING PLATE
TIE ROD
SUPPORT PILLAR
EJECTOR PLATE
SUPPORT BLOCK
EJECTOR ROD
BACK PLATE
A
A
TD20N25994D

Fig 13

Fig 14



1.MOULD OPERATING UNIT 2.ITALIAN HOOK 3.KEY
4.CENTRE CORE 5.SIDE CORE 6.WEDGE 7.CAM

TD20N25994E

Blow moulding

Objectives: At the end of this lesson you shall be able to
- **state what is blow moulding**
- **list the types of blow moulding**
- **explain the process of blow moulding**
- **state the advantages and disadvantages of blow moulding**
- **list and blow moulding fault and its remedy.**

Introduction

The capability to produce high volume of, small to medium sized parts in short cycles makes injection moulding, the most widely, used process for making plastic components. But, injection moulding has limitations, especially, when small volume of components, large components, continuous shapes or hollow parts required. A wide range of alternative processes are available, for producing plastic components. Some of the common methods are discussed below.

Blow mould

Blow moulding is used to produce hollow products. Bottles, fuel tanks, toys, oil-containers, chemical tanks, furniture etc are blow moulded.

In conventional blow moulding, parison or preform (tube, pipe etc) is first produced. This is then, placed inside a mould. Air is injected into the heated parison to blow it out against the wall of the mould cavity. The mould is then, cooled and the part is taken out.

Different processes are used, but, principally all are alike. The different blow moulding methods can be broadly classified into three:

1 Extrusion blow-moulding

2 Injection blow-moulding

3 Stretch blow-moulding

Extrusion blow moulding (Fig 1)

The extrusion blow moulding process uses an unsupported parison produced by an extruder. The wall thickness obtained is not uniform. The moulding process involves the following steps:

The melted plastic is extruded out of the die and forms a hollow tube called parison. The parison is dropped vertically down between the two halves of the open mould.

Fig 1

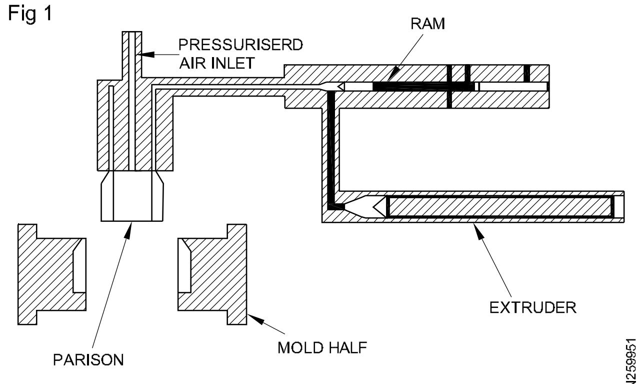


ILLUSTRATION OF THE INTERNAL WORKING OF AN EXTRUSION BLOW MOLDING MACHINE

The mould halves close around the parison and the bottom is pinched. Separate pinch bars also may be used.

Hot air is blown through the blow pin forcing the soft plastic against the cavity.

The mould is cooled while the pressure is maintained.

The mould is opened and the part is removed.

Various methods are used to introduce air onto the parison. It may be done through extrusion die mandrel, through a blow pin over which the parison is formed, through a blow head or through blowing needles that pierce the parison.

In the conventional injection moulding machine, the first stage, the plastic melt is injected around a core rod and a completely finished neck is formed.

In the second stage the preformed component is transferred to the blow mould via the core rod or the neck ring. Then, the compressed air is introduced into the preform through the core rod or seal ring to blow the part against the die wall. The part is, then, cooled and the air removed, before, ejecting the part. The part produced by this method will have, true neck which will suit cap and other bottle enclosures.

Stretch - blow moulding

Stretch moulding is a specialized form of blow moulding used to give certain structural properties to the material being moulded. The parison is stretched along its axis before blowing air. It can be done along with either extrusion or injection blow moulding process. In two axial orientations, bottles are first stretched longitudinally by an outside gripper or internally by a stretch rod. Secondly, they are stretched radially by blowing air inside to form it against the cavity wall. This process aligns molecules along two planes, giving extra strength and impact resistance to the component.

Big carbonated beverage bottles and water containers are made by this method. The stretch moulding process allows the container to be made with a crown bottom. The mould is made of three pieces instead of two halves. The third part is a movable bottom that moves away, before, the part is ejected. The top and bottom of the containers are not biaxial oriented, so, the properties are not same through out.

Stretch-blow mouldings are of two types; 1. In-line and 2. Double-stage

In-line stretch -blow moulding is done on a single machine, where, it passes through a conditioning station that brings it to the proper grain orientation. In double stage, first, the parison is formed, then, it is cooled. The parison is again heated in an oven to bring it to the proper orientation. At blown station, a centre rod is inserted into parison stretching it axially and properly position the parison in the center of the mould.

The in-line blow moulding system offers more flexibility from the material point of view, but, do not give the freedom of parison programming available in two stage systems. The advantages of two stage stretch moulding are that reduced scrap production, increased finish and higher out-put.

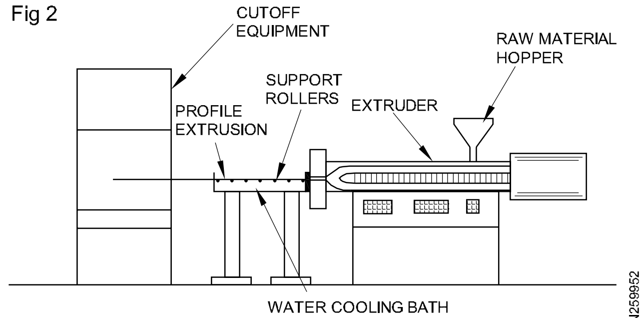
Extrusion moulding (Fig 2)

Extrusion moulding is the process in which a continuous work piece is produced by forcing the molten plastic through a shaped die opening. Sheets, films, tubes, rods, profiled shapes etc are manufactured by this method.

Thermos-plastic material is fed from a hopper into the heated barrel and a rotating screw take the material forward. A die is located at end of the machine as material progress along the barrel. Molten plastic, then forced through the die opening around a mandrel (core) to produce continuous hollow work piece.

The equipment includes an extrusion machine, a cooling tank, a sizing pipe and a puller. As, the molten plastic is forced through the die opening, it assumes the shape. Accurate size is maintained by passing the pipe through a properly sized fixture which cooled. When the plastic has cooled enough, it is pulled and cut into desired length.

Fig 2

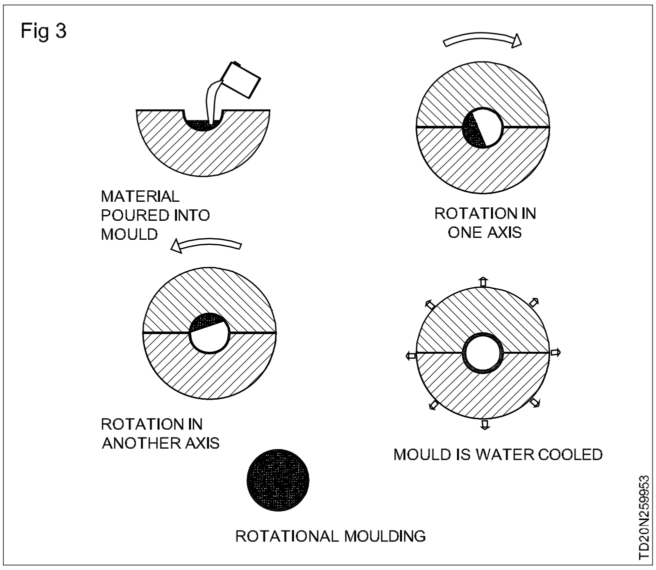


EXTRUSION MOULDING

Rotational moulding (Fig 3)

Rotational moulding is a specialized moulding technique used for producing hollow enclosures, like, a ball or open sided parts, or like, water tank. Recent developments in technology, have, made it possible to make a wide variety of shapes and sizes, both, in thermos plastic and thermos set materials.

In contrast, to other methods of processing plastics, rotational moulding is quite unique, because, the products are manufactured with in closed mould and without the use of pressure.



Manufacturing takes place in four stages

1 Filling

2 Heating

3 Cooling

4 Demoulding

Plastic material is placed in a completely closed cavity formed of two halves that is slowly rotated inside a heated air chamber. The rotation is two axes that are perpendicular to each other. When the mould is rotated in this fashion, the material flows uniformly around the cavity due to centrifugal force. The mould is, then, cooled for solidifying the part. For Thermo-set material, the heating would continue, until, curing is completed.

The advantages of rotational moulding are:

- Simple tool modification
- Wall thickness variability
- Uniform wall thickness
- No seams or welding
- No internal stress
- Fast realization time
- Multiple colour choice

-Low tooling cost

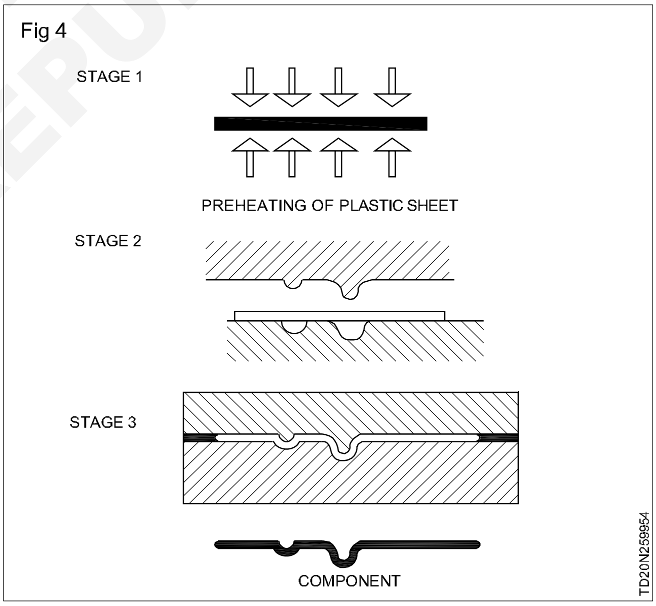
- Economic production of small batches
- Environmental friendly
- No production waste
- High flexibility

The rotational moulding has many uses in the automotive industry, because, components can be moulded using a very diverse range of plastics. The products include fuel tanks, roofs, bumpers, dash boards, air ducts and components to fit out vehicle cabs. There is hardly any field of industry which does not benefit from rotational moulding. Products range from simple water tanks, containers for special purpose, housing, insulation cladding and containers for drinks through to hi-tech components for aircraft.

Thermos-forming (Fig 4)

Thermos-forming is, almost same, as blow moulding. It does not have closed cavities and no parison is used.

Thermo forming processes make use of a plastic sheet which is kept over an open mould and heated. It is, then, forced in to the cavity by means of forming punch, air pressure or vacuum. The air is extracted through a number of small holes in cavity. Sometimes, it is pushed from top by a forming punch and also, pulled from bottom by vacuum. The part is cooled and then, ejected.



Faults and remedies for blow moulding

Objectives: At the end of this lesson you shall be able to
- **list the defects in blow moulding**
- **brief the causes of defects in blow moulding**
- **sugest the possible remedies to over come the defects.**

Defects	Causes	Remedies
1 Parison not being blown	Defective blow timer, clogged low lines, or too sharp pinchoffs	• Replace blow timer • Clean blow lines • Stone pinch-offs to create more pinch area • Reset cushion • Clean tooling
2 Parison curling	Mandrel and bushing not flush, bushing too cold, low container weight, or stock resin temperature too low	• Remachine tooling • Raise bushing temperature • Check heat controllers • Increase container weight • Increase stock resin temperature • Center mandrel in die • Check screw tip design
3 Drawdown, parison sag/stretch	Too high parison temperature, melt index of resin too high, or mold open time too high	• Decrease stock temperature • Increase extrusion pressure/rate • Decrease extrusion back pressure • Use lower melt index resin • Reduce mold open time • Decrease container weight
4 Parison tail sticking to parts	Parison too long	• Shorten parison length or increase pinch-off land area to cool compressed tail
5 Rough parison surface/uneven parison thickness	Extrusion speed too fast, cold parison, stock temperature too low, resin melt index too low, loose mandrel, or insufficient venting	• Adjust extrusion rate • Increase stock temperature • Use resin with higher melt index • Check mold alignment • Check tool design • Add venting by either sandblasting the mold surface channels or venting the interior of the mold
6 Black specks in containers	Resin hang-up in die, or material contamination	• Clean die surface and tooling • Check material for contamination
7 Bubbles/fish eyes	Blow air orifice too small or restricted, low stock temperature, mold temperature too low, tooling damage, or wet and contaminated resin	• Check the orifice for restrictions and size • Increase air pressure • Increase melt temperature • Increase mold temperature • Check tooling • Check resin for moisture • Check for contamination • Check for resin lines and/or streamers
8 Streaks	Stock temperature too high, contamination in die head, or degraded material on tooling	• Decrease stock resin temperature • Check heat controllers • Clean die head • Check for contamination in material • Clean tooling • Decrease extrusion back pressure • Decrease regrind level • Check design of flow path in die

9 Scratches and die lines	Stock temperature too low, die surface poorly polished, extrusion rate too slow, or damaged tooling	• Increase stock temperature • Clean die surface • Increase extrusion rate • Check tooling for damage • Check tooling for burnt materials • Check tooling for contamination
10 Orange peel	Parison temperature too low, sweat on mold surface, or melt index too low	• Increase melt temperature • Increase mold temperature • Check mold vent surface • Decrease cycle time • Use higher MFI resin
11 Containers stick in mold	Parison and mold temperature too high, blowing air pressure too low, part wall too thick	• Decrease stock temperature • Decrease mold temperature • Increase blowing air pressure • Check mold for damage • Center mandrel in die • Check for contamination in tooling
12 Parts blow-out	Blow-up ratio too large; mold separation, pinch-off too sharp or hot, or parts blow too fast	• Use large die • Increase clamp pressure or decrease blow pressure • Provide wider land in pinch-off • Cool mold pinch-off • Use low pressure blow followed by high pressure blow
13 Excessive shrinkage	Stock or mold temperature too high, cooling cycle too short, blowing air pressure too low, or uneven parison wall thickness	• Decrease stock temperature • Decrease mold temperature • Check mold cooling • Increase blowing pressure and delay air release • Align mandrel and die • Program parison
14 Poor weld or seal at pinch-off	Stock temperature too low, mold temperature too high, mold closing speed too fast, incorrect design of pinch-off blade, or improper mold venting	• Increase stock temperature • Decrease mold temperature • Increase mold closing time • Check pinch-off blade land size • Check pinch-off of mold for clearance and damage • Check mold alignment • Check mold venting
15 Excessive flash	Melt too hot, blowing air pressure too high, clamping mechanism out of adjustment, or excessive material being forced into mold	• Decrease melt temperature • Decrease extrusion back pressure • Decrease pre-blow air pressure • Decrease pre-blow time • Reset clamp or increase clamp pressure • Increase recess at pinch-off areas to accommodate more material
16 Warpage	Stock or mold temperature too high, blowing air pressure too low, material density too low, tooling condition, or part wall too thick	• Decrease stock temperature • Increase cycle time • Check mold for cooling • Reduce cycle time to obtain proper mold cooling • Increase blow air pressure • Use resin of proper density • Check tooling design • Center mandrel • Decrease container weight

Hydraulics

Objectives: At the end of this lesson you shall be able to
- **state what is hydraulic**
- **list out the applications of hydraulics system**
- **brief the advantages and disadvantages of hydraulics system**
- **define theory pascal's law**
- **explain the working principle of Brama's press.**

Principles of hydraulic system

Introduction

Hydraulics is the transmission and control of forces and motion by fluids.

Classification of hydraulics

1 Industrial hydraulics

The hydraulic systems used for industrial applications are known as industrial hydraulics or oil hydraulics. This is further classified into:

i Stationary hydraulics (machines)

ii Mobiles hydraulics (earth movers)

2 Water hydraulics

The hydraulic system concerned with civil engineering applications are known as water hydraulics.

Application of hydraulic system

Hydraulic systems and equipment find widespread applications in engineering, e.g.

- machine tool engineering
- press operations
- process equipment
- vehicles
- aircrafts
- shipbuilding

The advantages of hydraulics are

1 The ease of control

2 The generation and transmission of large forces and power through comparatively small units

3 Hydraulic cylinders and hydraulic motors permit starting from rest with maximum torque

4 Quick reversals are possible

5 They are self-lubricating and have a long service life.

Disadvantages

1 The high fluid pressures involve dangers of accident, so care must be taken that all connections are tight and do not leak.

2 Fluid friction and leakage of oil reduce efficiency.

Function of hydraulic fluid

Hydraulic Fluid : The fluid in a hydraulic system performs the following tasks.

- Transmission of hydraulic energy (Energy source: hydraulic pump Energy conversion: hydraulic cylinder or motor).
- Lubrication of all component parts (bearing, sliding surfaces etc.)
- Prevention of corrosion of internal parts.
- Removal of dirt, abrasive matter etc.
- Dissipation of heat.

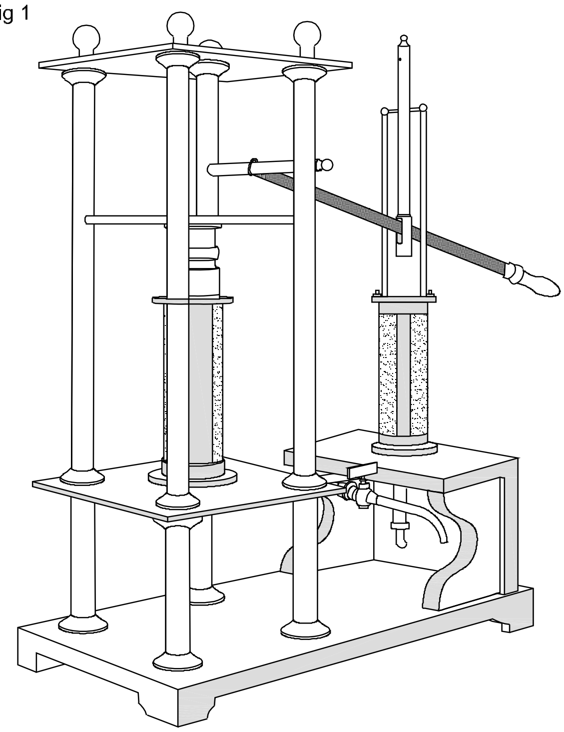
Theory of pascal's law (Pascal's principle)

It is the fundamental law in hydrostatics. It states that "a change in the pressure at any point of an enclosed incompressible fluid is conveyed undiminished to every part of the fluid and to the surfaces of its container". It is applied in hydraulic presses which are commonly used for forging, clinching, moulding, blanking, punching, deep drawing, and metal forming operations.

Brama's press

A hydraulic press is also known as Bramah press (Fig 1) after the inventor, Joseph Bramah, of England. It is a machine press using a hydraulic cylinder to generate a compressive force. It uses the hydraulic equivalent of the mechanical lever. As Bramah installed toilets, he studied the existing literature on the motion of fluids and put this knowledge into the development of the press.

Fig 1

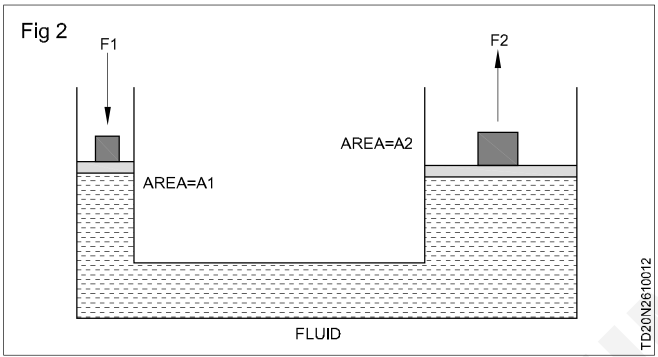


TD20N2610011

The hydraulic press depends on Pascal's principle-the pressure throughout a closed system is constant. One part of the system is a piston acting as a pump, with a modest mechanical force acting on a small cross-sectional area; the other part is a piston with a larger area which generates a correspondingly large mechanical force. Only small-diameter tubing (which more easily resists pressure) is needed if the pump is separated from the press cylinder. An important application of Pascal's law is the hydraulic press.

In Fig 2, A force F1 is applied to a small piston of area A1. The pressure is transmitted through a liquid to a larger piston of area A2. Since the pressure is the same on both sides, we see that P = F1/A1 = F2/A2. Therefore, the force F2 is larger than F1 by multiplying factor A2/A1. Hydraulic brakes, car lifts, hydraulic jacks, and forklifts all make use of this principle.

Fig 2



Pascal's rule is used to design water towers, but scuba divers must also understand this principle. At a depth of 10 meters under water, pressure is twice the atmospheric pressure at sea surface, and increases by about 100 kPa (about 1 atm) for each increase of 10 m depth.

Pressure and flow

Pressure: It is the result of a force applied over a specific area, and that pressure is therefore measured by the formula:

Pressure (P) = Force (F) / Area (A)

Pressure is denoted by the letter 'P'. Its unit is N/m^2.

Flow: It is part of fluid mechanics and deals with fluid dynamics. Fluids such as gases and liquids in motion are called fluid flow. It involves the motion of a fluid subjected to unbalanced forces. This motion continues as long as unbalanced forces are applied. A simple equation to represent this is:

Flow (f) = Quantity (Q) / Time (t)

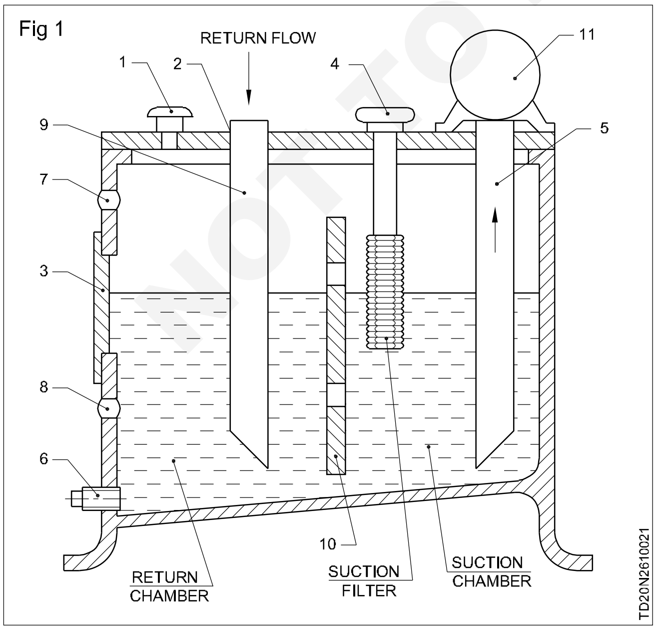
Flow is denoted by the letter 'F'. Its unit is

Elements of hydraulic system

Objectives: At the end of this lesson you shall be able to
- **list the elements used in hydraulic system**
- **brief hte functions of element used in hydraulic system and its related symbols.**

Reservoir : Each hydraulic system must have a reservoir (Fig 1), which serves the following purposes.

Fig 1



- Storage tank
- Separation of fluid and air
- Heat dissipation
- Support of pump and drive motor
- Base plate for mounting control equipment.

It consists of the following.

1 Air filter

2 Return line connection

3 Removable cover

4 Filter with dipstick and suction filter

5 Suction line

6 Fluid drain plug

7 Inspection glass(max. fluid level)

8 Inspection glass(min.fluid level)

9 Return line

10 Weir

11 Pump

Filter [4] : Contains a mesh strainer to retain any foreign matter during filling.

Drain plug [6] : Must be fitted at the lowest point. Oil can be fully drained by this.

Check of fluid level [7] & [8] : The fluid level is checked continuously by means of dipstick or inspection glass. The minimum and maximum levels should be marked.

Ventilation [1] : Reservoir must be provided with adequate ventilation. This ventilation is required to ensure that the atmospheric pressure can always act on the fluid surface so as to keep the pump primed and the oil free from air inclusions. The returning oil balances the level and drains at zero pressure.

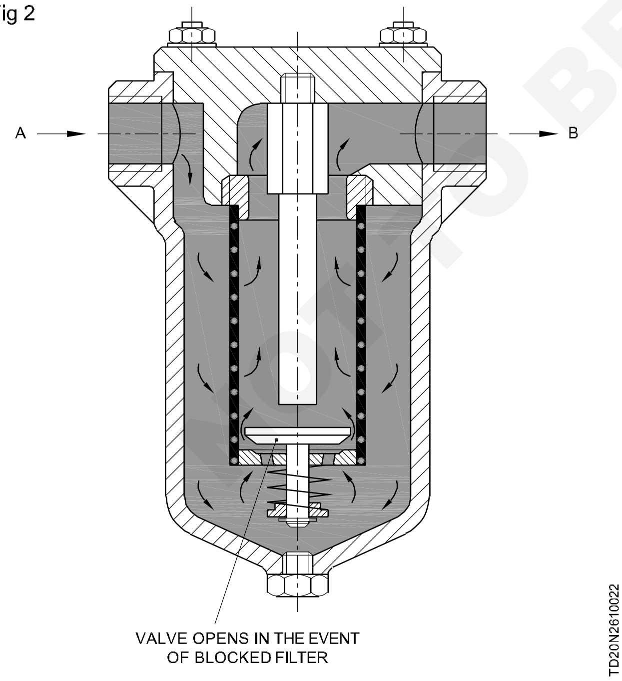
Weirs [10] : Partitions the reservoir into the suction chamber and return line chamber. In the latter chamber the fluid is allowed to settle and any foreign matter will be deposited.

Filter : Filtering of the fluid in hydraulic system is of major significance for their working and service life.Metal and seal particles, as well as dust and dirt in the air are prone to contaminate the fluid. This particles of various sizes must be continuously removed. Otherwise they will eventually block the ducts and ports in the system leading to extensive malfunctions.Contamination causes substantial wear of the moving parts in the hydraulic system.

Magnetic mesh filters provide adequate filtration by a strong magnetic "candle" and a close-mesh wire cloth filter cartridge.

The filter shown in Fig 2 is designed for fitting in the return line.

Fig 2



The distinction is drawn between the following filtration methods:

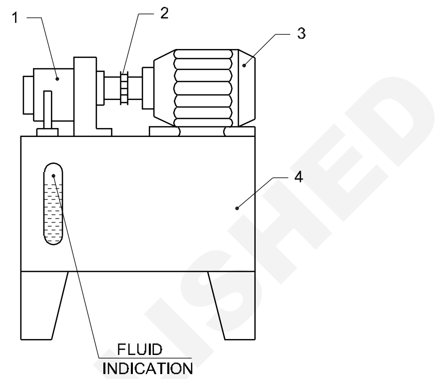
Suction-line filtration : The filter is installed in the suction line. It protects the pump against damage caused by foreign matter.

Pressure-line filtration : The filter is installed in the pressure line to protect hydraulic equipment (e.g. pilot controlled valves) against foreign matter(not commonly used).

Return-line filtration : The filter is installed in the return line (most commonly used method).

Power pack : The power pack consists of the following (Fig 3)

Fig 3



1 Hydraulic pump (gear pump)

2 Coupling

3 Electric motor

4 Reservoir

- Pressure relief valve(safety valve)
- Pipes and unions.

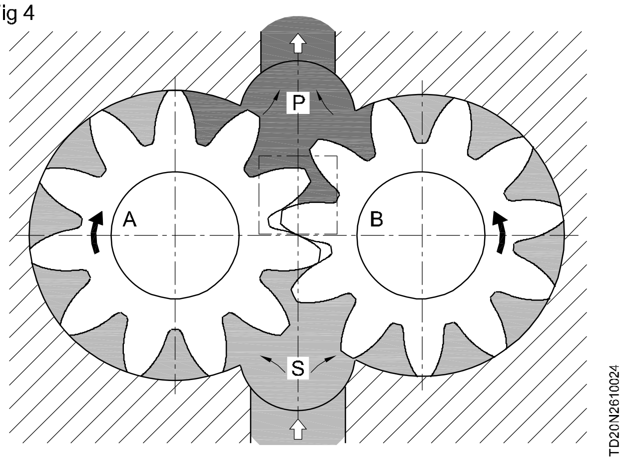
The gear pump [1] is connected via a couping [2] to the electric motor [3].

The gear pump [1] reservoir [4] and safety valve are interconnected by pipelines. The lines in the reservoir terminate below the fluid level so that no air can enter the lines.

Gear pump (Fig 4)

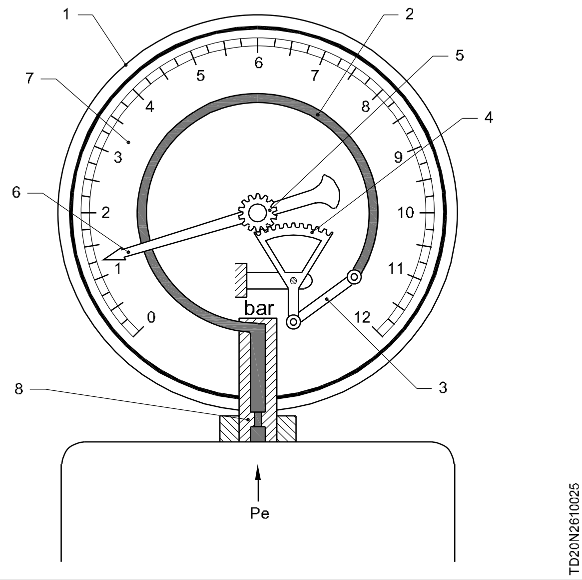
Function : The gear pump converts the mechanical energy supplied by the drive motor into hydraulic energy. It generates a certain fluid flow (delivery).

Fig 4



Pressure gauge (Fig 5) : Pressure gauges are used to indicate pressures.

Fig 5



1 Housing

2 Bourdon tube (spring tube)

3 Connecting link

4 Sector gear

5 Pinion

6 Indicator

7 Scale

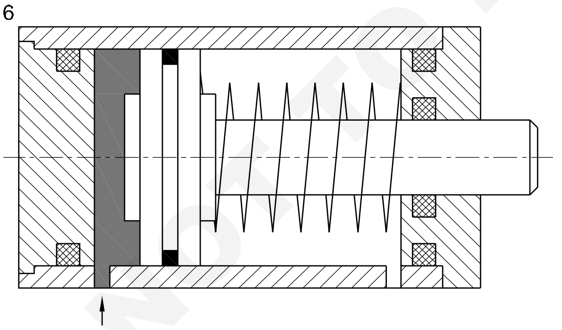
8 Inlet pipe connection with restriction

Cylinders

Single acting cylinder (Fig 6)

Function : The single-acting cylinder converts the pressurized fluid flow into a linear force and motion.

Fig 6



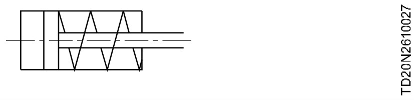
Operation : The fluid enters the cylinder housing on the piston side. Pressure builds up at the piston. The piston moves out.It returns after the directional control valve has been actuated.

The force for return motion is provided by a built-in compression spring (return spring).

Application: Lifting, clamping, lowering loading and unloading of workpieces and tools.

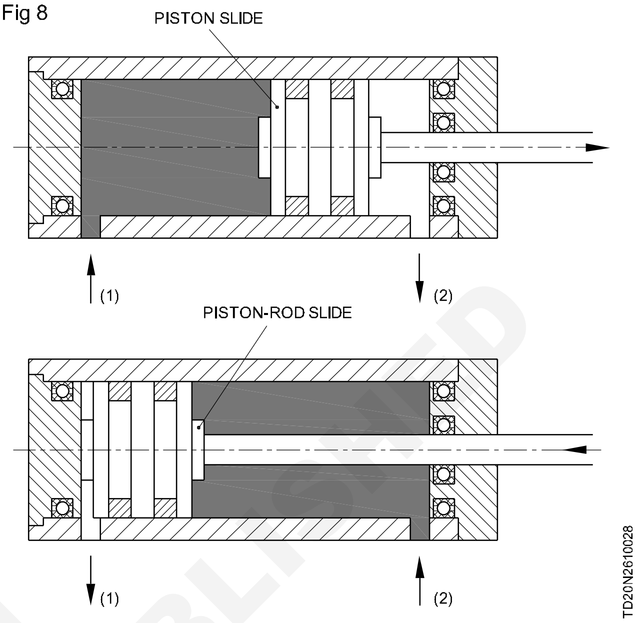
Symbol : Single - acting cylinder with spring return (Fig 7).

Fig 7



Double acting cylinder (Fig 8)

Fig 8



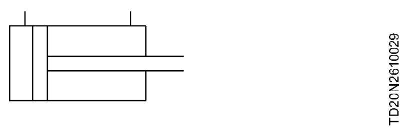
Function: The double-acting cylinder converts the pressurised volume flow into a linear force and motion. The pressurised fluid is used for effecting the forward and return motions.

Operation : During the working stroke the fluid enters the cylinder at(1)and acts on the piston side. Pressure builds up, causing the piston to move out. The fluid on the opposite side flows into reservoir via(2). During the return stroke the fluid enters the cylinder at (2), causing the piston to move in. The fluid on the opposite side flows into the reservoir via (1).

Application : To produce linear reciprocating motion. Used for feed slides in machine tools, presses etc. This is because the return stroke is also possible under load, as compared with the single acting cylinder.

Symbol : (to ISO 1219) Double-acting cylinder (Fig 9).

Fig 9



Types of valves used in hydraulic and pneumatic systems

Hydraulic valves

Hydraulic valves are components which control and regulate the flow, pressure and direction of hydraulic oil. Hydraulic valves may broadly be classified in to five types namely 1. Direction control valves 2. Pressure control valves 3. Flow control valves 4. Non-return valves 5. Shut-off valve.

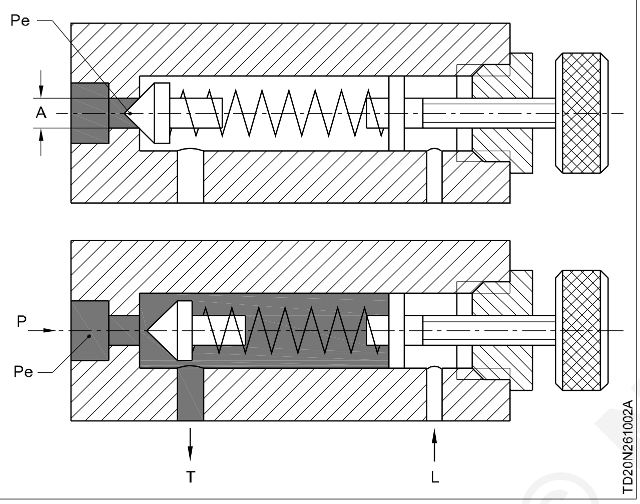
Directional control valves

Directional control valves are very essential in any hydraulic circuit in order to control the direction of flow of hydraulic oil and to achieve the desired motion. Directional valves are used mainly in piston constructions. The symbols show the connectors, the passages and the control positions. Valve specifications are deducted depending on the number of connections and control positions. The first number gives the connection and the second the control position. 3/2 directional value means: valve with three connections and two control positions.

Pressure relief valve : The pressure relief valve (Fig 10) is used for

- Limiting the working pressure to specific value.

Fig 10



This Protects the system against excessive pressures.

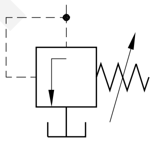
Operation : In the initial position compression spring forces the taper seat against the bore.

The spring force acting on the taper seat can be adjusted by means of the setscrew and compression spring. When fluid force rises above the set spring force (opening pressure), the taper seat is lifted, creating an annular gap through which the fluid returns to the reservoir.

Application : Each hydraulic system must be fitted with a pressure relief valve on the delivery line of the pump to prevent accident and damage caused by excessive pressure.

Symbol : Pressure relief valve (Fig 11).

Fig 11

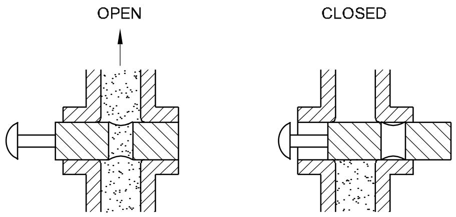


Symbols : Symbols are used to represent valves in circuit diagrams. They only indicate the function of the valves and not the various types.

These symbols have been standardized in accordance with ISO 1219.

The moving parts of the valves can assume different positions or spool positions (e.g. open close (two positions (Fig 12)).

Fig 12



Each position is represented by a square (Fig 13a).

Positions can be identified by means of letters. A valve with 3 positions (a - O - b) is shown, (Fig 13b). The centre position is designated “O”.

Within the squares, pipelines and flow directions are shown by lines and arrow heads respectively (Fig 13c).

A shut-off is indicated by a bar in the square (Fig 13d).

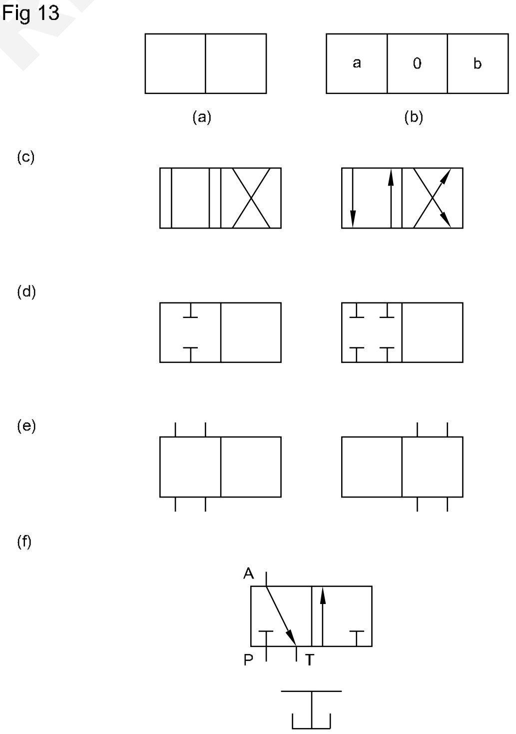
Connections are led to the square “neutral position” (Fig 13e).

The controlled connections can be identified by a capital letter.

Operating and cylinder supply lines	A ,B, C...
Inlet, pressure	P
Outlet	R, S, T

The outlet and return flow in the reservoir are identified by adding the reservoir symbol (Fig 13f).

Fig 13



Examples of controlled connections (main connections)

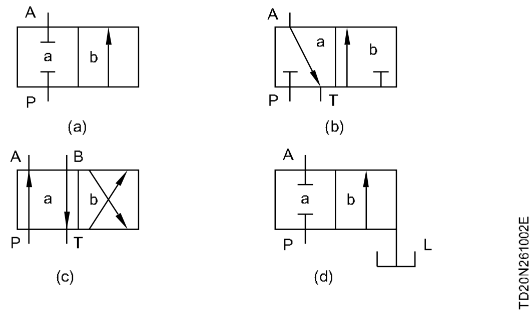
2 controlled connections -> two main connections (Fig 14a).

3 controlled connections->three main connections (Fig 14b).

4 controlled connections ->four main connections (Fig 14c).

Leakage fluid is removed by means of drain line (Fig 14d). For simplicity, it is no longer shown in the symbols and circuit diagrams.

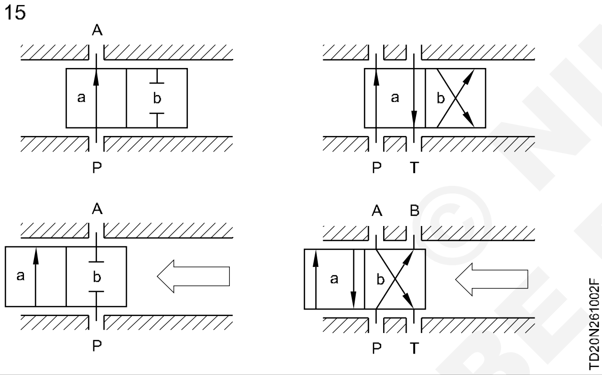
Fig 14



The drain and control line connections are not main connections.

Other positions are obtained by displacing the square blocks until the connections coincide with the lines of the other square (connections remain unchanged), (Fig 15).

Fig 15



Directional control valves

3/2-way directional control valve

Function : 3/2-way directional control valves are used to control the direction of fluid flow. The flow is open in one direction and simultaneously blocked in other.

Operation : The sliding spool of the 3/2 way directional control valve (3 connections, 2 positions) blocks inlet P in the neutral position (Fig 16) and opens the return A to T. When the lever is actuated, the outlet T is closed, P is connected to A (Fig 17).

Once the lever has been released, the compression spring returns the sliding spool to the neutral position (Fig 16); inlet P is blocked once more, while the return from A to T is opened.

Leakage oil is removed by drain lines.

Application : The 3/2-way directional control valve is used to control single acting cylinders.

Fig 16

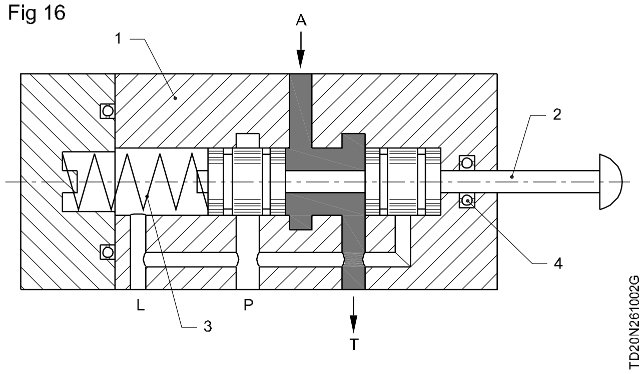
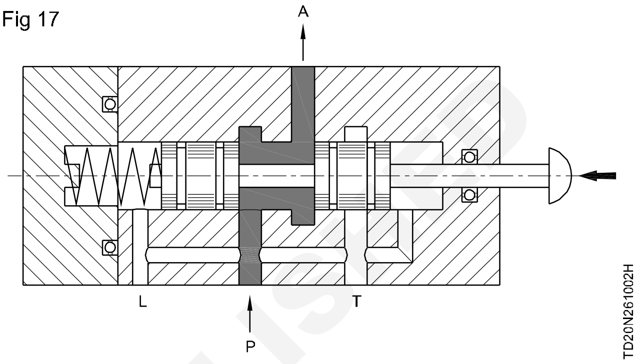
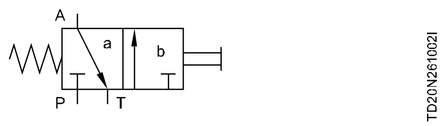


Fig 17



Symbol : 3/2-way directional control valve (Fig 18) in normally closed position (flow from P to A blocked).

Fig 18

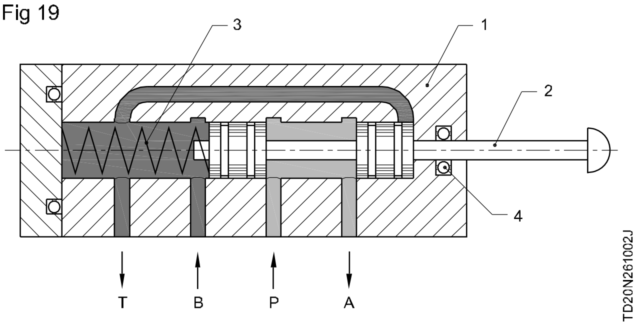


4/2-way directional control valve

Function : 4/2-way directional control valves are used to control the direction taken by the fluid flow with flow in both directions.

Operation : The sliding spool of the 4/2 way directional valve (4 connections, 2 positions) releases the flow from P to A and B to T in the neutral position. (Fig 19)

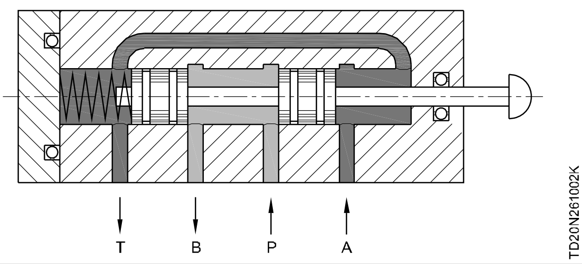
Fig 19



When the lever is actuated (Fig 20), the flow from P to B and A to T is reversed.

Once the lever has been released sliding spool returns to the neutral position. (Fig 19)

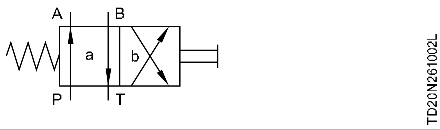
Fig 20



Application : The 4/2-way directional control valve is used to control double-acting cylinders.

Symbol : 4/2-way directional control valve (Fig 21).

Fig 21



Check valve

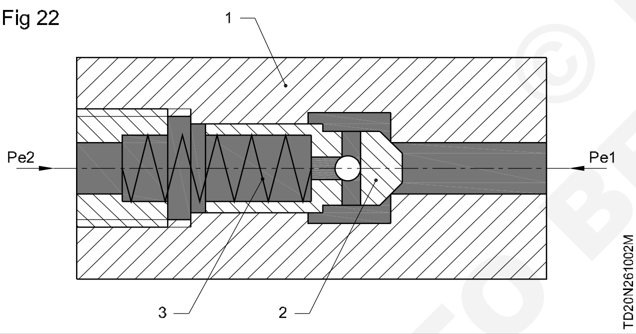
Function : The check valve is used to permit fluid flow in one direction, flow being blocked in the opposite direction.

Operation : Pressure Pe1 acts upon valve cone[2], causing it to lift off from its seat and permit flow.

Pressure Pe1 must overcome the slight force exerted by the compression sping[3].

If pressure Pe2 acts in the opposite direction, its adds to the spring force, causing the valve cone to be forced against its seat. The flow is blocked. (Fig 22)

Fig 22



If pressure Pe1 and Pe2 act together, flow occurs when Pe1 > Pe2 + spring force.

Application : A check valve is used in a hydraulic circuit to permit flow in one direction and block the flow in the other.

It is used to prevent pressure surges from the hydraulic system into the pump.

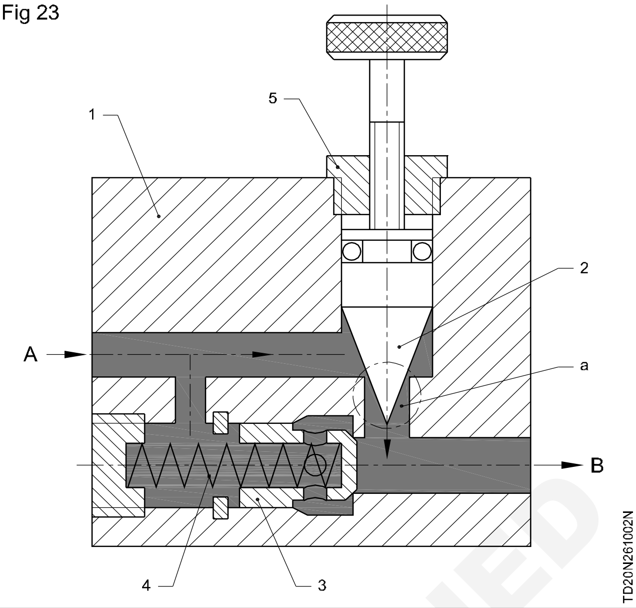
It prevents "emptying" of pipes and hoses

Variable return orifice check valve

Function : The variable return orifice check valve limits the fluid flow in one direction (restriction), ensuring the full flow cross-section in the opposite direction (check valve).

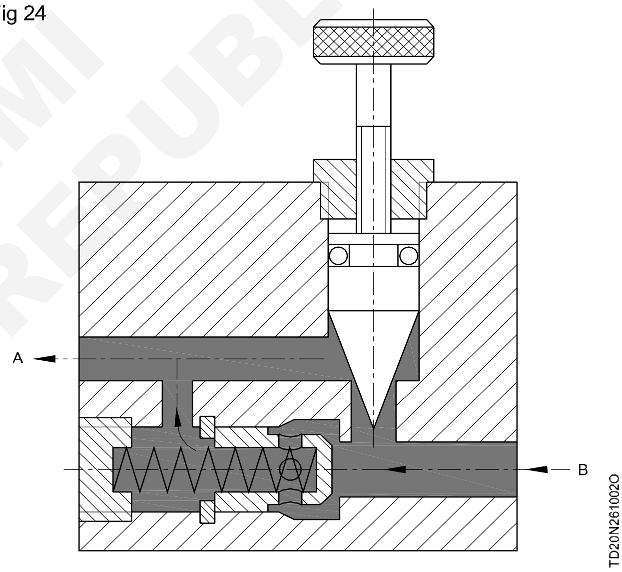
Operation : The free, annular cross-section at restriction [a] is increased or reduced in size by turning the throttling screw. Consequently, the fluid flow in the direction A to B can be varied(restriction), (Fig 23).

Fig 23



As the fluid flows from B to A, the valve cone is forced against the low-rate spring and opens, resulting in unrestricted delivery; flow through the restriction is negligible. (Fig 24)

Fig 24



Application : Return orifice check valves are used where only an approximately constant flow rate is required in one direction and unrestricted flow in the opposite direction.

Symbol : Variable return orifice check valve. (Fig 25)

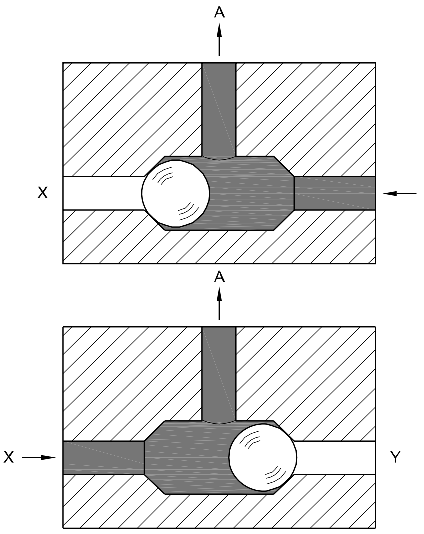
Fig 25

Shuttle valve : This valve is also called "Double control valve or Double check valve". This valve has two inlets X and Y, and one outlet A. If pressurised fluid is applied to inlet X, the ball seals off inlet Y and the fluid flows from X to A. (Fig 26)

Alternatively, the pressurised fluid flows from Y to A and inlet X is closed. When the fluid (i.e. oil from cylinders is returning through the drain line) flow is reversed, the ball remains in its previously assumed position because of the pressure conditions. (Fig 26)

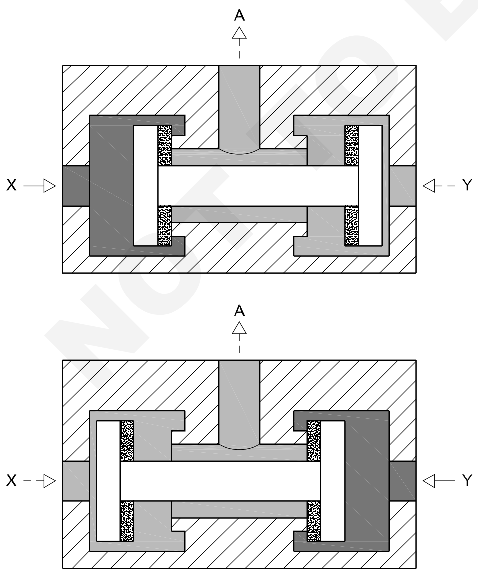
This valve is also called "OR Component". It separates signals emitted from signal valves in different positions and prevents the pressurised fluid from being diverted through a second signal valve. If a cylinder or control valve is to be actuated from two or several positions, a shuttle valve must be used.

Fig 26



Two pressure valve: The two pressure valve has two inlets X and Y, and one outlet A. Pressurised fluid flows through only if the signals are applied on both inlets. One input signal to X or Y blocks the pressurised fluid from flowing to cylinder (out let),because the fluid closes the valve in the side of application of pressurised fluid. If input signals are not applied simultaneously to both sides, the signal, which is last applied, passes to the outlet. If the input signals are of different pressures, the larger of the two pressures closes the valve and smaller pressurised fluid pressure is transferred to outlet A (Fig 27).

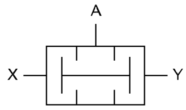
Fig 27



This valve is also known as AND element. It is mainly used for interlocking controls, check functions or logic operations.

Symbol : Two pressure valve (Fig 28).

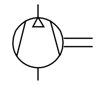
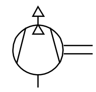
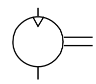
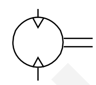
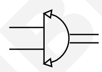
Fig 28



SYMBOLS OF FLUID POWER SYSTEM

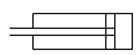
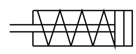
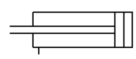
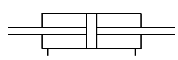
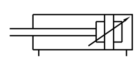
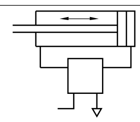
ENERGY CONVERSION

Table 1

	COMPRESSOR
	VACUUM PUMP
AIR MOTORS	
	NON - REVERSING (SINGLE DIRECTION)
	REVERSING TWO DIRECTION
	LIMITED ROTATION

CYLINDERS

Table 2

	SINGLE - ACTING RETURN BY EXTERNAL FORCE
	SPRING RETURN
	DOUBLE - ACTING SINGLE PISTON ROD
	DOUBLE PISTON ROD
	ADJUSTABLE CUSHIONING E.G. BOTH SIDES
	CONTINUOUS DRIVE (RECIPROCATING)

PRESSURE CONTROL VALVES AND FLOW CONTROL VALVES

Table 3

	PRESSURE LIMITING VALVE
	SEQUENCE VALVE
	PRESSURE REGULATOR. NO RELIEF PORT
	PRESSURE REGULATOR. WITH RELIEF PORT
	RESTRICTOR VALVE
	ORIFICE VALVE
	ADJUSTABLE RESTRICTOR VALVE
	RESTRICTOR VALVE. MECHANICALLY ADJUSTABLE BY LEVER ACTUATOR SPRING RETURN
	SHUTOFF VALVE

ENERGY TRANSMISSION

Table 6

	PRESSURE SOURCE
	WORKING LINE
	PILOT OR CONTROL LINE
	EXHAUST LINE
	DASH DOT DOT - ENCLOSURE TO INDICATE ASSEMBLY OR COMBINED GROUP
	ELECTRICAL LEAD
	FLEXIBLE LINE
	LINES JOINING
	LINES PASSING (NOT JOINED)
EXHAUST POINTS	
	NO PIPE CONNECTION
	CONNECTED TO PIPE
PRESSURE CONNECTION	
	PLUGGED
	CONNECTING LINE

ENERGY CONTROL AND REGULATION

Table 5

DIRECTIONAL VALVES	
	2/2 WAY VALVE FLOW P TO A CLOSED IN NORMAL POSITION
	2/2 WAY VALVE FLOW P TO A OPEN IN NORMAL POSITION
	3/2 WAY VALVE FLOW P TO A CLOSED IN NORMAL POSITION
	3/2 WAY VALVE FLOW P TO A OPEN IN NORMAL POSITION
	3/3 WAY VALVE CLOSED CENTER (ALL PORTS CLOSED IN NORMAL POSITION)
	4/2 WAY VALVE
	4/3 WAY VALVE CLOSED CENTER (ALL PORTS CLOSED IN NORMAL POSITION)
	4/3 WAY VALVE OPEN CENTER (OPERATING OUTLETS B AND A OPEN TO EXHAUST. INLET P CLOSED IN NORMAL POSITION)
	CHECK VALVE
	SHUTTLE VALVE
	RESTRICTOR CHECK VALVE. ADJUSTABLE RESTRICTOR (SPEED CONTROL VALVE)
	QUICK - EXHAUST VALVE
	TWO - PRESSURE VALVE

OPERATORS

Table 4

	MANUAL		MECHANICAL
	GENERAL		PLUNGER
	BUTTON		ROLLER
	LEVER		IDLE - RETURN ROLLER
	PEDAL		SPRING
ELECTRICAL		SPRING	
	SOLENOID		PRESSURE
			BLEED
	SOLENOID AND PNEUMATIC PILOT VALVE		DIFFERENTIAL PRESSURE

Pneumatics

Objectives: At the end of this lesson you shall be able to
- **state what is pneumatic**
- **state why the air is used as working substance in pneumatic**
- **list out advange and disadvantages of pneumatic system**
- **list the types of compressions**
- **brief the functions of elements used in pneumatic system and its related symbols.**

Introduction

Pneumatics deals with the mechanics of compressible fluids. Pneumatic systems make use of compressed air or a pressurized gas as a source of power.

Air is used as working substance because of the following reasons:

1 Air as a working fluid is less costlier as it is available free.

2 Large amount of energy can be stored in a limited space.

3 Air has less viscosity than oil.The drop in pressure due to viscosity is less compared to hydraulic fluids.

4 Air has less density. The time required to accelerate a column of air is negligible compared to a liquid column.

5 Air can be used over a wide range of temperatures.

6 Weight to the power ratio of pneumatic systems are much lower than electric motors.

7 Lesser fire hazard.

8 Can be transported through pipes.

9 Cleanliness.

Advantages

- Air is available everwhere.
- Return piping can be dispensed with.
- Air has a comparatively high flow speed in the piping and valves.
- Compressed air can be stored well in the containers.
- Pollution by leakage losses through untight points doesn't occur.
- Easy assembly with plug fasteners and flexible pipes is possible.

Disadvantages

- Air must be treated.
- Leakage losses are costly because of less continuous energy.
- Above 6 bar, technical safety control is necessary.
- Higher force is required for large cylinder diameters.

Uses: For loading, transferring, locating, clamping in machine tools, press tools, jigs and fixtures and in conveyors etc.

To drive pneumatic drills, grinders and hammers etc.

To feed stock strips and ejection of components etc.

In CNC machines and SPM's.

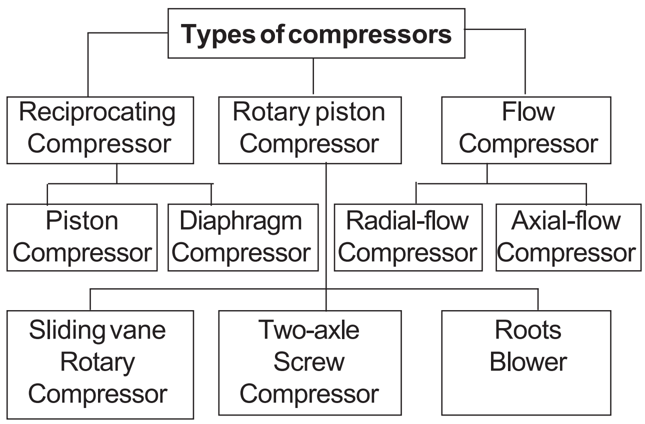
Compressed air production

Production plant : Compressors are used to compress the air to the desired working pressure. The working element is supplied with compressed air by pipe lines from a compressor plant.

Types of compressors : The types of compressors used depend on the operational demands with regard to the working pressure and delivery volume.

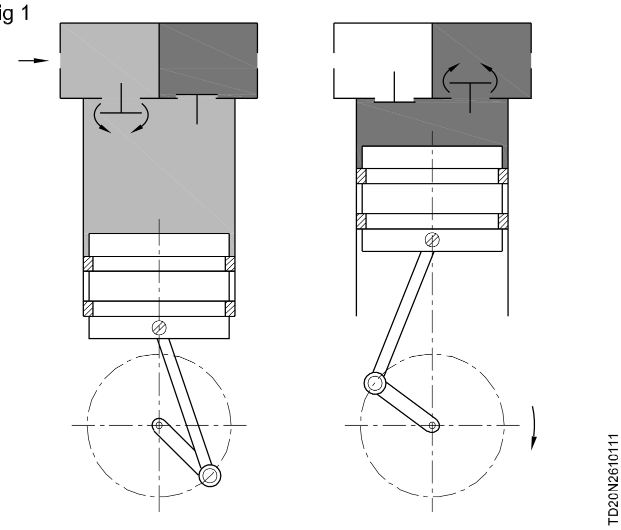
Compressors fall in two groups: The first group operates on the displacement principle where air is compressed by containing it in a chamber and then reducing the volume of the chamber. This type is called piston compressor (reciprocating piston compressor, rotary piston compressor).

The second group operates on the air-flow principle, by drawing air on one side and compressing it by mass acceleration (Turbine).



Reciprocating piston compressor : The reciprocating compressor is the most widely used compressor. Fig 1 shows reciprocating Piston Compressor.

Fig 1



Pressure

Distinction: Working pressure is the compressor outlet pressure or the pressure in the receiver and the pressure in the pipelines to the user.

Operation pressure is the pressure which is required at the operating position. Standard operation pressure is 600kPa (6bar/87psi).

Important: Constant pressure is essential for reliable and accurate operation.

Drive: Depending upon the operational requirements, compressors are either driven by an electrical motor or by internal combustion engine. In factories, compressors are mostly driven by electrical motors.

If the compressor plant is not stationary, the drive is mostly by means of internal combustion engines (petrol or diesel).

Regulation: In order to match the delivery volume of the compressor to the fluctuating consumption, it is necessary to regulate the compressor. Various types of regulation are available for this purpose.

On-Off Regulation is the most commonly used regulation method for stationary compressors.

On-Off Regulation: In On-Off Regulation the compressor has 2 operating conditions (full load or stand still).

The drive motor of the compressor is switched off reaching pmax.When the pressure has dropped to pmin,the motor is switched on again and the compressor starts up.

The switching limits can be set on a regulator. A large compressed air receiver is necessary to reduce the switching periods to an acceptable level.

Fig 2 On-Off regulation.

Cooling: When compressing air in the compressor,heat is generated and must be removed.

Cooling fins on smaller compressor allow the heat to be removed by radiation. Larger compressors are equipped with an additional fans to remove heat. (Fig 3)

Fig 2

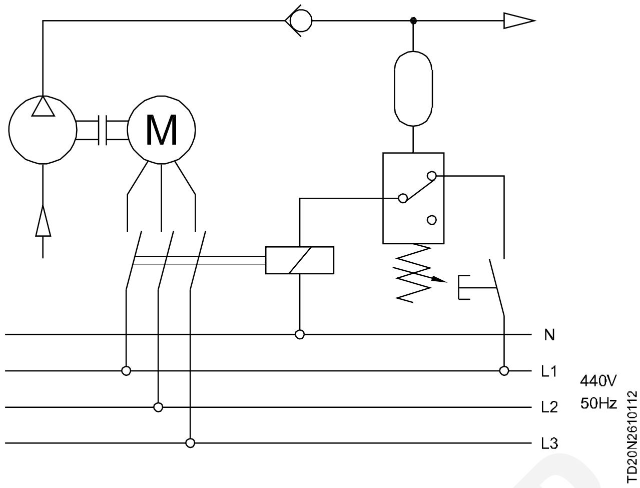
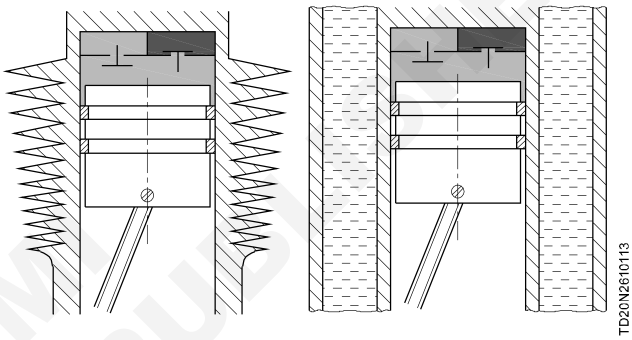


Fig 3



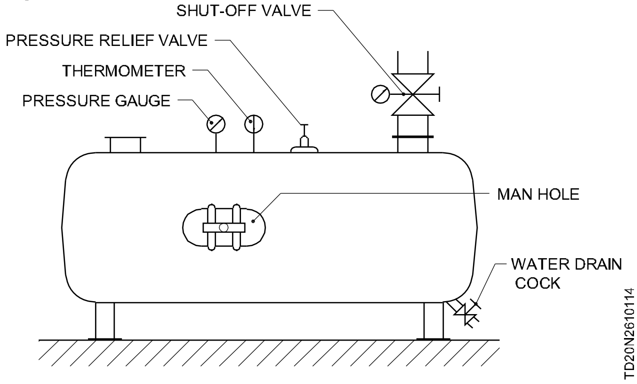
In case compressor plant with a drive power in excess of 30kW, air cooling is no longer sufficient. The compressors are then equipped with water circulation cooling or fresh water cooling.

Installation site: The compressor should be installed in a room on its own and sound proofed towards the outside.The room should be well ventilated. The free air should be cool, dust free, and dry as possible.

Compressed air receiver : The compressed air receiver stabilizes the compressed air supply. It smoothens pressure fluctuations in the network when air is consumed.

In addition,the large surface area of the receiver cools the air. Thus, a portion of the moisture in the air is separated as water directly from the receiver. (Fig 4)

Fig 4



Service unit

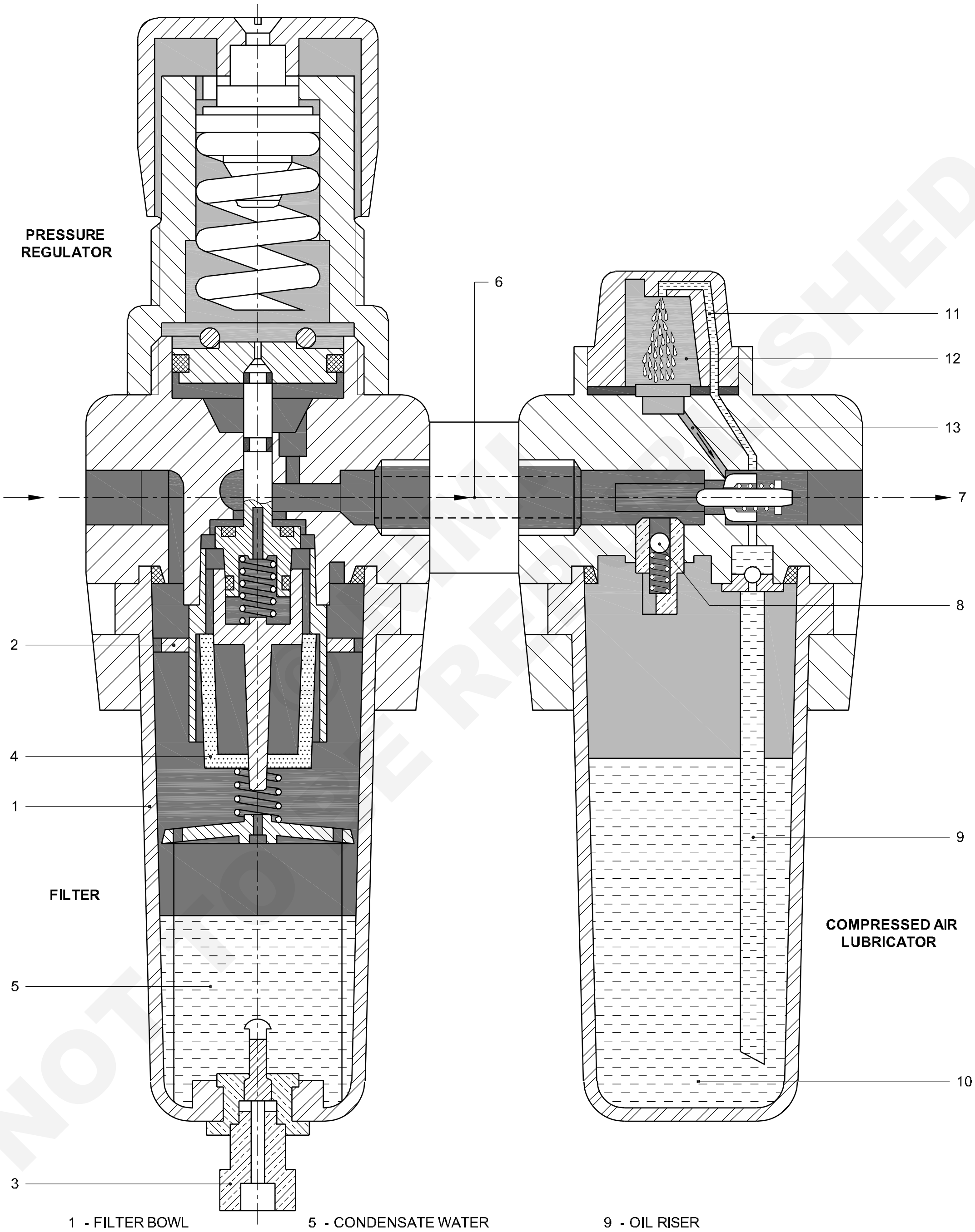
The service unit consists of the following.

Filter : This removes all the minute parts of dust, rust etc.in the air. The air supplied to the working element should be free from foreign matter for efficient and long service life.

Regulator : The operation pressure (pressure of the air supplied to the working elements) should be free from fluctuations. The regulator ensures the fluctuation in pressure to be within limits.

Lubricator : The lubricator adds a fine mist of lubricating oil to the air flowing to working elements. This provides sufficient lubrication to pneumatic components. This unit is also known as F-R-L unit. (Fig 5)

Fig 5



1 - FILTER BOWL
2 - BAFFLE PLATE
3 - DRAIN SCREW
4 - SINTERED FILTER
5 - CONDENSATE WATER
6 - COMPRESSED AIR INLET
7 - OUTLET FROM LUBRICATOR
8 - CHECK VALVE
9 - OIL RISER
10 - LUBRICANT OIL
11 - OIL DUCT
12 - DRIP CHAMBER
13 - OUTLET

COMPRESSED AIR FILTER WITH PRESSURE REGULATOR AND LUBRICATOR (SERVICE UNIT)

TD20N2610115

Maintenance of service unit

Compressed air filters : The condensate level must be checked regularly. The height specified on the sight glass must not be exceeded. The accumulated condensate could otherwise be drawn in to the compressed air pipe. The drain screw on the sight glass must be opened to drain off the condensate.

The filter cartridge in the filter must also be cleaned if it is dirty.

Compressed air regulator : This requires no servicing provided it is preceded by a compressed air filter.

Compressor air lubricator : The oil level in the sight glass should be checked and topped up with oil to the level indicated if necessary.

Pneumatic working elements : The pneumatic power is converted to straight-line reciprocating and rotary motions by pneumatic cylinders and pneumatic motors.

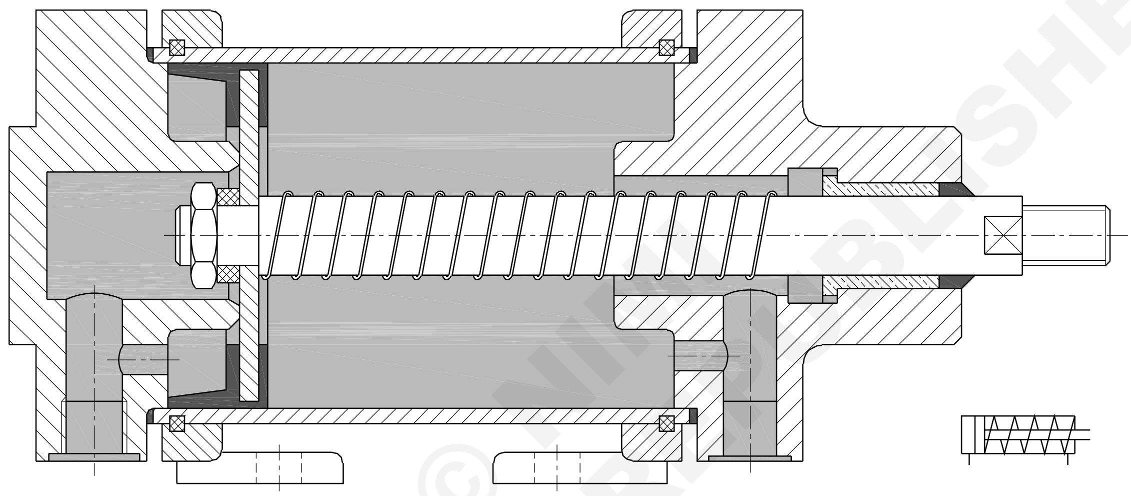
Pneumatic cylinders

Single acting cylinders : In single-acting cylinders compressed air is applied on only one side. These cylinders can do work only in the forward direction. Therefore air required for only the forward movement. A built-in spring moves the piston in the opposite direction.

The spring force of the built-in spring is designed to return the piston to the starting position with a sufficiently high speed. (Fig 6)

In single-acting cylinders with built-in spring the stroke is limited by the natural length of the spring.

Fig 6



TD20N2610116

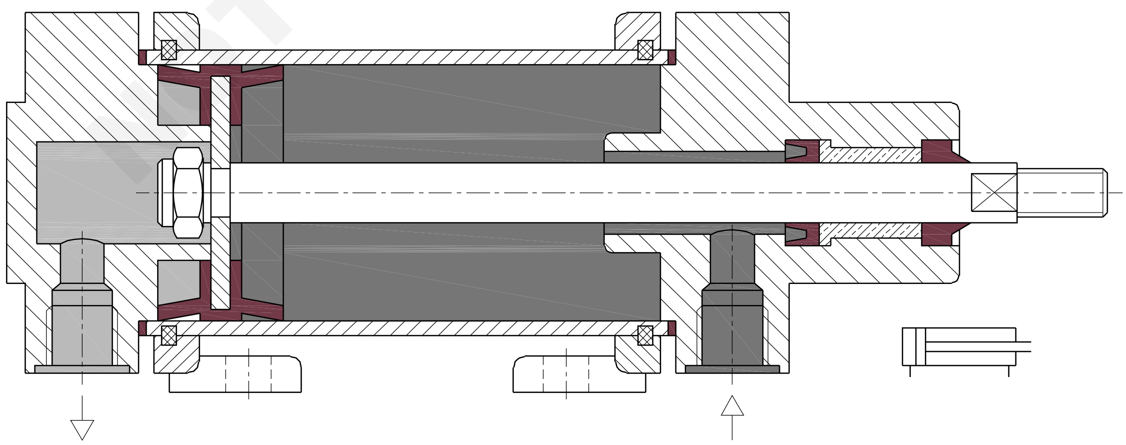
These cylinders are therefore built with stroke length of upto approx. 100mm.

These working elements are used mainly for clamping, ejecting, pressing in, lifting, feeding etc. Fig 6 Single Acting Cylinder.

Double acting cylinders : The compressed air moves the piston in a double acting cylinder in both directions. A definite force is applied on both advance and return movements.

Double acting cylinders are used when the piston is require to perform a work function not only on advance movement but also on the return movement.(Fig 7)

Fig 7



TD20N2610117

Valves

Introduction : Pneumatic control systems consists of signal components, control components, and a working part. The signals and control component influence the operating sequence of the working elements and is termed as valves.

The valves are devices for controlling or regulating “start”, “stop”, and the direction of movement as well as pressure and rate of flow of the medium.

Valves are divided into 5 groups according to their function

1 Directional valves (Way valves)

2 Non-return valves

3 Pressure control valves

4 Flow control valves

5 Shut-off valves

Directional valves : Directional valves are devices which influence the path, taken by the air stream. It controls the start, stop and direction of the flow of air.

Representation of directional valves : Symbols are used for representing valves in circuit diagrams. These symbols specify only the function of the valve and do not indicate the design principle on which it is constructed.

Valve switching positions are represented as squares. (Fig 8a)

The number of adjacent squares shows how many switching positions the valve has. (Fig 8b)

The function and working principle is drawn inside the boxes(squares).

Lines indicates flow paths. Arrows show the direction of flow. (Fig 8c)

Shut-off positions are identified in the boxes by the lines drawn at right angles. (Fig 8d)

Flow path junctions are represented by a dot. (Fig 8e)

Eg.3/2 Way valve (Fig 8f) actuated by push button, reset by spring.

3 flow paths, 2 switching positions 4/2 Way valve (Fig 8g) actuated by direct application of pressure, reset by spring

4 flow paths, 2 switching positions

AB Represents flow path

R Represents exhaust line

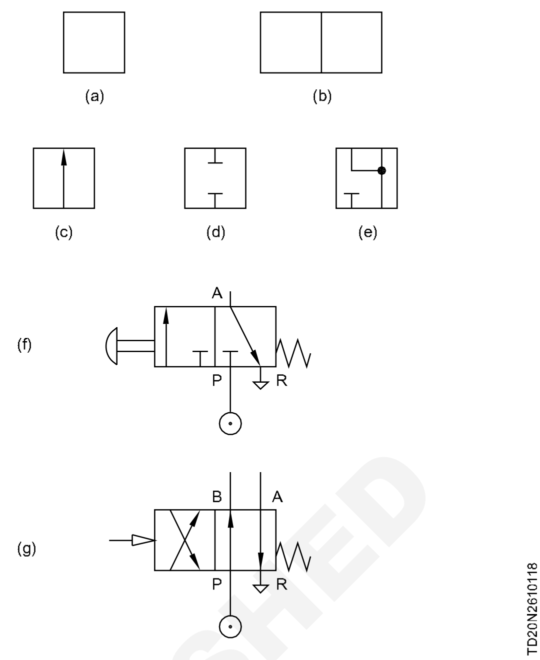
P Represents pressure line

Direction control valves (DC valves)

Types of DC valves

Poppet valves	Ball seat valves Disc seat valves
Slide valves	Longitudinal slide valve Longitudinal flat slide valve Plate slide valve (Butterfly valve)

Fig 8



Poppet valves : In Poppet valves, the connections are opened and closed by means of balls, discs, plates or cones. The valve seats are usually sealed using elastic seals. Seat valves have few parts which are subject to wear and hence they have long service life. They are insensitive to dirt and are robust.

The actuating force is relatively high, as it is necessary to overcome the force of built-in reset spring and the air pressure.

3/2 Way valve

The valves have 3 flow paths and 2 switching positions. 3/2 Way Valve Normal Position Closed:\In normal position pressure line ‘P’ is closed. Exhaust ‘R’ is connected to working line ‘A’. Actuation of the plunger causes the exhaust air line from A to R to be closed.

On pressing further, the disc is lifted from the seat, and the compressed air flows from P to A. On releasing the plunger, the reset spring brings the valve to the closed position.

Fig 9. Normal position closed.

The 3/2 Way valves are used for controls employing single-acting cylinders or for controlling final control elements.

3/2 Way valve normal position opened : In a valve which in normal position is open the pressure line P is connected to A. The connection from P to A is closed with a disc when operated. On pressing further, a second disc opens the sealing seat from A to R. When the plunger is released, the connections P to A is opened and R is closed.

(Fig 10) Normal position open

The valves can be actuated manually, mechanically, electrically or pneumatically.

Fig 9

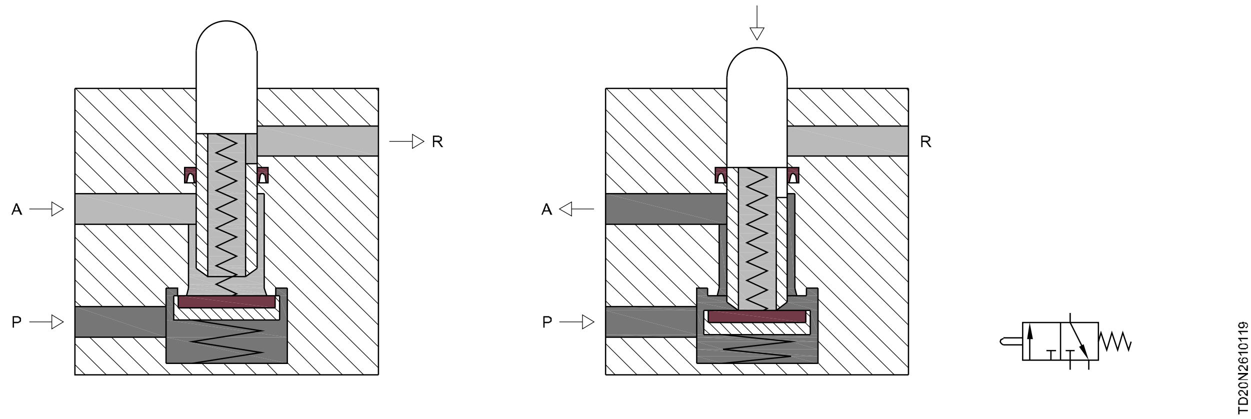
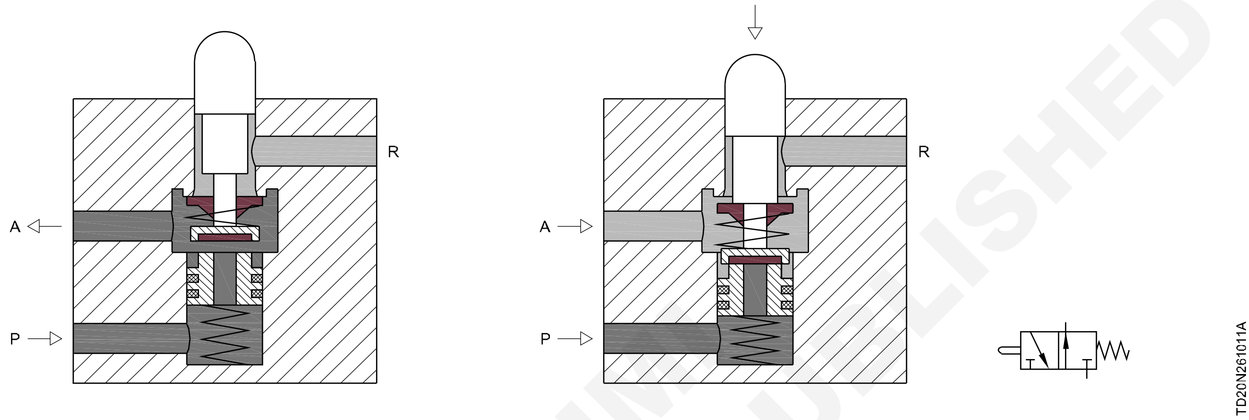


Fig 10



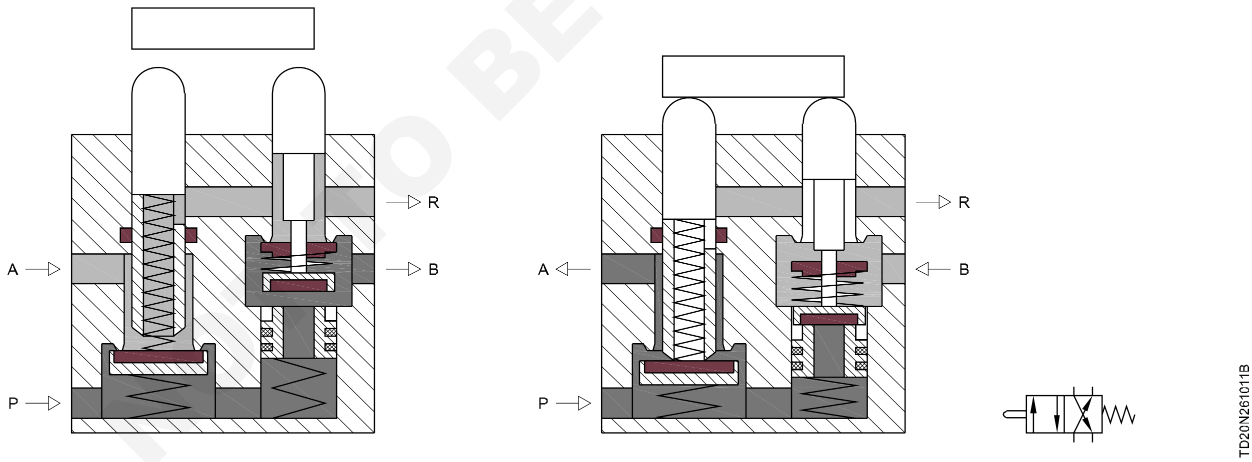
4/2 Way valve

4/2 Way Valve has 4 flow paths and 2 switching positions.

One position of the valve shows the paths P to B and A to R are open. When the two plungers are actuated. Simultaneously P to B and A to R are closed. The connection from P to A and from B to R are opened.

The valve is used for controlling double acting cylinders. 4/2 Way Valve. (Fig 11)

Fig 11



Solenoid valves : These valves get the switching impulse from an electrical timing device, electrical limit switches or electronic controllers. Electrical actuation is normally selected for controls involving extremely long distances and short switching times. (Fig 12)

Slide valves : In slide valves the connections are opened or closed by means of spool slides, spool flat slides or butterfly valves. Example of an 4/2 Way slide valve is shown in Fig 13.

Non-return valves : Non-return valves are devices which stop the flow in one direction and permit flow in the opposite direction. The pressure on the downstream side acts against the restrictive component, there by assisting the sealing effect of the valve.

Check valve : Check valve can stop the flow completely in one direction, and the air flows in the opposite direction with as low pressure loss as possible. Blocking of the one direction can be effected by cones, balls, plates or diaphragms. (Fig 14)

Fig 12

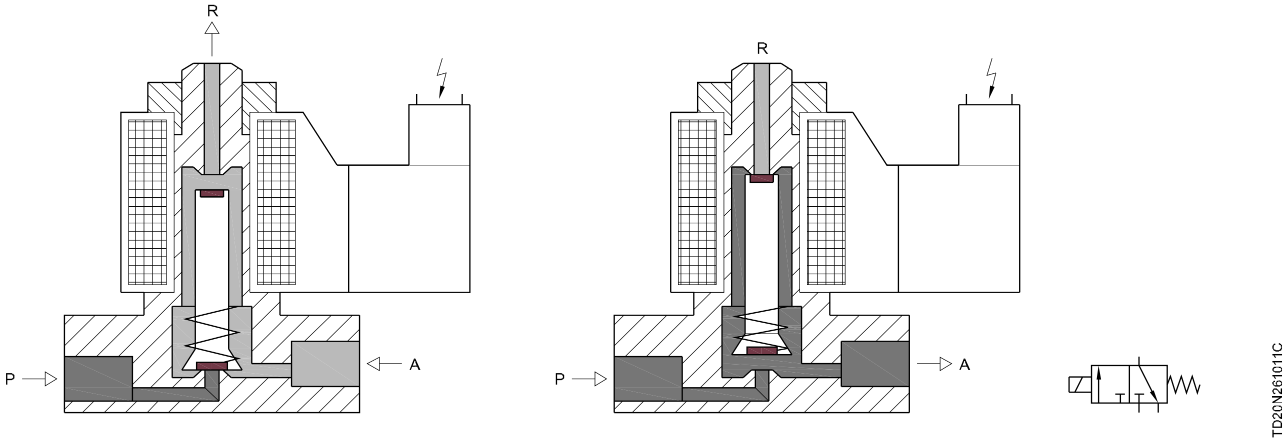


Fig 13

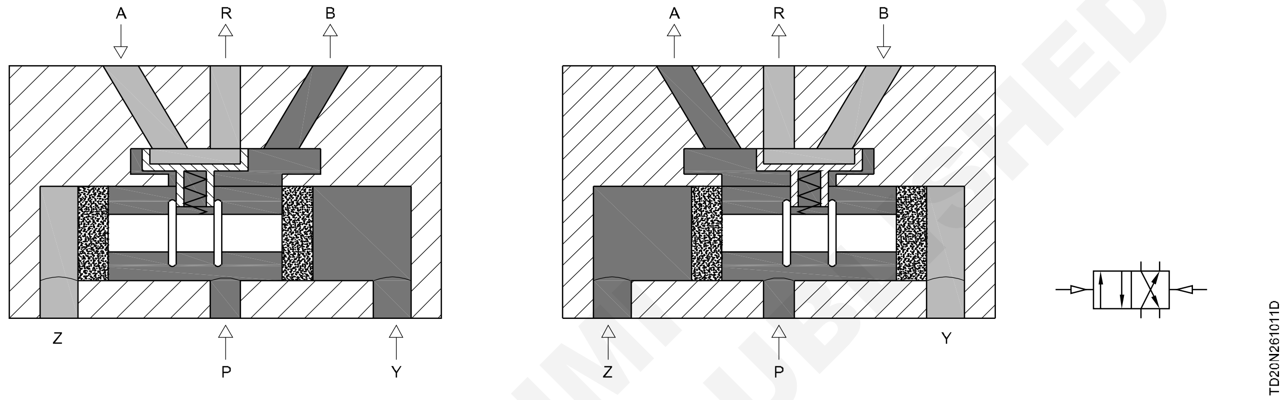
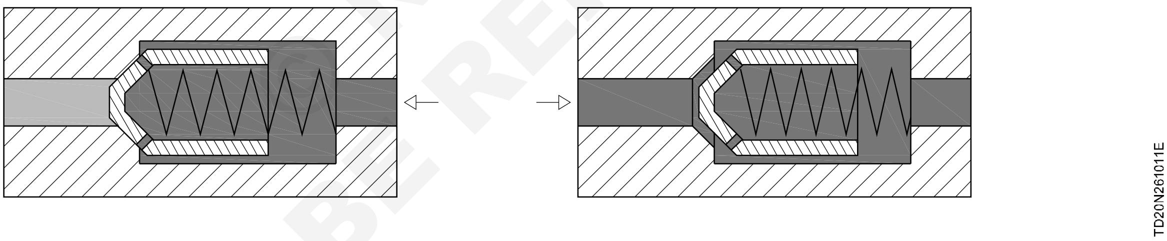


Fig 14



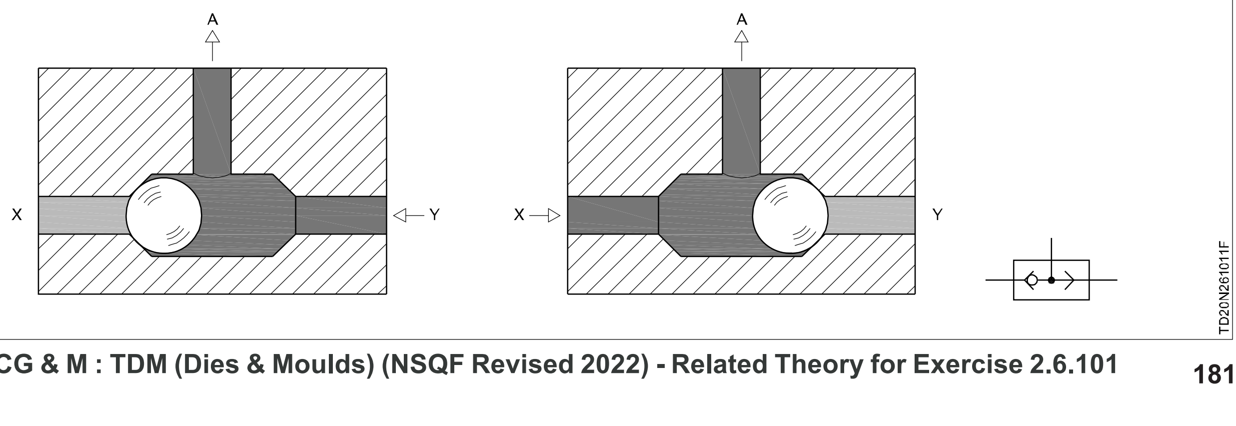
Shuttle valve: This valve is also called “Double control valve or Double check valve”. This non-return valve has two inlets X and Y, and one outlet A, if compressed air is applied to inlet X, the ball seals off inlet Y and the air flows from X to A.

Alternatively, the air flows from Y to A and inlet X is closed. When the air flow is reversed, i.e., a cylinder is exhausted, the ball remains in its previously assumed position because of the pressure conditions. (Fig 15)

This valve is also called “OR Component”. It separates signals emitted from signal valves in different positions and prevents the air from being diverted through a second signal valve. If a cylinder or control valve is to be actuated from two or several positions, a shuttle valve must be used.

Throttle relief valve : This is also known as speed regulating valve. In a throttle relief valve, the air flow is throttled in one direction only.

Fig 15



A check valve blocks the flow of air in one direction and the air can flow on through the regulated cross section. In the opposite direction, the air can flow freely through the opened check valve. These valves are used for speed regulation of pneumatic cylinders. (Fig 16)

Quick exhaust valve : Quick exhaust valves are used to increase the piston speed of the cylinders. This enables lengthy return times to be avoided, particularly with single acting cylinders.

The valve has a blockable pressure connection P, a blockable exhaust R, and an outlet A.

If the pressure is applied at the connection P, the sealing disc completely covers the exhaust orifice R. The compressed air thus flows to A. If the pressure is removed from P, the air coming from A moves the sealing disc against connection P and closes it. The exhaust air can flow directly to atmosphere without having to follow a long and possibly narrow path through the control lines to the pilot valve. It is best to mount the quick-exhaust valve directly on the cylinder or as near as it is possible. (Fig 17)

Fig 16

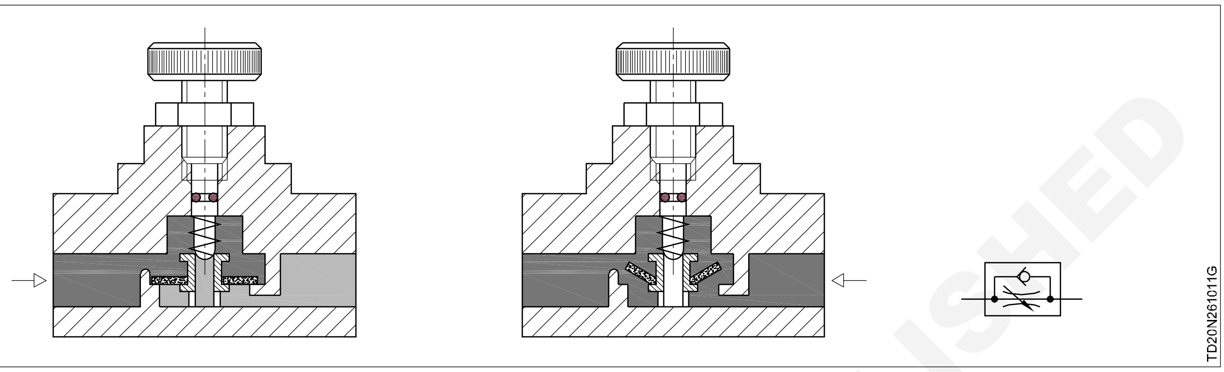
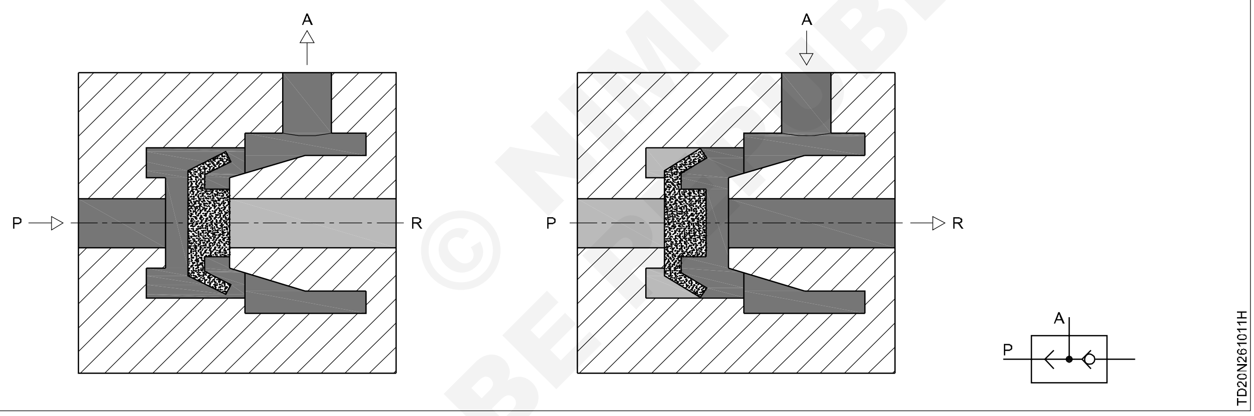


Fig 17



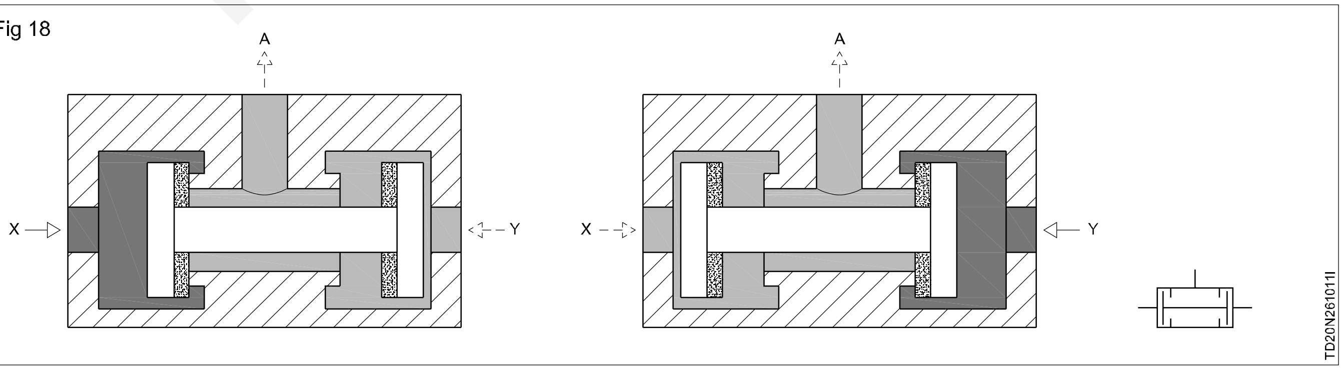
Two pressure valve : The two pressure valve has two inlets X and Y, and one outlet A. Compressed air flows through only if the signals are applied on both inlets. One output signal to X or Y blocks the air because the compound air closes the valve in the side of application of compound air. If input signals are not applied simultaneously to both sides, the signal, which is last applied, passes to the outlet. If the input signals are of different pressures, the larger of the two pressures closes the valve and smaller air pressure is transferred to outlet A.

This valve is also known as AND element.

It is mainly used for interlocking controls, check functions or logic operations.

Two pressure valve (Fig 18) : This valve is also known as AND element. It is mainly used for interlocking controls, check functions or logic operations.

Fig 18



Automatic lubricating system

Objectives: At the end of this lesson you shall be able to
- **state what is automatic lubricating system**
- **list the different types of automatic lubricating system**
- **brief the various types of automatic lubricating system.**

An Automatic Lubrication System, sometimes referred to as a Centralized Lubrication · System, is a system that delivers controlled amounts of lubricant to multiple locations on a machine while the machine is operating

There are several different types of automatic lubrication systems including:

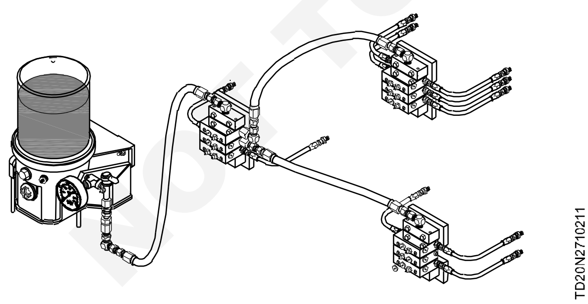
- Single Line Parallel systems .
- Dual Line Parallel systems.
- Single Point Automatics.
- Single Line Progressive systems (or Series Progressive)
- Single Line Resistance.
- Oil Mist and Air-Oilsystems.
- Oil re-circulating.
- Chain lube systems.

The 4 most commonly used Automatic Lubrication System types are:

- Single Line Parallel,
- Dual Line Parallel and
- Single Line Progressive.
- Multi port direct lubricators

Single line progressive (Fig 1)

Fig 1

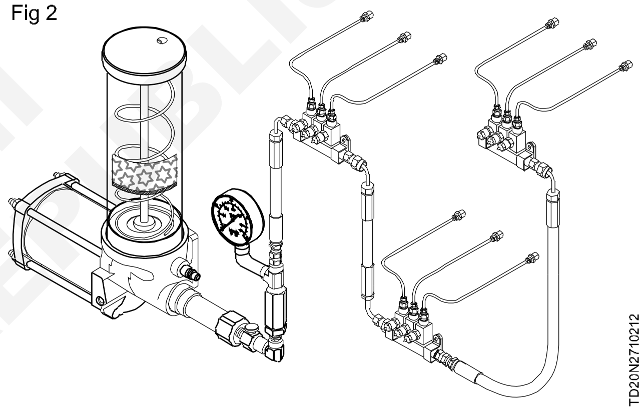


A single line progressive system uses lubricant flow to cycle individual metering valves and valve assemblies. The valves consist of dispensing pistons moving back and forth in a specific bore. Each piston depends on flow from the previous piston to shift and displace lubricant. If one piston doesn't shift, none of the following pistons will shift valve output is not adjustable.

Operation begins when the controller/timer sends a signal to the pump to start the lube event. The pump then feeds lubricant into the supply line which connects to the primary metering valve, for either a preprogrammed amount of time or number of times as monitored through a designated piston cycle switch. Lubricant is fed to the multiple lubrication points one after another via secondary progressive metering valves sized for each series of lubrication points, and then directly to each point via the feed lines.

Single line parallel (Fig 2)

Fig 2



A single line parallel system can service a single machine, different zones on a single machine or even several separate machines and is ideal when the volume of lubricant varies for each point. In this type of system, a central pump station automatically delivers lubricant through a single supply line to multiple branches of injectors. Each injector serves a single lubrication point, operates independently and may be individually adjusted to deliver the desired amount of lubricant.

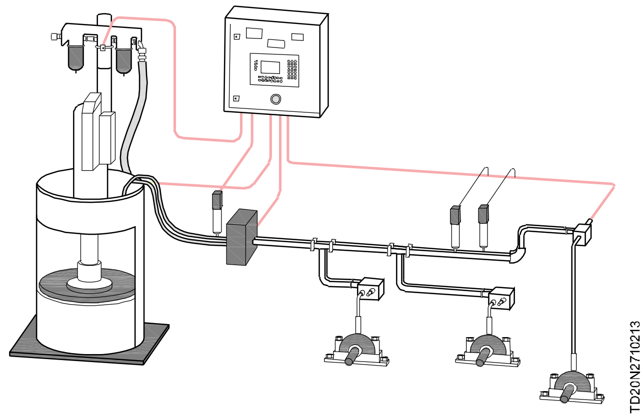
Operation begins when the controller/timer sends a signal to the pump starting the lube cycle. The pump begins pumping lubricant to build up pressure in the supply line connecting the pump to the injectors. Once the required pressure is reached, the lube injectors dispense a predetermined amount of lubricant to the lubrication points via feed lines.

Once the entire system reaches the required pressure, a pressure switch sends a signal to the controller indicating that grease has cycled through to all the distribution points.

The pump shuts off. Pressure is vented out of the system and grease in the line is redirected back to the pump reservoir, until the normal system pressure level is restored.

Dual line parallel (Fig 3)

Fig 3



A dual line parallel system is similar to the single line parallel system in that it uses hydraulic pressure to cycle adjustable valves to dispense measured shots of lubricant. It has 2 main supply lines which are alternatively used as pressure I vent lines.The advantage of a two-line system is that it can handle hundreds of lubrication points from a single pump station over several thousand feet using significantly smaller tubing or pipe.

Operation begins when the controller/timer sends a signal to the pump to start the lubrication cycle. The pump begins pumping lubricant to build up pressure in the first (the pressure) supply line while simultaneously venting the second (vent) return line. Once the required pressure is reached, a predetermined amount of lubricant is dispensed by the metering devices to half of the lubrication points via feed lines.

Once the pressure switch monitoring main supply line pressure indicates a preset pressure in the line has been reached, the system is hydraulically closed. The controller shuts off the pumri and signals a changeover valve to redirect lubricant to the second main supply line.

The next time the controller activates the system, the second main line now becomes the pressure line while the first line becomes the vent line.The second line is pressurized and the entire process is repeated lubricating the remaining lube points.

Multi point direct lubricator

When the controller in the pump or external controller activates the drive motor, a set of cams turns and activates individual injectors or pump elements to dispense a fixed amount of lubricant to each individual lubrication point. Systems are easy to design, direct pump to lube point without added accessories and easy to troubleshoot.

Maintenance and its types

Objectives: At the end of this lesson you shall be able to
- **state what is maintenance**
- **list the types of maintenance**
- **brief the each types of maintenance.**

Maintenance

Maintenance is a combination of any actions carried out to retain an item in, or restore it to an acceptable condition.

Types of maintenance

Breakdown maintenance

In breakdown maintenance the equipment is usually attended to only after it breaks down. Despite its numerous disadvantages this type of system may be suitable in certain conditions such as the equipment is non critical and standbys are available or the plant capacity exceeds market demand.

Preventive maintenance

Organising maintenance before the needs being developed would minimize the possibility of anticipated break downs.

Scheduled maintenance

Analysis of routine maintenance like cleaning greasing etc. which will keep equipments running efficiently and in a state of reliable operational readiness.

Reconditioning

At periodic intervals, depending on type and nature of the machine or equipment and its condition, it needs to be overhauled or reconditioned. It is the process of bringing the machine back to its new conditions.

Corrective maintenance

A study of failure of equipment in service may warrant a change in design; materials or working conditions of the equipment and corrective steps should be taken thereafter.

Maintenance prevention

While designing and developing the equipment objective is set to provide no maintenance or higher maintainability which would reduce the maintenance effort in the life time.

Predictive maintenance

While the equipment is in actual operating condition a study of performance of the equipment would reveal whether unexpected deterioration is taking place in it and the range of frequency of scheduled maintenance to reduce such deterioration.

Condition based maintenance

In this system of maintenance, the machine is continually monitored to see the health of machine derived from different electrical instrument, in the form of mechanical vibration, noise, sound, thermal emissions, change in chemical composition, small pressure and relative displacement and so on. From this result the severity of faults can be evaluated for decision making.

Pro- active maintenance

The most recent innovation in maintenance is called Pro-active Maintenance.

It includes a technique called "root causes failure analysis" in which the primary cause of the machine failure is sought and corrected.

The root causes of the majority of machine faults are imbalance and misalignment. Both of these condition place undue force on bearing, shortening their service life. Rather than continual · replacing worn bearing, a far better policy is, to perform precision balance and alignment on the machine and to verify the result by vibration analysis.

Total productive maintenance (TPM)

To improve the company, the attitudes and skills of all the personnel's from top management down to shop floor workers , the idea that, "I am the operator and you're the maintenance man" was deep rooted and the operator had no interest in maintenance.

Changing this idea on the part of all the employees and so the operator autonomously maintain the equipment he use by himself is the first step in TPM.

TPM takes on the challenge of

ZERO LOSSES

ZERO FAILURES

ZERO DEFECTS

System of symbol and colour coding for lubricants

Objectives: At the end of this lesson you shall be able to
- **state the necessity of the colour coding system for lubricants**
- **list the points that helps the management by colour coding**
- **brief the colour and shapes in colour coding.**

A color code or colour code is a system for displaying information by using different colors. The earliest examples of color codes in use are for long distance communication by use of flags, as in semaphore communication. ... On forms and signage, the use of colour can distract from black and white text.

The Ease and Simplicity of Color-Coded Lubrication Management

The idea behind color-coded lubrication management is simple: Everything that comes into contact with lubricant during the maintenance process is marked with a color and symbol that correspond to a particular lubricant. This means all the drums, dispensers, transfer containers, and fill points in a color-coded system have colored tags·or labels of some kind. Workers can easily see which items are to be used with which lubricant and where each lubricant should be applied. Following the path of any given lubricant from storage to application is as easy as connecting the dots -or colored symbols

Color-coded lubrication management helps you do the following

1 Prevent Lubricant Misapplication

2 Eliminate Cross-Contamination

3 Foster Workplace Safety

4 Safeguard Your Workflow

The colour of lubricating oils can range from transparent to opaque. The color is based on the crude from which it is made, its viscosity, method and degree of treatment during refining, and the amount and types of additives included.

In addition, each viscosity grade has its·own colour that can be standardized throughout the plant. For oils, the LIS tag is square-shaped, while for greases the tag is round.

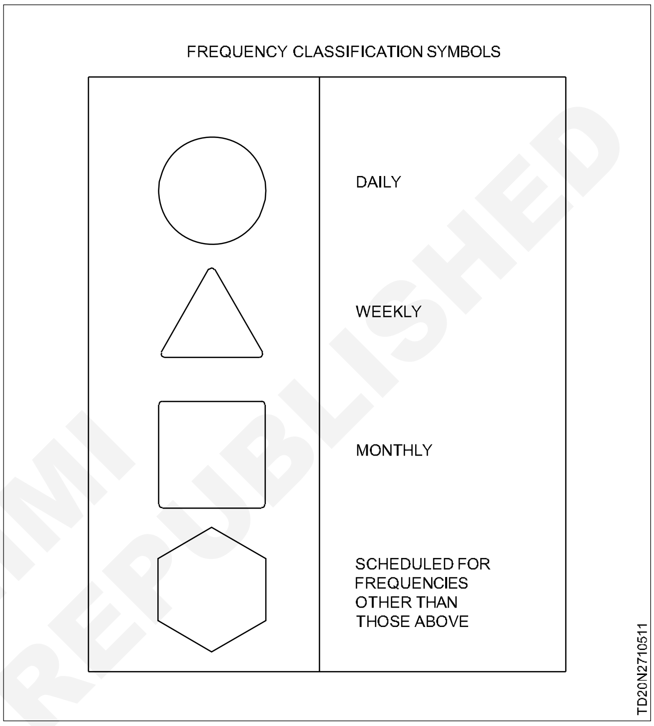
Example of the colour coding system is shown in Table 1

However

A circle to represent the need for daily lubrication, A triangle for weekly lubrication, and

A square to represent monthly intervals between lubrication activities.

Table 1



For activities conducted on a quarterly basis (or over longer periods), the square was to again be used, but this time with a number painted inside the square to highlight the number of interval months.

Lubricant storage and transfer systems, though, reflect just one area where colorization pays off for a site.Another important use of color identification involves a condition-based approach to filling oil reservoirs.

Fig 1 is a good example of this Hi-Lo technique. Itinvolves using red, amber (yellow), and green lines taped on the side of an automated-lubrication-system reservoir. Tills arrangement is known as a RAG (red/amber/green), or the traffic-light indicator system:

- The green line indicates the upper fill level.
- The amber (yellow) line indicates a level at whlch the operator is to contact the maintenance department with a first request to fill the reservoir.
- The red line alerts the operator to call ina priority request to fill the reservoir

Fig 2

Possible causes for Failure and Remedies

Objectives: At the end of this lesson you shall be able to
- **list out the various causes for failures in machines**
- **suggest the remedial measure to overcome the failures.**

Possible causes for Failure and Remedies

A machine tool is considered to have failed or impaired under one of the three conditions when:

- It becomes completely in-operable.
- It is still operable but is no longer able to perform its intended function satisfactorily.
- Serious deterioration has made it unreliable or unsafe for continued use.

The fundamental causes of failure include many aspects of which the following are of prime importance if design and manufacturing aspects have attained desired standard.

- Improper installation.
- Improper assembly and misalignment.
- Improper lubrication.
- Overloading and mishandling.
- Excessive temperature generation.
- Improper service condition.

Improper installation

Proper functioning of machine tool is very much depending upon its foundation. It should be such that the dead load of the machine is evenly supported and the alignments are maintained. To achieve this, proper foundation must be made and levelling of the machine tool is done after installation. Improper foundation causes uneven stress development in the machine body which leads to vibration, misalignment and in accuracy in its output. Vibration will cause false brinelling failure in ball bearing.

Improper assembly and misalignment

Failure also causes due to error in assembly and misalignment. Faulty bearing assembly can lead to bearing failure. Skew running marks in the outer face is caused due to misalignment. Fatigue failure is caused due to misalignment. Misalignment of plain bearing housing on assembly leads to bearing failure. Excessive interference fit cause overheating and fatigue in bearing.

Improper lubrication

Failure due to improper lubrication can be caused by:

- Improper selection of lubricant.
- Inadequate lubrication.
- In correct lubricating intervals.
- Sub - standard lubricant.

Such failures often can be attributed to improper bearing etc, besides the above causes.

Proper grade of lubricant as recommended by the manufacturer of machine tool to be used in between the mating components to minimize friction and wear. If machine manual is not available, suitable lubricant is to be selected considering the load, speed, temperature generation and the type of bearing. This calls for considerable experience in the field.

Sub - standard lubricant likely to cause increased rate of wear including corrosive wear, so genuine lubricant of proper grade is of great importance for lubrication.

Overloading and mishandling

Overloading is generally caused due to the following reasons.

- Increased depth of cut.
- More speed and feed considering the metal to be cut.
- Sudden shock load.
- Improper handling of machine tool.

Excessive temperature generation

Stress development at elevated temperature produces continuous strain on the components. This results in creep and eventually leads to fracture, depending upon the type of metal. Besides, high temperature generation bring about quality deterioration in lubricant which helps in premature bearing failure. Principle types of elevated temperature failures are

- Creep and stress rupture.
- Thermal fatigue.
- Tension over load.

Due to temperature generation, components expand and this expansion varies as per the metal. The expansion provisions must be kept in shaft and bearing assembly, otherwise this will lead to failure of parts.

Improper service condition

Improper environmental condition such as dust prone areas, toxic fumes and chemicals causes failure to mechanisms.

Antifriction bearings fail very easily if abrasive particles enter into it. In hostile environmental condition schedule maintenance and inspection should be carried out regularly to avoid premature failure.

Check points

Checkpoints for nuts & Bolts

Slight defects	Loose nuts & bolts, missing nuts & bolts
Bolt length Washers	All bolts should protrude from nuts by 2 -3 threads • Are flat washers used on long holes? • Are tapered washers used on angle bars & channels? • Are spring washers used for parts subjected to vibration? • Are identical washers used in identical parts?
Fixing of nuts & bolts	• Are bolts inserted from below, are they visible from outside? • Are devices such as limit switches fastened at least by 2 bolts? • Are wing nut directions clockwise?

Lubrication Checkpoints

Lubricant Storage	• Are lubricant stores always kept clean, tidy, are 5S principles applied? • Are lubricant containers always capped? • Are lubricant type clearly indicated and is under proper stock control practices?
Lubricant inlets	• Are grease nipples, speed reducer lubricant ports and other lubricant inlets always kept clean? • Are lubricant inlets dust proofed? • Are lubricant inlets labeled with correct type and quantity of lub?
Oil Level gauges	• Are oil level gauges always kept clean, are oil levels easy to check? • Is the correct oil level clearly marked? • Is equipment free of oil leaks, and oil pipes and breathers unobstructed?
Auto Lub Devices	Are Auto lubricating devices operating correctly & supplying correct quantity of lubricants?
Lubricating condition	• Are rotating, sliding parts, and transmission well lubricated? • Are the surroundings free of contamination by excess lubricant?

Transmission System Checkpoints

V - Belts & Pulleys	• Are any belt cracked, swollen, worn out, or contaminate by lubrication? • Are any belts twisted or missing. Are any belts stretched or slack? • Are multiple belts under uniform tension and call of the same type? • Are top surfaces of belts protruding above the pulley rims? Are the bottoms any pulley grooves shiny ?(indicating a worn belt or pulley) • Are pulleys correctly aligned?
Roller chains	• Are any chains stretched indicating worn out pins of bushings? • Are any sprocket teeth worn, missing or damaged? • Is lubrication between pins and bushings sufficient? • Are sprockets correctly aligned?
Shafts, bearings &Couplings	• Is there any overheating, vibration, abnormal noise due to excessive play, or lack of greasing? Are any keys or set bolts loose or missing? • Are any couplings misaligned or wobbling? Are any seals worn out?
Gears	• Are gears rightly lubricated with proper amount of lubricant? • Are any teeth missing, damaged worn out or jammed? • Is there any unusual noise or vibration?

Hydraulic Check Points

Hydraulic Check Points	• Is there correct quantity & Level indicated for fluid in the reservoir? • Is fluid of correct Temp and is minimum and maximum for temperature indicated? • Is fluid cloudy (indicating air entrapment) • Are all fluids inlets & strainers clean, are any suction filters blocked? • Are any fluid reservoir breather filters blocked? • Are fluid pumps operating without any unusual noise or vibration? • Is hydraulic pressure correct, and are operating ranges displayed?
Heat exchangers	• Is any fluid for water leaking from fluid coolers or pipes? • Are temperature differences between fluid & water inlets & outlets are correct? Are any tubes blocked?
Hydraulic Equipment	• Are there any fluid leaks? Are hydraulic devices properly fastened? • Are hydraulic devices operating correctly without speed loss or breathing? Is hydraulic pressure correct, and all gauges working.
Piping and Wiring	• Are all pipes and hoses fastened tightly? • Are there any fluid leaks? Are any hoses cracked or damaged? • Are all valves operating correctly? Is it easy to see whether valves are open or closed I Are any pipes, valves, unnecessary?

Pneumatic Checkpoints

FRLs	• Are FRLs clean? Is it easy to see inside of them? Are they fitted right way around? • Is there sufficient oil, and are the drains are clear? • Is there oil dip rate correct? (approx 1 drop for every 10 strokes) • Are the FRLs installed not more than 3m from the equipment? • Are the pressure adjusted to the correct value and operating ranges are clearly indicated?
Pneumatic Equipment	• Is compressed air leaking from pneumatic cylinders or solenoids? • Are all pneumatic cylinders and solenoid valves firmly fastened? • Are any pistons dirty, worn and damaged? • Are the speed controllers installed right ways around? • Is there any abnormal sound or overheating of solenoid valves and are any lead wires chafted or trailing?
Piping and Wiring	• Are there any places in pneumatic pipes or hoses where fluid is liable to collect? Are all hoses and pipes clipped firmly in place? • Are any hoses cracked, damaged, are there any air leaks? • Are valves operating correctly and open and close position known?

Electrical Checkpoints

Control Panels	• Are the interiors of distribution boards, switch boards and control panels kept clean, tidy and well organized by the application of 5S activities? Have any objection or flammable materials been left inside? • Is the wiring inside control panels in good condition? Are any wires coiled or training? • Are all ammeters and voltmeters operating correctly? • Are any switches, bulbs broken? Do all switches work correctly? • Are panel doors in good condition? Do they open & close easily?

	• Are they any unused holes? Are control panels water & dustproof?
Electrical Equipment	• Are all motors free of overheating, vibration, unusual noise & smell? • Are all motor cooling fans and fins are clean? • Are any bolts loose? Are pedestals free of cracks and damaged?
Sensors	• Are all manual switches clean and free of damages & excessive play? • Are all switches installed in the correct position? • Are the interiors of limit switches is clean? Are any wires coiled or trailing? • Are any limit switch dogs worn, deformed, or of the wrong shape? • Are all photoelectric switches and proximity switches clean and free of excessive play? • Are any sensors out of position? Are correct positions clearly indicated? • Are all lead wires unchafed, and is insulation intact at entry points?
Switches	• Are all switches installed in the correct position? • Are all manual switches clean and free of damages & excessive play? • Are emergency stop switches installed in the appropriate locations and are they in qood working condition?
Piping and Wiring	• Are any pipes, wires or power leads loose or unsecured? • Are any ground wires damaged or disconnected? • Are any pipes damaged or corroded? Are there any bare wires or wires with damaged insulation? • Are any wires coiled on the floor or dangling overhead?

Hot runner moulds

Objectives: At the end of this lesson you shall be able to
- **state what is hot runner mould and runner less mould**
- **list the type of nozzle used in runner less mould**
- **brief the advantages and disadvantages of hot runner moulding system**
- **explain the functions of different types of hot runner mould**
- **brief the functions of manifold, manifold drops**
- **explain the valve system used on the mould**
- **brief the steps in starting/restarting nozzles in manifold application.**

Hot- runner moulds

The term "hot-runner moulds" refers to any injection moulding process, in which, it is not required to remove the sprue and runners between each successive cycle. The purpose of the hot runner system is to distribute the hot plastic coming out from the machine nozzle to all gates of the mould at the same time and pressure.

Runnerless mould

Any mould in which a conventional runner system is not incorporated is called runnerless mould. A mould, which incorporates a direct feed from the nozzle, is a basic runner less mould.

This basic arrangement has been developed in order to reduce the sprue length or to eliminate the sprue altogether by using different types of nozzle design. There are four types:

Types of nozzle

1 Long-reach nozzle

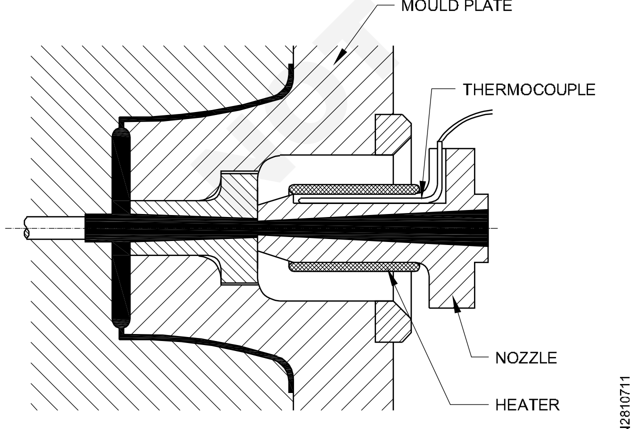
2 Barb nozzles

3 Anti-Chamber nozzles

4 Multi-nozzle manifolds

1 Long-reach nozzle (Fig 1)

Fig 1



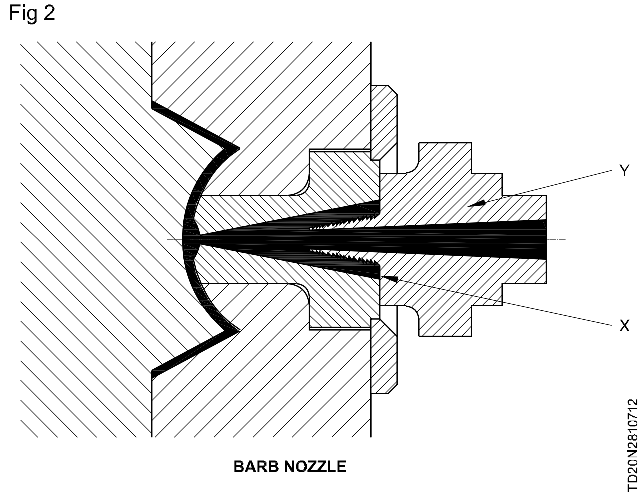
LONG-REACH NOZZLE

The advantage of keeping the length of the sprue as short as possible is to minimize the pressure drop across the gate and also, to minimize the blemish left on the moulding, when the sprue gate is removed. With the standard nozzle design, the length of the sprue gate is controlled by the depth of the mould plate. The length of the sprue gate can be reduced quite simply by designing a special nozzle variously known as "Long-reach nozzle or Extended nozzle". This extended nozzle protrudes into a pocket machined in the mould plate, thus, reducing the length of sprue gate. Heating it, using a resistance type of band heaters or a thermocouple controls the temperature of nozzle.

2 Barb Nozzle (Fig 2)

Conventional sprue gate results in a sizeable blemish being left at the injection point. But sometimes this is undesirable. To reduce it to a minimum some form of pin gate is necessary. One method of achieving this form is by the use of a special nozzle in conjunction with a reverse-tapered sprue. This nozzle is variously termed as "barb nozzle or Italian nozzle". The barb nozzle is similar to the standard nozzle except that there is a projection at the front which incorporate barb.

Fig 2

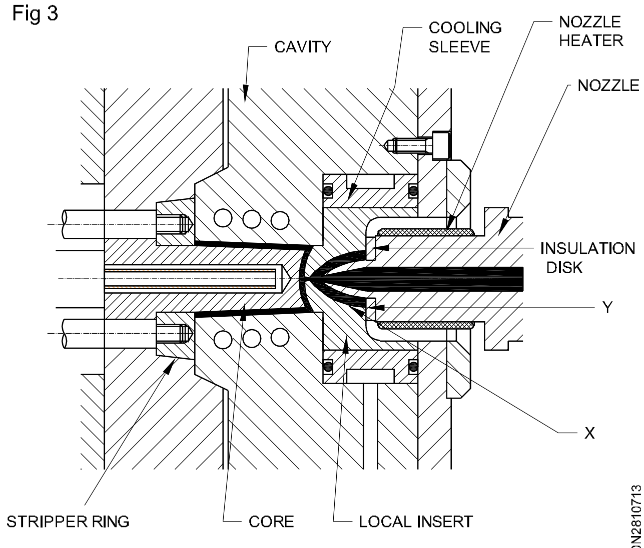


BARB NOZZLE

3 Ante-Chamber Design (Fig 3)

This is, also, known as the hot-well design. A small mass of material is retained in the ante-chamber. The plastic material adjacent to the mould wall partially insulates the central core of plastic material from the cold mould, allowing it to pass through the ante-chamber into the impression. As, it is not necessary to remove the sprue from the mould, fast moulding cycle can be achieved.

Fig 3

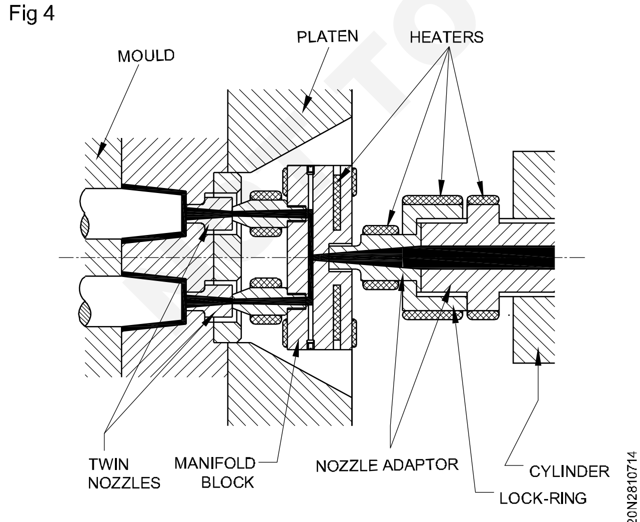


ANTI-CHAMBER DESIGN

4 Multi-nozzle manifold (Fig 4)

Multi-nozzle manifold is used for direct feeding of plastic material for multi impression mould. It consists of a manifold block suitably drilled to provide a flow path for the material in to which the required number of nozzles is fitted. The manifold block is fitted to the cylinder of the machine via nozzle adapter and a lock ring. This permits even filling of the impression but separate temperature control of each nozzle is necessary. Each impression will have its own sprue-bush. The barb nozzle or ante-chamber design can be used, if pin gating is required. The number of impression is limited to the number of nozzles which can be accommodated with in the aperture in the platen. In view of this and other limitations, this design is not commonly used.

Fig 4



MULTI-NOZZLE MANIFOLD

Hot runner mouldings system

Advantages

1 Reduce cycle time.

2 Eliminates the cold runner that would be scrap or require regrind.

3 Improves part consistency and quality.

4 Reduce gate mark

5 Reduce injection pressure since there is less pressure loss through out the system.

6 Can accommodate more number of cavities in limited space.

7 Sequential filling and family parts moulds are made possible by using valve gates.

8 Better finish on the component.

9 Very suitable for high volume production.

10 Low cost per component.

11 Better mechanical properties.

12 Increased process control for fine tuning mould and part.

13 Part need not be over packed hence less weight per part.

14 Thin wall section can be easily filled as the melt is at a higher temperature close to the impression than conventional mould.

15 It eliminates the need for a three plate mould for multi point back feeding.

Disadvantages

1 High mould cost.

2 Complicated setting required.

3 Frequent material change becomes difficult.

4 Color changing is difficult.

5 Maintenance cost is high.

6 Frequent power failure is a concern.

7 Part positioning is determined by the manifold.

8 The area of the moulding adjacent to the gate may be discolored with heat marks.

9 Material oozing out from gate may cause problems.

10 Some materials like PVC there are degradation problems.

Types of hot runner systems

The various types of hot runner systems are

1 Extended heater nozzle.

2 Thermal gate nozzles. (hot sprue bushings)

3 Hot runner manifolds.

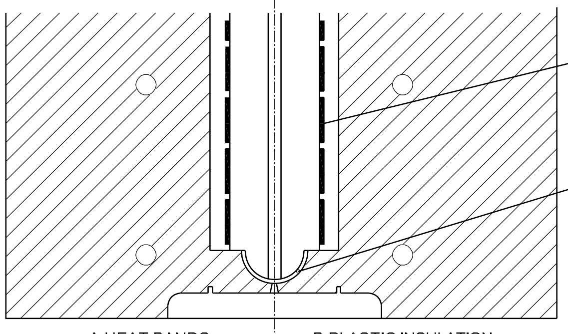
4 Insulated runner moulds

5 Modified insulated runner moulds.

6 Hot half.

Extended heater nozzle (Fig 5)

Fig 5



It is a special type of machine nozzle which extent into a pocket machined in to the mould plate.

The operation of this kind of mould is by maintaining the plastic material inside the machine nozzle at a sufficiently high temperature to prevent freezing, but, not sufficiently high to material oozing out into the cavity.

The nozzle heated by means of band heater controlled via thermocouple. To minimize, the transfer of heat from nozzle to the mould, a radial gap of minimum 6mm is maintained.

Thermal-gate nozzles

These heated bushings eliminate conventional cold sprue.

The benefits of using hot-sprue bushings are

1 Direct part gating eliminates the need to trim sprue and leaves no witness marks on the component.

2 Due to shorter injection path, larger shots faster fill and minimal gate mark, the part quality is improved.

3 A unique range of sizes copes with both large and small components.

4 Better temperature controllers.

5 Reduced cycle time due to more positive cooling.

6 Faster start- ups provided by positive temperature control at the gate area.

7 Easier installation and operation.

8 Saves material.

9 Saves energy.

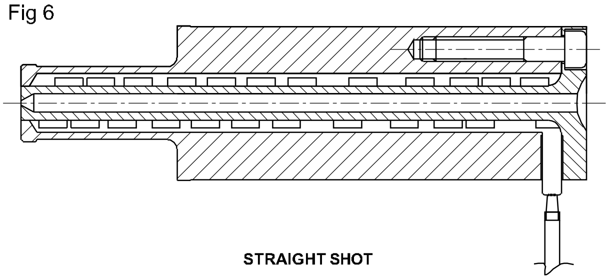
10 Low cost per gate.

Types of hot sprue bushings

Straight shots: (DME) (Fig 6)

The straight shot was designed to remove the unnecessary direct sprue on single cavity. The melt is guided through a straight hole into the cavity. This type of gate is used when a small gate is allowed and also a circular mark is accepted. It can also feed in to runner channel. This can be used for shot capacity upto 1000g.

Fig 6



Gate mate: (DME)

The gate mate incorporates a high thermal conductivity central channel in a steel housing. The melt channel is made of nickel plated beryllium copper for general use or conductive high wear resistant material for plastic with abrasive fillers.

Advantages of gate mates are

1 Improved part quality.

2 Minimal gate mark on the part.

3 Prevent direct contact between the nozzle and the moulded component.

4 Reduce part cycle time.

5 Reduce part stress.

6 Create more space for cooling.

7 Excellent for color and material change due to free metal flow.

8 Heat distribution is optimum.

9 Trouble free application.

Multiple Gate nozzles: (DME)

Multiple gate nozzles are most suitable for small parts in single cavity moulds providing upto 6 points of injection. It is located on a small circle self contained in a round manifold. Although designed for multiple gating of small parts in single cavity moulds, it can also be used for multiple point injection one large cavity where balancing of the melt flow is a problem. (Fig 7)

The advantages of multiple gate nozzles are

1 It eliminates runners often associated with the multiple injections of small parts.

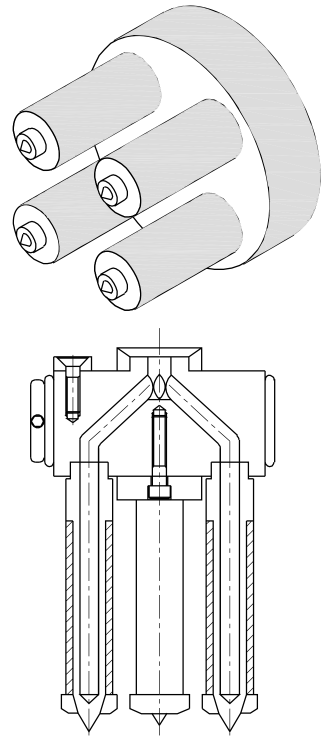
2 Heat distribution is uniform.

3 Multiple point injection in one large cavity.

4 Perfect balance of the melt flow.

5 Simplified mould design and construction.

Fig 7

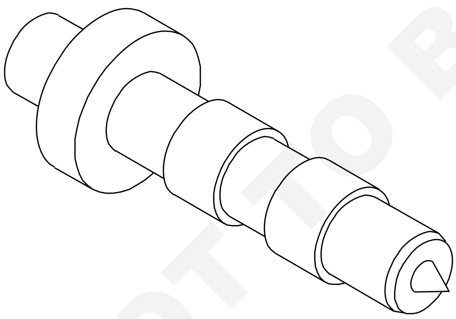


Auto nozzles: (DME) (Fig 8)

Auto nozzles are hot sprue bushings for single application to produce very large parts. The nozzle consists of a body and standard interchangeable tips.

The auto nozzle is heated by several band heaters which can be relocated depending on the length of nozzle and plastic material to be used.

Fig 8



The benefits of auto nozzle are

1 Direct part gating of very large and deep parts.

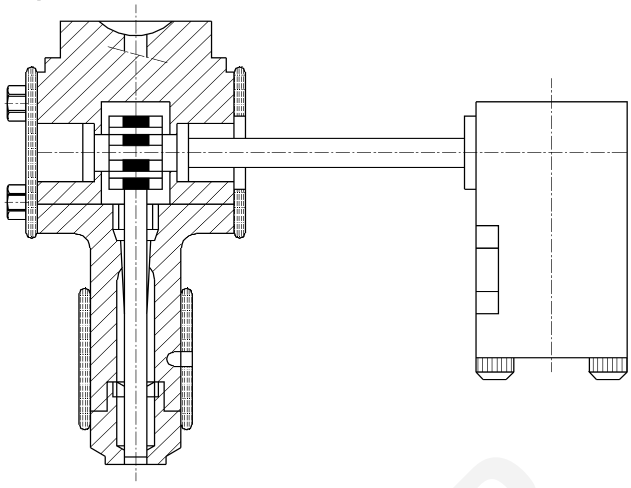
2 The loss of one heater band does not stop production.

3 Shot capacity upto 10000cm3 / sec with good flowing materials.

Valve gate nozzle (Fig 9)

With wide range of gate diameters, valve gate nozzle is suited for medium to very large moulded parts. The large melt flow channel allows the use of shear sensitive materials.

Fig 9



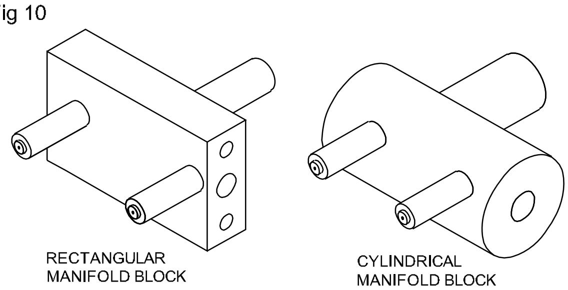
Hot-runner manifolds

This is the name given to a mould which contains a heated runner plate or block, insulated from the rest of the mould. This carries permanent molten flow-ways which directly feeds into the cavity.

Manifold (Fig 10)

The manifold can be a hot plate cut into various shapes, according to the runner layout or block, either in rectangle or round shape. In the plate or rectangular block design, they are heated by cartridge, coil or flat type heating elements fitted into suitably shaped holes or channel machined into it. In round type manifold, coil or band heaters are mainly fixed outside the rod.

Fig 10



Manifolds with Drops

These drops are secondary hot sprue bushings which provide a connecting path from the manifold to the cavity plate into the impression. So, a leak-free joint is maintained between the manifold and the Drop. Manifolds from single drop to 32 drops are standard. Manifolds with 64,128 and 256 drops are also, made. All manifolds are balanced, except, in some special cases.

Manifold lay-out options (Fig 11)

Insulated Runner Moulds (Fig 12)

This is similar to a conventional mould. The basic difference is the outer layer of the melt in the larger runner (13-25mm) solidifies and acts as an insulator for the central liquid core. There are no heating elements in this mould. The, only, heat is supplied by the conventional machine nozzle.

Fig 11

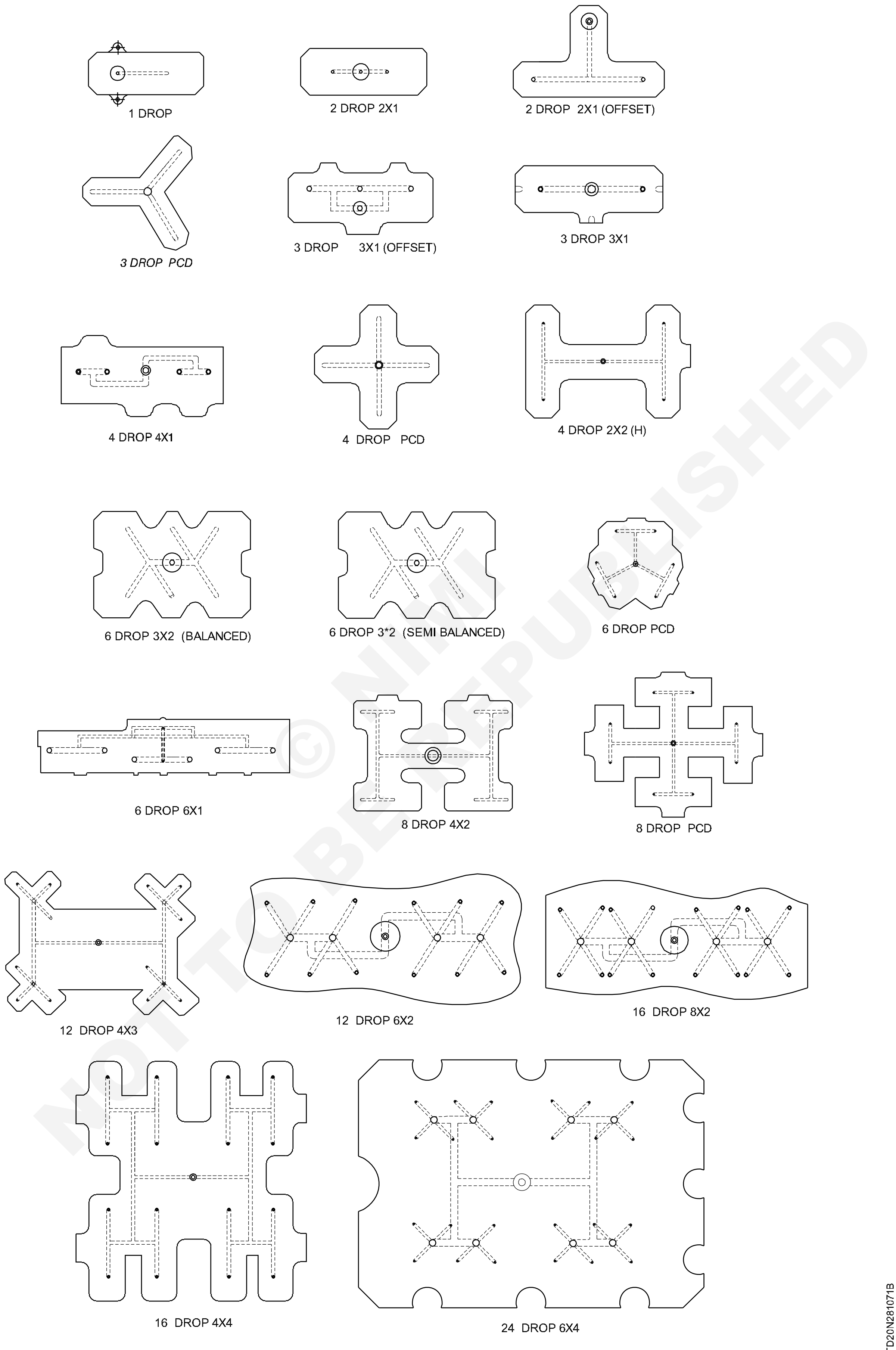
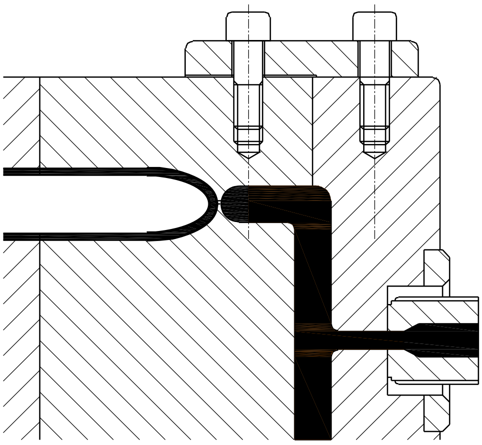

1 DROP
2 DROP 2X1
2 DROP 2X1 (OFFSET)
3 DROP PCD
3 DROP 3X1 (OFFSET)
3 DROP 3X1
4 DROP 4X1
4 DROP PCD
4 DROP 2X2 (H)
6 DROP 3X2 (BALANCED)
6 DROP 3*2 (SEMI BALANCED)
6 DROP PCD
6 DROP 6X1
8 DROP 4X2
8 DROP PCD
12 DROP 4X3
12 DROP 6X2
16 DROP 8X2
16 DROP 4X4
24 DROP 6X4
TD20N281071B

Fig 12



INSULATED RUNNER MOULD

Insulated runner moulds operate at a temperature below the melting point of the plastic being moulded. The melt flows through the large diameter runner machined in the butting surfaces of the cavity plate and the feed plate. The two plates are clamped together by quick swing clamps or top mounting screws to facilitate easy removal of the solidified feed system after production is over or after any interruption during regular production.

The material from the standard machine nozzle or extended nozzle enters into the large runner. As soon as the material in the large runner comes in contact with the cold mould the outer layer becomes solid. Since plastic is a poor conductor of heat the inside layers remain liquid. The inside material enters the cavity through a reverse sprue and gate. When the mould is opened the part is separated from the gate like a normal three plate mould.

Advantages and Disadvantages of Insulated Runner moulds

Advantages

Insulated runner mould is the cheapest among hot runner family.

This has got some of the advantages of the hot runner mould like material saving, less scrap etc.

Disadvantages

The chances of gate freeze off are more owing to the very low temperature around gate.

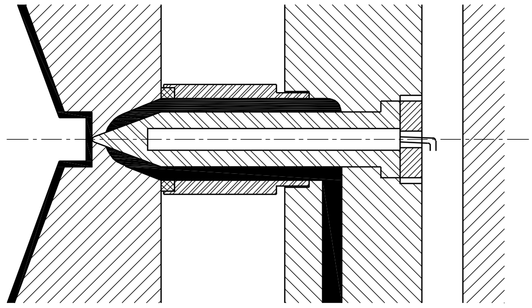
This is suitable only for polymer having low melting point and a wide injection temperature range.

The runner system works efficiently, only, with a faster cycle compared to a similar conventional mould. This is, because, the runner solidifies gradually towards the center due to higher heat from the stagnant melt between consecutive cycles.

Modified Insulated Runner Moulds (Fig 13)

This is a combination of the above two types in which the primary runners are insulated type and the secondary nozzle is heated by internal heaters to prevent melt freeze, around the gate.

Fig 13



The alternate combined flow way system available from D-M-E contains cylindrical shell of ID 9.52, 12.7 and 15.87mm with respective OD 16, 22.22 and 41.27mm fitted in a hole of diameter 24, 32 and 50mmrespectively. A cartridge heater with thermocouple is kept inside the shell ID. The material flows through a main hollow runner into a sub-hollow runner and into the secondary sprue and finally, into cavity. The secondary sprue contains a probe cartridge heater with thermo-couple. (Fig 14)

Advantages and Disadvantages of Modified Insulated Runner Mould

Advantages

1 Cost wise it is cheaper than hot runner moulds.

2 Gate Freeze-off and drooling is prevented by the special nozzle used.

3 By this method also almost all polymers can be moulded as in the case of Hot -Runner

Disadvantages

Here, the problem is to maintain a steady running condition such as proper temperature control of secondary nozzle, proper cooling etc.

Hot half

Hot-half consist of a complete hot runner system including manifold plate and back plate, allowing the mould maker time to make the rest of the mould. The hot half is fully wired and assembled.

The advantages of hot-half are

1 Reduced manufacturing time.

2 Hot-half allows for removal of cavity plate while mould is still in machine.

3 Hot-half construction allows for easy servicing of nozzles, tips, thermocouples and heaters.

Valve systems (Fig 15)

Valve Gated moulds are, generally, used for manufacturing in big components in a hot runner system. It can be used in association with other heating elements. The main advantage with valve gate is that the material can be pre-compressed by the machine and can be injected with high force and large quantity.

Fig 14

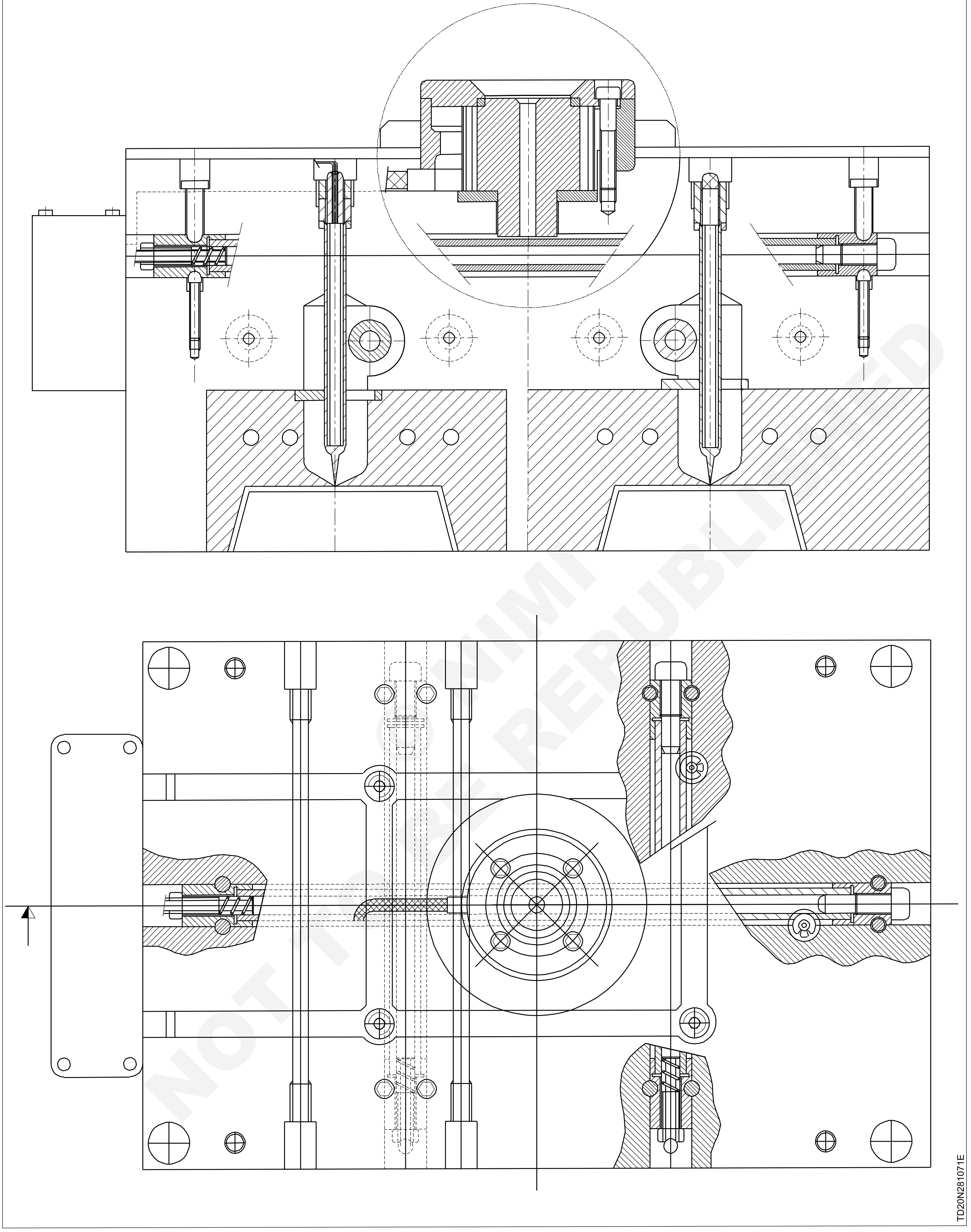
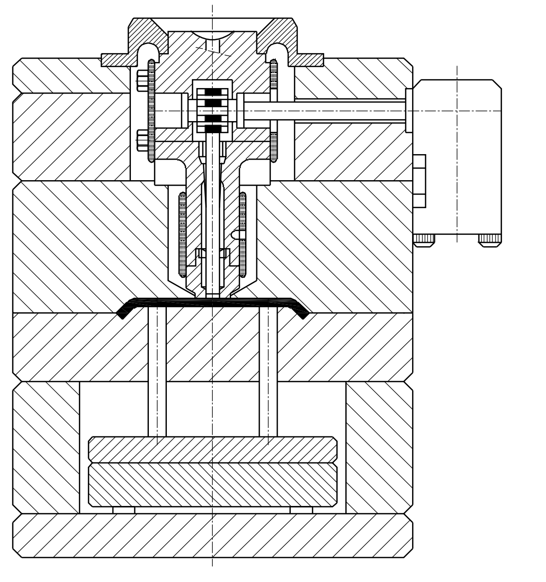


Fig 15



In conventional moulding, the material begins to flow as soon as the injection pressure builds up, whereas, in valve gate system, the filling is delayed, until, the full pressure is developed. The sudden opening of the valve gate allows rapid decompression of the material and the melt fills the cavity at a much faster rate.

The advantages of valve gated hot runner system are

1 Longer flow path possible.

2 Less injection pressure required.

3 Less part weight due to less injection pressure.

4 There is no loss of injection pressure.

5 The valve prevents drooling of material when the mould is open.

6 There is no suck back of material hence no holding pressure required.

7 No gate vestige except the pin mark.

8 Good for poor flow materials and reinforced materials.

9 A large gate can be used with the valve system hence fast filling of large components

Type of valve systems

There are two types of valve systems.

1 Automatic shut of valve (Fig 16)

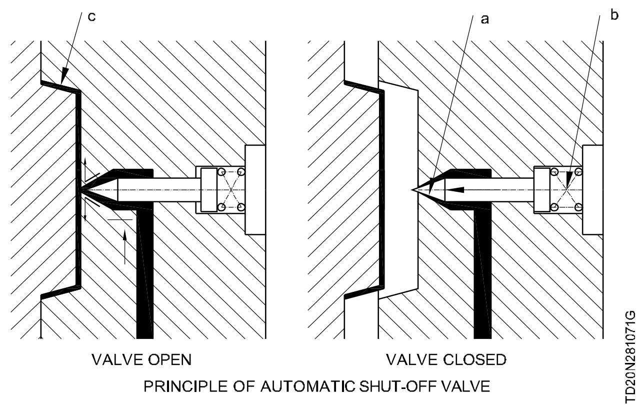
The automatic shut of system works with springs. The functions of the spring is to

1 Push the pin against valve seating thereby closing the gate.

2 To allow the pin to retract when the pressure in the system builds up.

3 To return the pin to the initial position when the pressure drops.

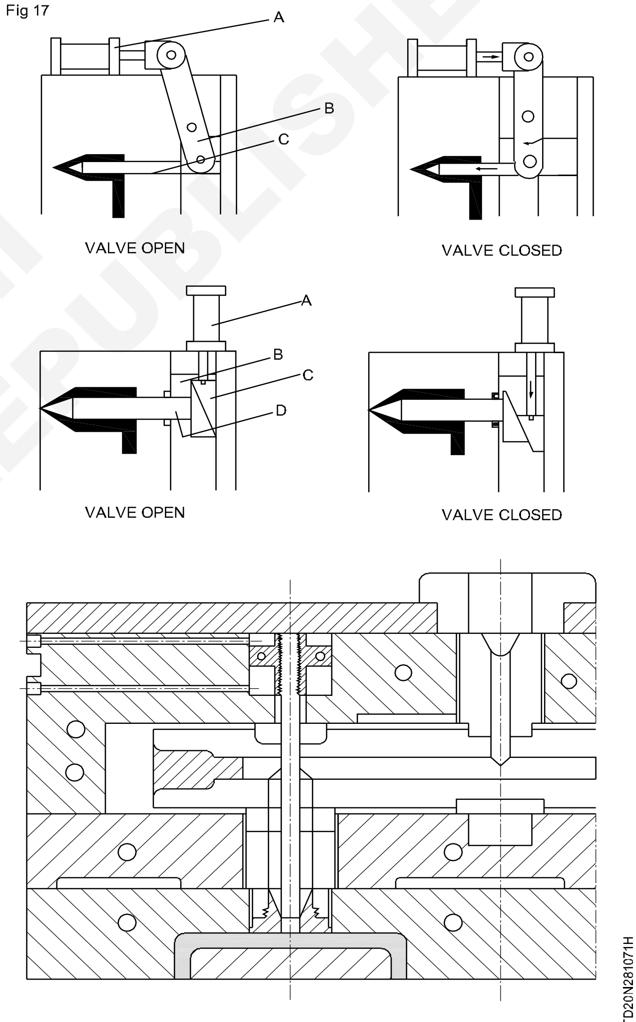
Fig 16



2 Externally operated valve (Fig 17)

This system uses an out-side actuating device which can be mechanical, hydraulic or pneumatic.

Fig 17



A needle valve is actuated when required. In the forward position, the front end of the needle valve closes the tapered opening of the gate. The valve is kept closed until, the injection plunger has moved forward compressing the melt. High pressure developed with in the injection cylinder and the hot runner unit. The needle valve then suddenly with drawn by the external actuator and the plastic melt enters the impression at high velocity. After the impression is fully filled the valve is closed allowing the screw to return. Then the process continues.

Selecting a hot runner system

When selecting a hot runner system the part wall thickness, part shape and the time required to fill the part are the first considerations. The second considerations are the gate selection, gate design, plastic material and the mould design.

Hot-runner moulds

As described above, it has got separate heated manifold insulated from the rest of the mould and operates at a temperature above the melting point of the resin being moulded. The manifold is usually heated by fire rods and temperature is accurately controlled by thermocouple according to the material used

This system is often successfully used with all polymers. The important parts to be studied in detail are (1) Manifold Block (2) Flow ways (Runner) (3) Secondary Nozzle etc.

Expansion & Fixing Problems

Since, the manifold block along operates at an elevated temperature of about 200 C, it expands in all three dimensions. Due to the heat transfer and expansion problem, the manifold block cannot be tied to the mould firmly.

Hence, it is fixed by a pin at the center and rotation is prevented by separate pin which leaves room for the expansion of the block. The manifold block is supported opposite each nozzle by pressure screws due to the enormous back pressure in the nozzle area. The expansion also affects the distance between the center of the nozzles. The variation being directly proportional to the center distance (e = L x C x dT) where e - expansion in mm L - Center distance in mm, C - soft of thermal expansion which is 13 x $10^{-6mm/mm/°C}$ for die steels and dT increase in temperature).

The center distance should be as small as possible or other compensating methods should be thought of as it offset the nozzle center from gate center.

Advantages and Disadvantages of Insulated Runner Mould

Insulated runner mould is the cheapest among hot runner family.

1 This has got some of the advantages of the hot runner mould like material saving, less scrap etc.

Disadvantages of insulated runner mould are as follows

1 The chance of gate freeze off is more owing to the very low temperature around gate.

2 This is suitable, only, for polymer having low melting point and a wide injection temperature range.

3 The runner system works efficiently only with a faster cycle compared to a similar conventional mould. This is because the runner solidifies gradually towards the center due to higher heat from the stagnant melt between consecutive cycles.

Advantages and Disadvantages of Modified Insulated Runner Mould

Advantages

1 Cost wise it is cheaper than hot runner moulds.

2 Gate Freeze off and dribble is prevented by the special nozzle used.

3 By this method also, almost all polymers can be moulded as in the case of Hot Runner.

Disadvantages

Compared to insulated runner mould the disadvantages are very less.

Here also the only problem is to maintain a steady running condition such as proper temperature control of secondary nozzle, proper cooling etc.

Starting or Restarting Nozzles in a Manifold Application

Use the following steps to start or restart a nozzle in a manifold application

1 Follow the wiring diagram of the system to ensure all cabling (power and thermocouple) are connected to the correct zones.

2 Turn on the manifold zones only and set the temperature to 100 degree Celsius. The temperature controller must be set on Soft Start mode to dry any moisture absorbed into the elements during transit.

3 Observe that all zones increase in temperature evenly and consistently.

4 Once all zones have reached 100 degree Celsius, wait 5 minutes for soak and then set to required melt temperature to +10 degree Celsius. (Machine barrels should be set at melt temperature +5 degree Celsius.)

5 While waiting for the manifold temperatures to reach set point, purge the machine with the appropriate plastic material in a natural state (preferably one with a lower melt than that to be used to make the parts). The stiffer natural material is used to fill the insulation gaps and is beneficial for faster colour changes. It is less effected by the continuous temperature and therefore reduces the build up of the colour pigments and or resident melt degradation.

6 Check machine settings. (For first startup, all speeds and pressures should be set low to limit any unforeseen damages.)

7 Maximum injection pressure set to 70MPa (avoid first shots from flashing tool)

8 Injection speed set at 30%

9 Injection volume set at 70% of expected shot weight(if part detail allows)

10 Injection pack pressure to 30MPa

11 Mould daylight correct set for tool

12 Mould close pressure safety set

13 Slow mould close set

14 Manifold should be approximately at 90% of the set temperature by now (20 minutes) turn on nozzles drop zones

15 The zones can be turned on in banks to more easily monitor rise in temperature

16 A similar method to heating the manifold should be followed with the first heating of the nozzles elements

17 Set to100 degree Celsius, controller must be set on soft start. (For drying any moisture absorbed into the elements during transit)

Observe that all zones increase in temperature evenly and consistently.

18 Once all zones have reached 100 degree Celsius , wait 5minutes for soak and then set to required melt temp + 10 degree Celsius (MX series) or +35 degree Celsius (MT series).

19 Nozzles will reach set point normally within 5 minutes.

20 Check all zones are at set point and stable.

21 Purge machine barrel to fresh material.

22 Hunt injection unit slowly to sprue bush.

23 Switch machine to semi automatic engage cycle

24 The moulding machine will cycle through hunt, inject, pack, screw, recharge, and then cooling.

25 Depending on the cycle time set, the machine can be stopped at this point opened and the process (11) re initiated.

26 Repeat this process 2 to 3 times before plastic appears at the gate as all the channels where empty.

27 Some moulding machines can manually inject into a manifold and so the filling of the manifold can be more easily observed.

28 Keep a careful eye on the cavities for the first sign of plastics, it will require clearing from the gates and cavity before the next shot is possible.

29 Cycle the machine fully obtaining a 70% shot.

30 Change to the correct material and colour.

31 Turning of the mould machine to production setting can now begin. Take care with the injection pressures and shot size as you can easily flash the tool causing damages.

Note: Often the nozzle will need to run hotter than barrel temperature to achieve a good result.

If mould is left idle and needs to be restarted, repeat the above procedure.

Note: Do not increase manifold or nozzle temperature by large amounts as increases of temperature above the design figures can damage the sealing faces of the manifold and nozzle due to excessive expansion.

Starting or Restarting a Nozzle in a Single Nozzle Application

1 Use the following steps to start or restart a nozzle in a single nozzle application:

2 Ensure "Soft Start" is selected on the temperature controller and the correct operating temperature is set.

3 Allow a minimum of 10 minutes for the nozzle to heat up to operating temperature.

4 Purge machine barrel before connecting to the nozzle

5 Slowly bring machine nozzle up to hot nozzle to avoid damage.

6 When nozzle is at the correct temperature, inject the plastic into the mould.

7 Check material comes out the gate and correct if needed.

8 Adjust nozzle temperature to get suitable moulding.

Note: Often the nozzle will need to run hotter than barrel temperature to achieve good results.

If the machine is left idle, it is strongly recommended to gently purge the first shot through the mastip nozzle. This will clear any cool slug that may have formed near the head.

For a single cavity heated sprue bush or nozzle, check that the set temperature is reached and the plastic is molten at the head of the nozzle. If the plastic is not completely molten, remove the cold slug from the bushing before injection.

Maintaining a Hot Runner System

Recommended procedure for Colour Change

1 Make the first shot in the mould in natural material.

2 Increase machine barrel temperatures by 20ºC.

3 Increase mould temperatures by 15ºC by either turning off the water or if using a temperature control unit.

4 Increase manifold temperatures by 15 to 20ºC.

5 Increase nozzle temperature by 25 to 30ºC.

6 Retract nozzle and purge machine barrel using a low melt flow (MFI) natural polypropylene until clear.

7 Restart normal cycle with the low MFI polypropylene.

8 IMPORTANT: Some components may not be able to be moulded in a low MFI PP. If this is the case try using a mixture of say 50:50 of the existing MFI and a low MFI PP.

9 When mouldings are clean, purge machine and entire hot runner system with the natural material that is being used for the production run.

10 When satisfied that natural material moulded parts are OK reduce all temperatures to the required settings and add new color. Temperatures should be reduced in the reverse order.

11 Proceed with full production run.

12 Replacing a Heater and Thermocouple on a Single Nozzle Application

13 You can only use MT range nozzles in a single nozzle application. To replace the heater or thermocouple, you must remove the MT range nozzle from the machine.

14 Remove nozzle from the mould. Do not damage the heater or thermocouple wires.

15 Remove circlip and heater cover.

16 Grip the head and remove heater by turning the tip end of the heater in a clockwise direction to "unwind" or loosen the heater coils, and at the same time pull the heater off the body.

17 Remove the thermocouple.

18 Check the resistance of the thermocouple with a multi meter, the resistance should be 10 ohms (or less).

19 When replacing the thermocouple you need to bend the end.

20 To replace heater, push heater as far as it will go onto the nozzle body, with bottom heater connection in line with slot on body. Turn the tip end of the heater in a clockwise direction to "unwind" or loosen the heater coils as you continue to push the heater towards the head of the nozzle. Make sure heater is fully forward on body.

21 Refit the heater cover. If the cover is tight check that the heater is not partially unwound.

22 Refit the circlip.

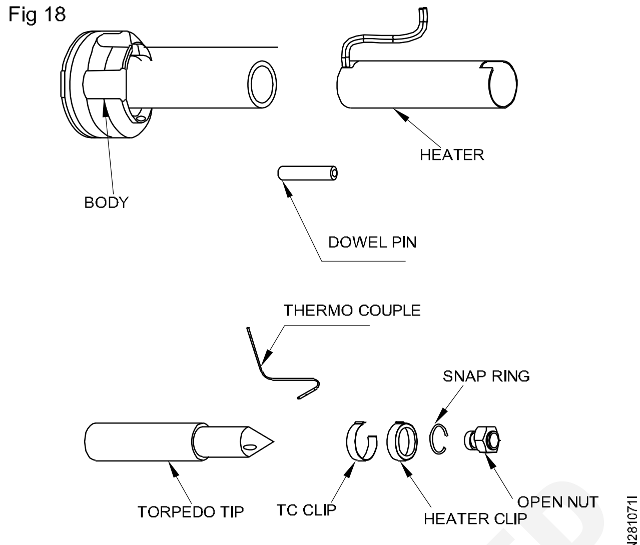
23 Recheck the thermocouple resistance as per step 5 above.

24 Refit nozzle into the nozzle cavity in the mould, taking care not to crush or excessively bend the heater or thermocouple wires.

Parts of the MX Range Nozzle (Fig 18)

The MX range nozzle can be configured for front loading or rear loading in a Hot Half or standard manifold. If the nozzle is configured for front loading, you need to only remove the snap ring and heater cap to access the thermocouple and heater.

Fig 18



PARTS OF MX RANGE

Removing and Installing a Snap Ring on a MX Range Nozzle

1 The snap ring is designed to fasten the heater cap in place during the operation of the nozzle. If the snap ring is not installed correctly the heater cap may become loose during operation of the nozzle causing incorrect readings of the thermocouple, poor performance of the nozzle, and possible failure of the heater.

To remove the snap ring

1 If the nozzle is not assembled in a Hot-Half it should be held securely to restrict its movement.

2 Locate the ends of the Snap Ring.

3 Use a strong thin tool (such as a blade) to wedge between the end of the Snap Ring and the heater cap.

4 Twist the end of the tool towards the end of the nut so that the end of the Snap Ring rides up out of the groove and over the nut shoulder.

5 Slowly slide the tool around the Snap Ring, moving the rest of the Snap Ring out of the groove and over the nut shoulder.

6 Repeat step 4 and 5 until the Snap Ring is fully removed from the nut.

Moulding defects and remedies

Objectives: At the end of this lesson you shall be able to
- **state the main factors causing defects**
- **list out the various defects in moulding**
- **brief the various causes and remedies for the defects.**

Some main factors causing defects

There are four different factors causing the defects. They are

1 Moulding Machine

a Operating condition

i Temperature, Pressure

ii Injection speed & time

2 Mould

3 Material

a Design of product

4 Management

The analysis of most molding problems focuses on the molding cycle. The molding cycle can best be described by what happens to the polymer in terms of

1 Fill time

2 Packing time/rate

3 Cooling time

4 Ejection time

5 Open time

6 Mold temperature

7 Sprue and runner design

8 Gate size and location

9 Section thickness

10 Length of flow path

Various defects

1. Sink Marks, 2. Streaks, 3. Blistering, 4. Weld Lines, 5. Gloss difference, 6. Jetting, 7. Record grooves effect, 8. Dull spots near gate area, 9. Incompletely filled part (Short shots), 10. Diesel effect, 11. Over sprayed parts (Flashes), 12. Stress whitening/Cracking, 13. Ejector marks, 14. Deformation during demoulding, 15. Ejection Grooves, 16. Flaking of surface layer), 17. Cold slug, 18. Filamentation , 19. Dark spots, 20. Plate outs, 21. Defects on electroplated parts, 22. Broken/Ragged film hinge, 23. Warpage, 24. Fish Eye, 25. Hesitation, 26. Over packing

Various causes and remedies for the defect

1 Sink marks (Fig 1&2)

Sink marks are the localized contractions or the depressions on the surface of the moldings

Causes

- Sink marks occur during the cooling process if certain areas of the part are not cooled sufficiently causing them to contract
- If these contractions are not compensated and the outside wall is not stable enough due to insufficient cooling, depressions occur

Fig 1

Fig 2

Casual Factors for Sink marks

1 Insufficient plastic in the mould to compensate for shrinkage due to:

- Thick sections, bosses, ribs
- Injection pressure too low
- Unbalanced gate
- Injection speed too low

2 Plastic too hot

3 Article ejected too hot due to insufficient cooling

4 Gate located at thin section

Possible solutions for sink marks

1 Relocate gates on or as near as possible to thick sections

2 Increase cooling time

3 If possible change the mold design to maintain an even wall thickness throughout the part

4 Increase injection speed & mold temperature. Do this if the sink marks are away from the gate or in thin walled areas

5 Dry the material

2 Streaks (Fig 3&4)

Streaks are the prominent stretchy marks on the surface of the moldings

There are different streaks appear on the component

- Charred streaks
- Moisture streaks
- Air streaks
- Colored streaks
- Glass fiber streaks

Fig 3

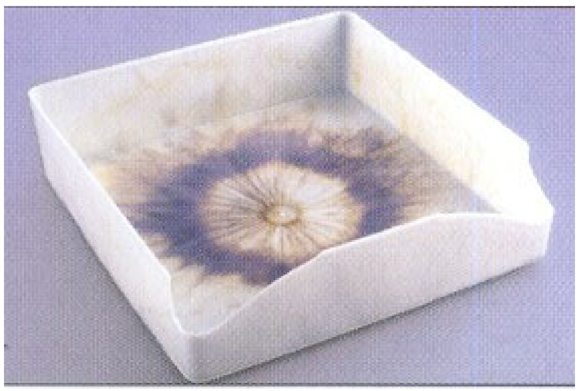


Fig 4

Causal factors for streaks

1 Temperature too high or residence time too long during pre drying

2 Melt temperature too high

3 Residence time in plasticizing unit too long

4 Insufficient pre-drying of the granules

5 Wrong storage of the material

6 Temperature at flange too low

7 Poor venting

Possible solutions for streaks

1 If melt temperature is above processing range reduce cylinder temperature,Screw speed, dynamic pressure

2 If the residence time is in critical range reduce the cycle time.

3 If burnt marks appear near gate decrease injection rate avoid sharp corners, edges in gate

4 Material to be sufficiently pre dried

5 Check material material storage and its packaging

6 Reduce residence time in material hopper

7 Reduce injection rate

8 Round sharp edges of transitions.

9 Reduce depth of engraving

10 Provide for venting

11 Use smaller colour pigments

12 Check solubility of the used die

13 Use shorter glass fiber

3 Blisters/bubbles (Fig 5&6)

This is hollows created on the part or in the molded part

Causal factors for blisters

1 Decompression is excessively high or fast

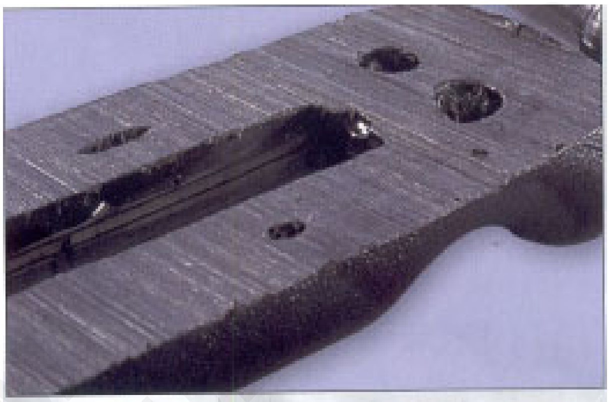
2 Insufficient dynamic pressure

3 Venting problems inside the mould

4 Regrind use

Fig 5

Fig 6



Possible solutions for blisters

1 Decrease melt temperature

2 Decrease screw speed

3 Dry the material

4 Increase back pressure

5 Increase mold temperature

6 Ensure regrind is not too coarse

7 Provide additional mold vents

8 Relocate gate

4 Weld line (Fig 7&8)

Weld Lines are created when two or more melt flow fronts meet possibly causing a cosmetically visible line. It can also create a weakened area in the finished molded part especially with filled resins

Causal factors of weld line

1 Low melt temperature

2 Injection pressure too low

3 Excessive mould lubricant on the mould

4 Improper venting in the mould

5 Low mould temperature

6 Injection speed very slow

7 Shift in mould resulting in one wall being too thin

8 Poor material flow properties

Fig 7

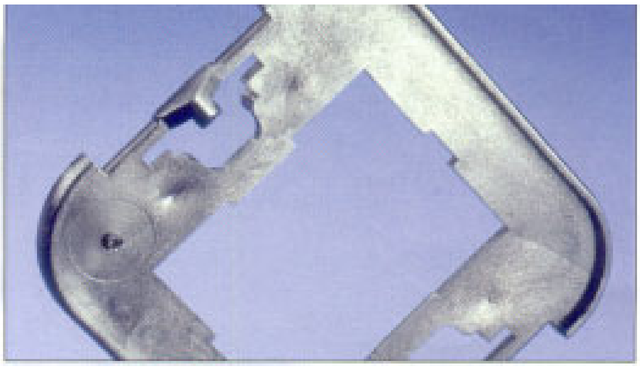
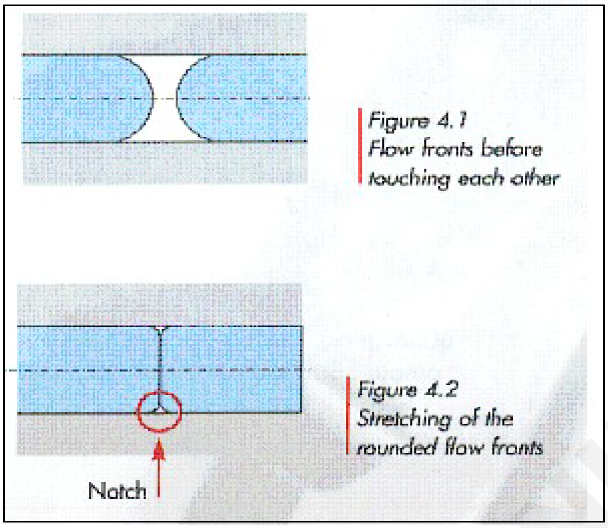


Fig 8



Possible solutions for weld line

1 Increase injection pressure
2 Increase injection speed
3 Increase injection hold
4 Increase melt & mold temperature
5 Make sure part contains no sharp variation in cross-sections
6 Vent cavity in the weld area
7 Move gate (Move weld line to invisible area)
8 Add over flows

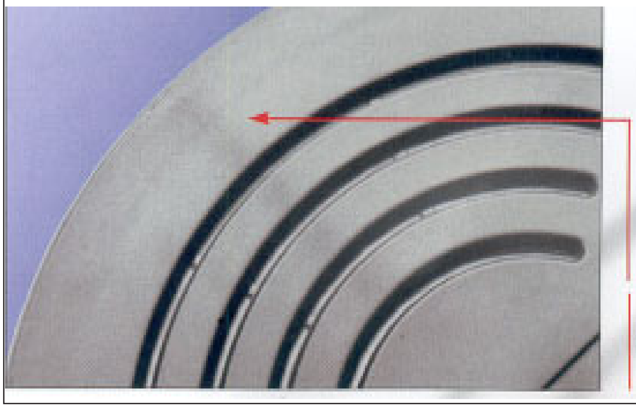
5 Gloss/gloss differences (Fig 9)

The gloss of a mould is the reflecting property of its surface when exposed to incident light. When a ray of light hits the surface its direction will change(Refraction).The smoother the surface of molding the smaller the scattering angle of reflecting ray and rougher the surface the larger the angle

Causal factors for gloss differences

1 Different cooling conditions
2 Shrinkage differences
3 Stretching of already cooled areas e.g. due to warpage
4 Molds that are textured or resins that are filled
5 Poor polished mold surfaces

Fig 9



Possible solutions for gloss differences

1 Clean mold surface
2 If the part design allows increase the polish of the molding surface
3 Increase melt temperature
4 Make sure venting is adequate

6 Jetting (Fig 10)

The prominent inhomogeneous snake .like strands on the surface of molding. Jetting originating at the gate, spreading over entire part

Causal factors for jetting

1 When cross section of the molded part is increased rapidly in conjunction with high injection speeds
2 Wrong position of the mold where melt fills the cavity from top to bottom

Fig 10

Possible solutions for jetting

1 Decrease injection speed
2 Change the melt temperature, up or down
3 Increase the gate diameter
4 Move the gate so that when the plastic first enters the cavity it hits an obstruction such as a rib or wall

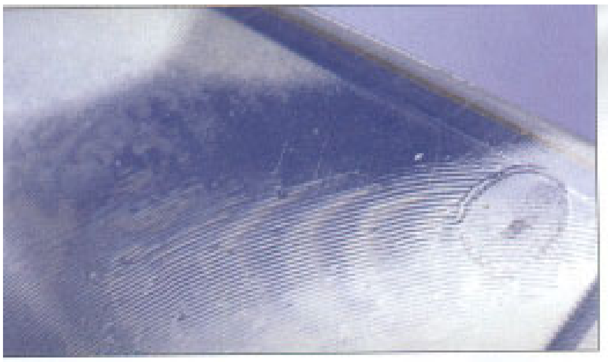
7 Record grooves effect (Fig 11)

Very fine grooves on the surface of the moulded part

Causal factors for record grooves effect

1 Insufficient mould & melt temperatures
2 Injection speed

Fig 11



Possible solutions for record grooves effect

1 Increase mould temperature

2 Increase melt temperature.

3 Avoid small runners

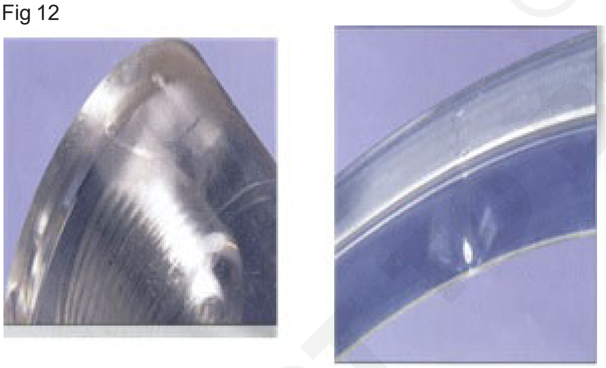
4 Increase injection rate

5 Increase maximum injection pressure

8 Dull spots in gate area (Fig 12)

High injection rates and small gate cross sections and bye passes create strong orientations of the molecules during injections. Since there is not enough time for the relaxation directly behind the gate the peripheral layers are frozen while they are still strongly oriented. Such peripheral layers can be stressed with a very minimal degree and crack under the slightest impact of high shear stress. The hot melt inside flows towards the wall and forms extremely fine notches. Dullness is due to widely spread reflections in this area

Fig 12



Possible solutions for dull spots in gate area

1 Reduce injection rate

2 Increase gate diameter

3 Round of transition from gate to cavity

4 Move the position of gate to unimportant area

9 Incompletely filled part (short shots) (Fig 13)

Failure to fill the mould or cavities of the mould completely is termed as a short shot

Causal factors for short shots

1 Injected compound volume is too small (shot volume)

2 Poor venting

3 Injection pressure or rate is insufficient

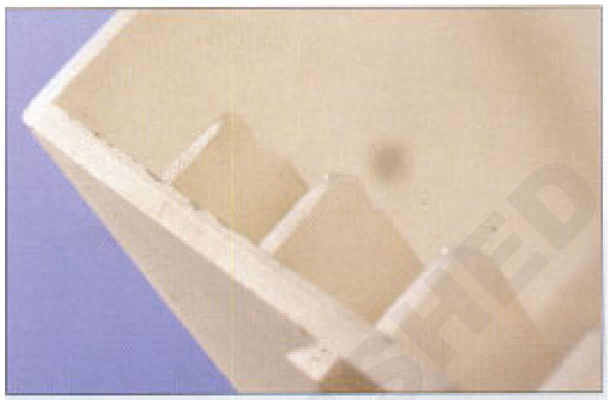
4 Wrong temperature control

5 Thin cross sections. Where in which the melt freezes pre maturely due to unfavorable flow conditions

6 improper balance of plastic flow in multiple cavity moulds

7 Insufficient gate opening

Fig 13



Possible solutions for short shots

1 Increase injection pressure, speed

2 Increase melt and mold temperature

3 Increase nozzle temperature. Ensure that the manifold and nozzles have reached the set temperature

4 Increase shot size and confirm cushion

5 Make sure mould is vented correctly and vents are clear.

6 Confirm that the non-return valve used is not leaking excessively

7 Change part design. Thin areas of the mould may not fill completely, especially if there is a thick to thin transition, or there is a long rib that cannot be vented very well. If the part design allows it, change in these areas can improve the situation

10 Diesel effect (burn mark) (Fig 14&15)

Burn Marks or Dieseling show up on the finish molded parts as charred or dark plastic caused by trapped gas and is usually accompanied by a distinctive burnt smell.

Fig 14

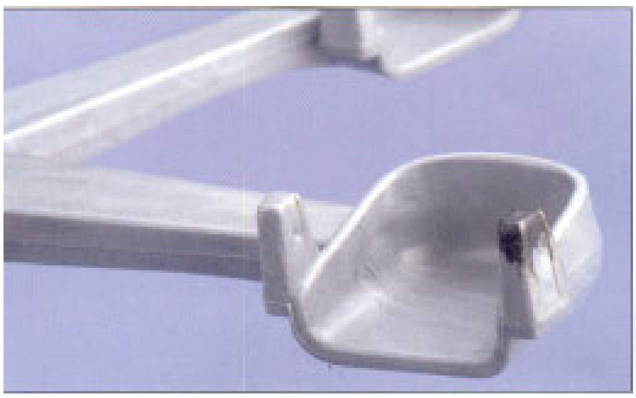


Fig 15

Causes of diesel effect (burn mark)

- This is purely a venting problem, the trapped gas is compressed and gets heated to a high degree causing the burn on the plastic.
- It can occur at blind holes, fillets,narrow ribs,near places where several flow fronts meet, end of flow paths etc

Possible solutions for diesel effect (burn mark)

1 Alter gate position and/or increase gate size
2 Decrease injection pressure
3 Decrease injection speed
4 Decrease melt and/or mold temperature
5 Improve mold cavity venting. Vents may become smaller over time due to wear and they will need to be brought back to their original depth

11 Flashes (Fig 16)

Flashes are often seen near sealing faces, out of vent grooves, or down ejector pins. It appears as thin or sometimes thick sections of plastic where it would not be on a normal part

Causal factors for flashes

1 Permissible gap widths exceeded,due to insufficient mold tightness, exceeding production tolerances,damaged sealing faces
2 Clamping force too high or too low
3 Internal mould pressure too high
4 Insufficient viscosity of melt

Fig 16

Possible solutions for flashes

1 Increase clamp pressure
2 Check mold venting. Vents may have been ground too deep for the material being used
3 Check sealing surfaces to ensure that they seal off properly by "blueing" them in under clamp tonnage
4 Check ejector pin bore diameter to pin diameter tolerances. The tolerances may be too large allowing plastic to flash down the opening. The tolerances may be too large for the material being used and can occur due to wear over time.

12 Stress whitening (Fig 17)

Cracking or Crazing on a part is called Stress whitening.

Causal factors for stress whitening

Cracking or Crazing is caused by high internal moulded in stress or by an external force imposed upon the part. They can also be caused by an incompatible external chemical being applied to the finished parts The cracks often don't appear until days or weeks after the parts have been molded.

Fig 17

Possible solutions for stress whitening

1 Decrease injection pressure
2 Dry material
3 Increase cylinder temperature
4 Increase mold temperature
5 Increase nozzle temperature
6 Modify injection speed
7 If the material is partially crystalline then it may help to reduce the mold and/or melt temperature
8 If the material is amorphous then it may help to increase the mold and/or melt temperature

13 Ejector marks (Fig 18)

Ejector marks are the depressions or the elevations causing abrupt changes in the wall thickness these may also cause punctures of the surface of part

Causes for ejector marks

- Wrong dimensioning or the design of mould
- High temperature differences within the mould or between mould wall and ejector
- Premature demoulding or high demoulding forces
- Wrong fittings or ejector lengths

Fig 18

Possible solutions for ejector marks

1 Check counter bores and contact surfaces of the ejector heads

2 Refit ejector

3 Change ejector size accordingly

4 Increase cooling time

5 Change ejector system

6 Check draft on core

7 Reduce demoulding force if it is high, reduce holding pressure

14 Deformation during demoulding (Fig 19)

A residual pressure builds up inside the moulded part which is suddenly released during demoulding.It may be impossible to demould the part.

Fig 19

Possible solutions for deformation during demoulding

1 Reduce holding pressure

2 Reduce holding pressure time

3 Increase cooling time

4 Check ejector dimensions

5 Use antifriction layer to reduce demoulding forces

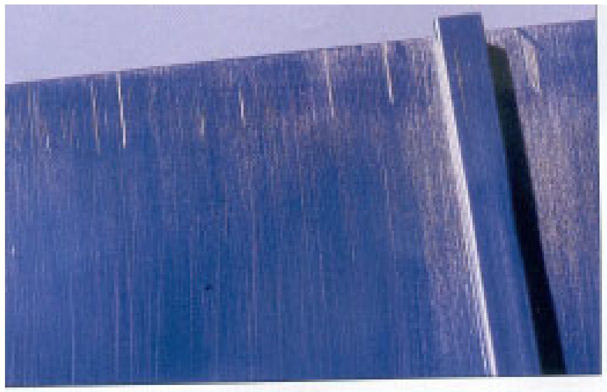
6 Adopt direction of polishing to direction of demoulding

7 Use mould release agent

15 Ejection grooves (Fig 20)

Ejection grooves are produced during demoulding and constitute damage to the surface of the molded part.They mainly occur on the structured surfaces.

Fig 20



Causes for ejection grooves

- Less draft.
- More surface roughness.
- Mold deformation due to excessive cavity pressure.
- Mold is not stiff.

Possible solutions for ejection grooves

- Increase cooling time
- Increase wall thickness
- Increase drafts
- Use lower surface roughness
- Increase mould stiffness

16 Flaking of surface layer (Fig 21)

Delamination occurs when single surface layers start flaking off the moulded part due to insufficient bondage between adjacent layers

Fig 21

Possible solutions for flaking of surface layer

1 Check for material contamination. Incompatible resins or colorants may have been accidentally mixed causing this condition to be seen

2 Dry material

3 Increase melt temperature

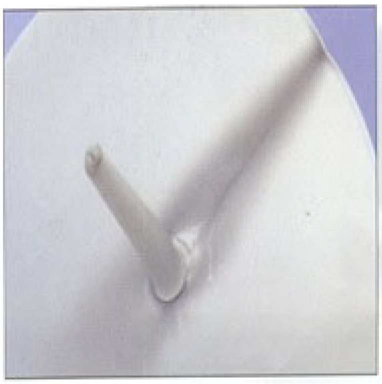
4 Increase mold temperature

5 Insufficient Blending. Check melt homogeneity and plasticizing performance

17 Cold slug (Fig 22)

Cold slugs are formed when melt solidifies in the gate or in the nozzle before the cavity is filled

Fig 22



Causes for cold slug

- Tip of the nozzle too cold
- Nozzle too long or unsuitable heater band

Possible solutions for cold slug

- Check nozzle design
- Check nozzle temperature regulation system
- Increase nozzle temperature
- Move injection unit backwards

18 Filamentation (Fig 23)

Due to insufficient cooling of the gate or sprue area, the part does not break cleanly from the gate area. The shape may range from a short pointed cone to a filament of several centimeters. The premature retracting can crack the layer which is still in fluid state in to a form of filament.

Fig 23

Possible solutions for filamentation

1 Provide sufficient cooling time during the cycle.

2 Excessive heat in the gate area. Check thermocouple in the nozzle or decrease the temperature of the hot runner manifold and nozzle.

3 Increase cooling at the gate area. Ensure that you have controllable turbulent flow in the gate area.

4 Reduce melt temperature.

5 Increase holding pressure time

19 Dark spots (Fig 24)

Dark spots are similar to burn marks or brown streaks but generally not as dark or severe. It may cause the part to be a darker shade than the virgin pellets and is often found nearest the gate area, however it can also appear as dark streaks throughout the part

Fig 24

Causes for dark spots

Tool Wear, thermal damage, dirt

Possible solutions for dark spots

1 Check hopper and feed zone for contamination

2 Decrease back pressure.

3 Decrease melt temperature.

4 Decrease nozzle temperature.

5 Move mold to smaller shot-size press.

6 Provide additional vents in mold.

7 Shorten overall cycle

20 Plate out (Fig 25)

Plate out is a surface defect on the mould surface e.g on inserts which eventually causes production of defective parts. It usually occurs in materials such as POM,PP,PET, ABS,PC, PE Materials with additives such as flame retardants, lubricants and colorants are also subject to plate out

Fig 25

Causes for plate out

- Wrong or excessive use of lubricants
- In compatibility of colorant ,lubricant and base material.
- Drying times too long.
- High material stress, such as excessive melt temperatures, shear stress, shear rates.

Possible solutions for plate out

- Use suitable lubricant

- Check metering of lubricant.
- Use antifriction layers for moving mould elements.
- Check compatibility of base material with additives/ colorants/lubricant.
- Check mixing ratio.
- Reduce temperature
- Ensure sufficient venting.
- Eliminate moisture

21 Defects on electroplated parts (Fig 26)

Defects on electroplated parts (pimples,blisters,insufficient layer adherence). Almost all defects on injection molded parts are also visible on electroplated parts.These defects look more prominent than on part without plating

Fig 26

Possible solutions defects on electroplated parts

1 Reduce injection rate
2 Increase mould wall temperature when using amorphous materials.
3 Increase melt temperature.
4 Avoid wall thickness variations.
5 Avoid deformation during transportations
6 Avoid internal stresses.
7 Do not use releasing agent.

22 Broken/ ragged film hinge (Fig 27)

Film hinges are moving and permanent joints. Their function is based on the elastic properties of the material. The failure of film hinges is mainly caused by overly stressed plastic. The film hinge may break partially or completely. Over stressing can also result in whitening

Fig 27

Possible solutions for broken/ ragged film hinge

If the film not filled

1 Increase melt temperature.
2 Increase injection rate.
3 Increase mold wall temperature.
4 Increase wall thickness of film hinge.
5 Use easier flow material
6 Move position of gating away from hinge.

If too much force required to operate hinge.

1 Reduce mold wall thickness
2 Use material of low modulus of elasticity.
3 Check hinge design

If breaks immediately or after few uses.

1 Remove weld line from film hinge area.
2 Reduce mold wall temperature.
3 Use material of higher viscosity
4 Ensure parallel flow of flow front

23 Warpage (Fig 28)

Warpage is the deviation of the mould part from its required shape.Warping, Part Distortion is shown up as parts being bowed, warped, bent or twisted beyond the normal specification outlined on the drawing

Fig 28

Causal factors for warpage

- Article ejected too hot.
- Variation in section thickness or contours of the screw.
- Excessive area discharged or packed into the area around the gate.
- Non-uniform mould temperature due to improper positioning if the cooling channels in the mould.
- Excessive feed
- Injection pressure too high.
- Insufficient cooling time.
- Poorly designed or operated cooling system.

- Unbalanced gates on articles with more than one gate.
- Holding time is more.

Possible solutions for warpage

1 Adjust melt Temperature (increase to relieve molded-in stress, decrease to avoid over packing). stress, decrease to avoid over packing). stress, decrease to avoid over packing).

2 Check gates for proper location and adequate size.

3 Check mold knockout mechanism for proper design and operation.

4 Equalize/balance mold temperature of both halves.

Increase injection-hold.

5 Increase mold cooling time.

24 Fish eyes (Fig 29)

Fish eyes are a surface defect that results from un melted material being pushed with the melt stream into the cavity and appearing on the surface of a molded part

Fig 29

Possible solutions for fish eyes

1 Reduce regrind material

Contact material suppliers to get the recommended levels of regrind to use.

2. Optimize melt temperature

3 Modify screw design

Contact material suppliers to get the right screw design information to avoid improper melt mix or overheating that leads to material degradation.

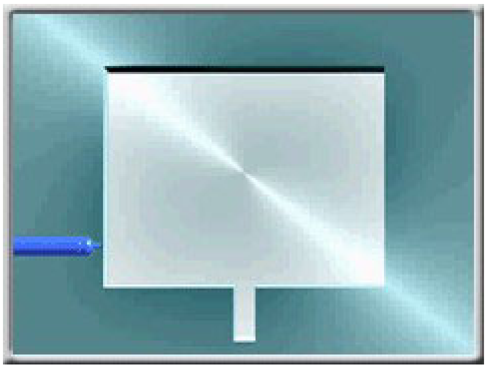
25 Hesitation (Fig 30)

- Hesitation is when flow slows down or stops along a particular flow path.
- If plastic filling a cavity has the option of filling either a thin section or a thick section, the plastic will tend to fill the thick section first as this route offers less resistance to flow.
- Hesitation can occur in ribs and in thin section of parts that have significant changes in wall thickness.

Possible solutions for hesitation

1 Move the polymer injection location away from the area of hesitation so that the bulk of the cavity fills before the melt reaches the thin area.

Fig 30



2 Move the polymer injection location to a place that will cause greater pressure to be applied where the hesitation occurred

3 It is useful to have thin ribs/bosses as the last point to fill

4 Increase the wall thickness where the hesitation occurr ed, to reduce the resistance to flow.

5 Use a less viscous material

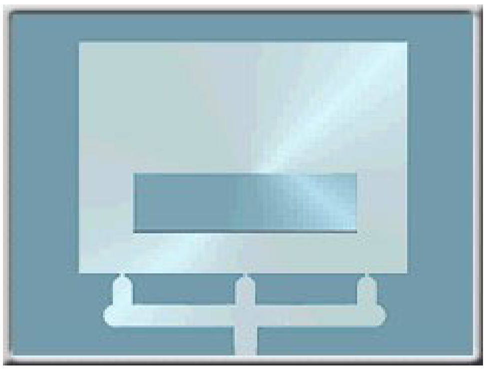
6 Inject more quickly

7 Increase the melt temp

26 Overpacking (Fig 31)

Over packing is when extra material is compressed in one flow path while other flow paths are still filling

Fig 31



Possible solutions for overpacking

1 Thicken or thin parts of the model to act as flow leaders or deflectors.

2 Move the injection location to a position that will define similar length flow paths.

3 Divide the cavity into imaginary sections, and use one injection location for each section.

4 Remove unnecessary gates.

6 Relocate gates on or as near as possible to thick sections.

Try increasing or decreasing injection pressure

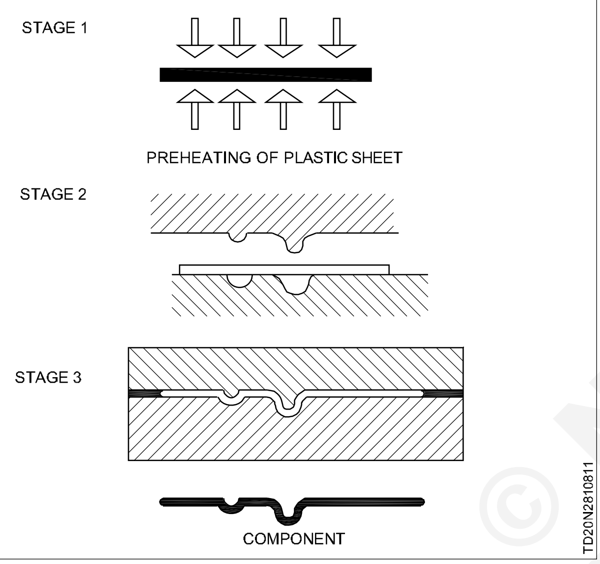
Alternative to injection mould

Objectives: At the end of this lesson you shall be able to
- **state what is thermo-forming**
- **brief the sheet and film forming process**
- **explain the process of multi colour moulding**
- **brief the process of multi material moulding.**

Thermos-forming (Fig 1)

Thermos-forming is, almost same, as blow moulding. It does not have closed cavities and no parison is used.

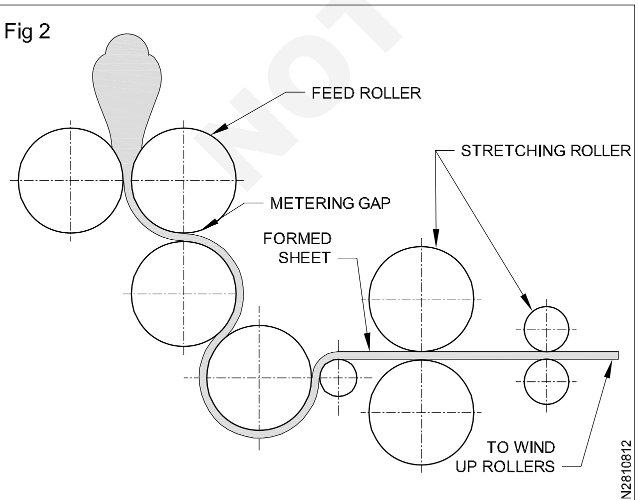
Fig 1



Thermo forming processes make use of a plastic sheet which is kept over an open mould and heated. It is, then, forced in to the cavity by means of forming punch, air pressure or vacuum. The air is extracted through a number of small holes in cavity. Sometimes, it is pushed from top by a forming punch and also, pulled from bottom by vacuum. The part is cooled and then, ejected.

Sheet and film-forming (Calendaring) (Fig 2)

Fig 2



THE PROCESS OF SHEET ROLLING

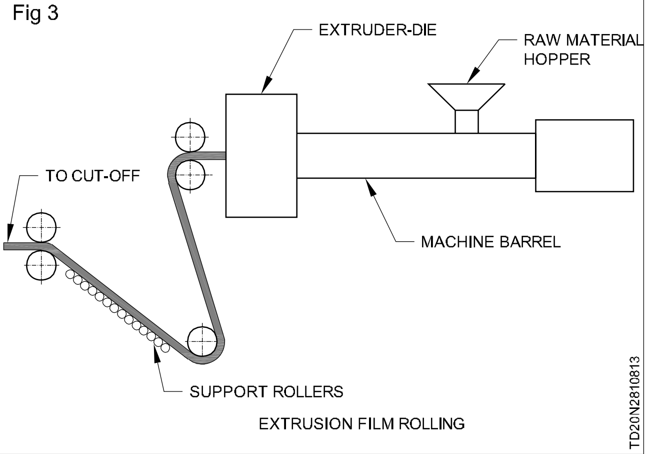
There are many methods of making plastic sheets and films. Sheet with less than 0.1 mm thickness is known as film. One exclusive method of producing sheet and film is by calendaring process.

The calendaring is the method of producing plastic sheets and films by forcing the material between rollers. The calendar usually consists of four rollers which form three opening as shown. One more roller can be added as an additional guide to lead the film away from the last roller.

The plastic heated granule is dropped between the first set of rollers known as the feed rollers. These rollers initiate the formation of the sheet from the granule. The sheet, then, passes over to the second roller gap known as sizing gap. Here, the sheet assumes the required thickness. It, then, moves to the final gap, which is the forming gap.

The sheet so, produced passes between another set of rollers rotating at different speeds. The second set is running faster than the first rollers. So, the sheet is stretched during the passage which gives required strength and thickness to the sheet. The sheet is, then, run over the last roller for winding. (Fig 3)

Fig 3



EXTRUSION FILM ROLLING

Another method of producing sheet is by extrusion method. The material is heated and extruded in the same manner as the other extrusions.

The sheet coming out of the extrusion die further forced between the two rollers to make it into a thin sheet.

One advantage of sheet extrusion is the ability to use different materials in the same sheet.

Another process is blown extrusion for making hollow sheets which are used for making plastic carry bags.

Multi-colour, multi-material and multi-process moulding

Multi-colour

Multi-colour moulding may be single part with different colours like buckets, cups, furniture etc or different colour components assembly like vehicle, knobs, and wheels of toy cars, etc moulded in the same mould. This method has been extensively used in the automobile industry for tail-lights. As many as, five colours can be sequentially moulded into one component in a single integrated operation.

Multi-colour moulding can be done in the following ways

1 Insert moulding (using one or more moulds and moulding operations.)

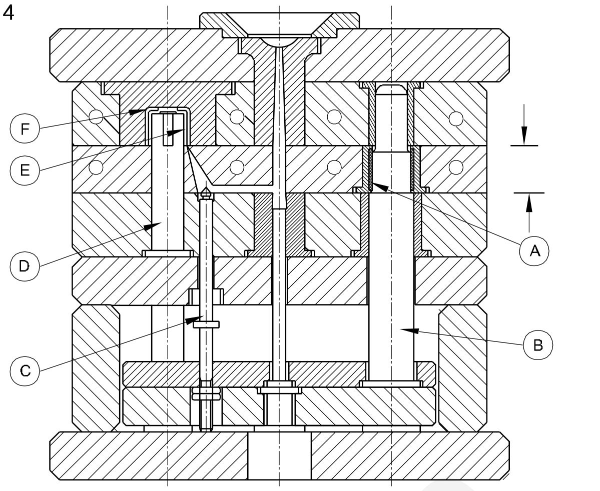
2 Using a multi-colour moulding machine with single injection unit.

3 Using a moulding machine with different injection units and index able mould.

Insert moulding or over moulding (Fig 4)

In this case, a part is first moulded and then, inserting this part in another mould a component with different colours, materials or both can be made.

Fig 4

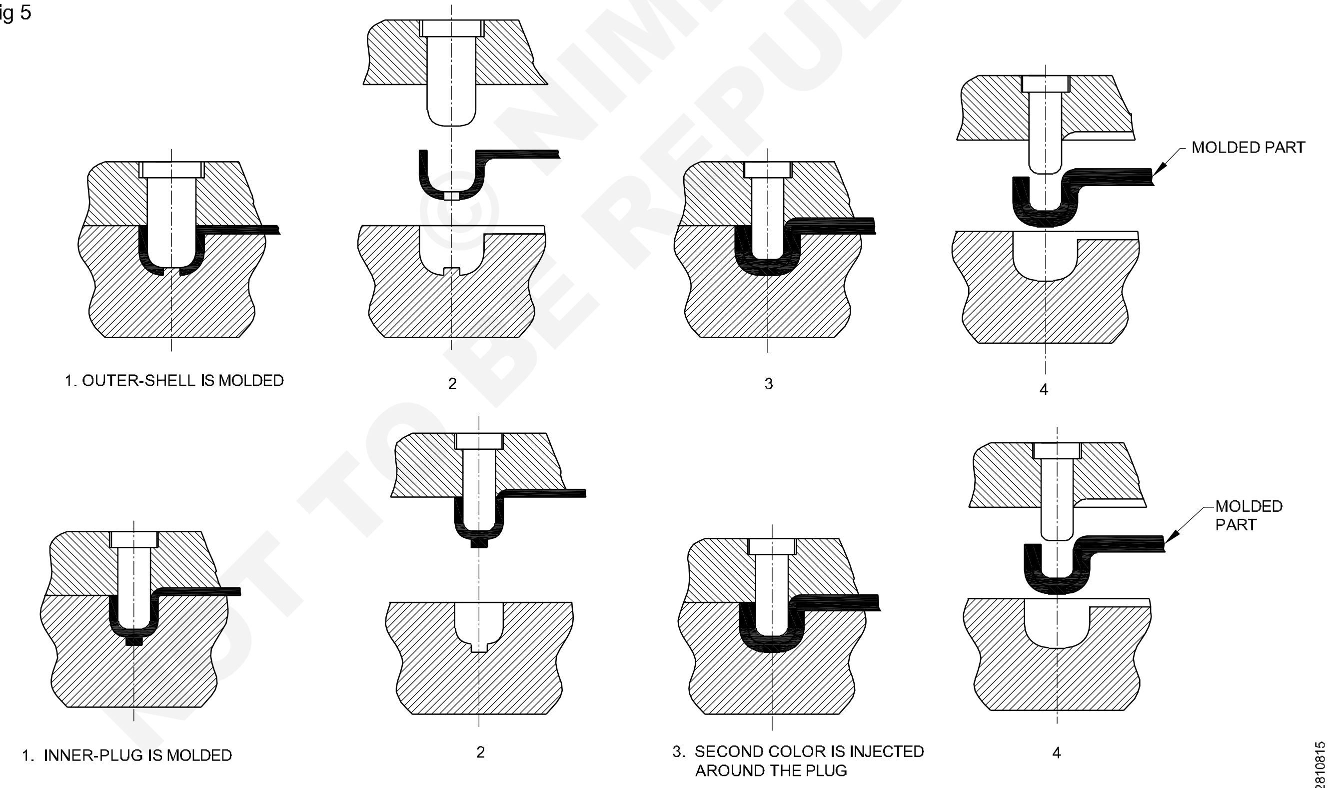


MULTI-COLOR MOLD (INSERT TYPE)

A. BEARING BUSH B. GUIDE PILLAR C. SPRUE PULLER
D. CORE INSERT E. INNER LAYER F.OUTER LAYER

Two methods of making two-colour mold parts (Fig 5)

Fig 5



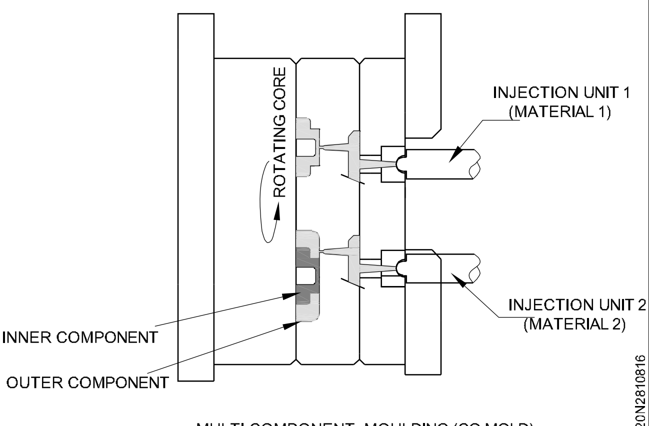
TWO METHODS OF MAKING TWO-COLOR MOLD PARTS

Multi colour moulding machine (Fig 6)

A multi-colour moulding machine has two cylinders and one nozzle. Different colour materials are simultaneously injected in to the mould.

Multi-barrel moulding machine with index-able mould

Fig 6



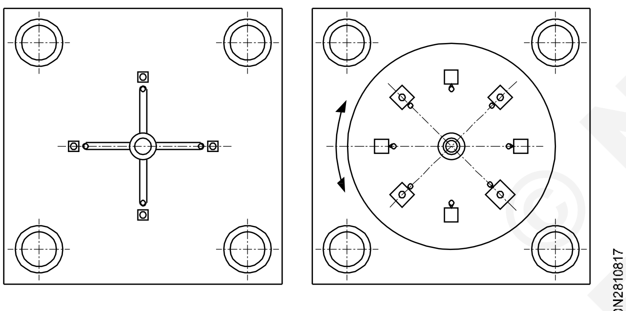
MULTI COMPONENT MOULDING (CO MOLD)

Index moulding (Fig 7)

In a multi-nozzle machine, different material and colour components are moulded by using index-able inserts.

The insert along with the part is indexed and moved into the next position. The indexing mechanism can be part of the mould or part of the machine. Moulding machines have been developed to mould up to four different colours and materials in a mould one after the other.

Fig 7



AUTOMATIC INDEXING TYPE MULTI -COLOR MOULD

Multi-material moulding

A component made of two or more different material is known as a multi-material component. The method of moulding two or more different materials in a single component is known as multi-material moulding. These types of components can be made through an integrated operation (Indexable insert moulding or co-injection moulding) or by means of a series of individual processes (insert moulding).

Multi-material moulding, now, accounts for a growing range of components worldwide. The best known example is the use of different coloured materials for multi-colour tail-light lenses. Expanding, beyond the technology, of combining different colours in a single part, capabilities to incorporate multiple materials via different process afford the design freedom to integrate application specific performance in multi-functional components. Examples, include, integration of stiffness or elasticity in multi-functional components.

The selective use of materials and processes to provide differential performance in localized areas of a component provide significant opportunities for component performance, quality and cost control:

The advantages of multi material processing are:

1 Design flexibility.

2 Modified performance.

3 Less material cost.

4 Increased quality.

5 Less labor cost.

6 Elimination of secondary operation.

7 Recycled materials can be used.

8 Weight reduction.

9 Flexible outer skin with hard core.

10 UV-protective outer skin.

11 Complex moulding.

Co-injection or sandwich moulding (Fig 8)

The method of injecting two materials into a single mould from two injection cylinders either simultaneously or in succession. Co-injection moulding can produce a part with

1 Light core and a hard skin.

2 Solid outer skin with solid core.

3 Flexible outer skin with solid core.

A co-injection moulding machine with two independently controlled injection units that inject the material through a common nozzle. The nozzle permits injection of material from one or both injection units simultaneously. Control of the injection units is critical and requires independent control of velocity, pressure and shot volume.

Multi-process moulding (Fig 9)

It is the method of moulding a multi material component using a combination of conversion processes.

Examples of combination processes used are

1 Injection- injection, e.g. - water sinks.

2 Compression-Injection moulded pallets, vehicle hood.

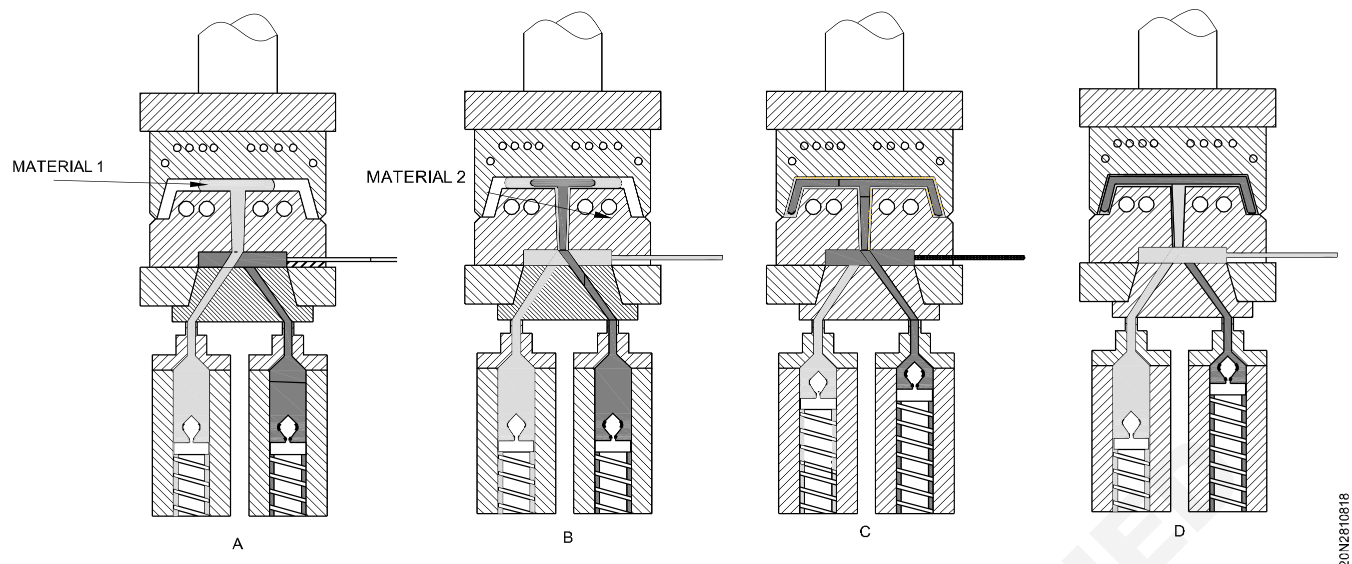
3 Stamping-injection

4 Injection-Compression of car bumpers.

5 Lamination-Injection, e.g. - car interior panel, door etc.

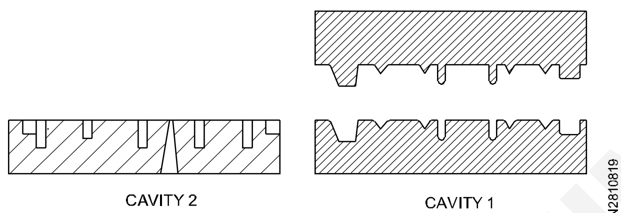
6 Injection- coating, e.g. - all out door equipments.

Fig 8



CO-INJECTION MOULDING

Fig 9



MULTI MATERIAL MULTI PROCESS PALLET TOOL SCHEMATIC

Sequential compression-injection

In this method we require one core and two cavities. The processing order is as given below:

1 Pre- heat the thermoplastic raw material blank.

2 Part is compression moulded using the core and cavity no.1.

3 Cool and open the mould. The part goes along with the core.

4 Move away cavity no.1 and bring cavity no.2 in position.

5 Place moulded component in cavity no.2.

6 Close the mould and inject the second material.

7 Cool and open the mould. The part remains in the cavity.

8 Remove the part and the process continues.

Stamping-injection process

In this process thermoplastic composite material is formed by stamping process and the second material is injected over the form.

E.g. Bumper with thermoplastic composite core with PC / PBT skin.

Step 1: Pre-heated thermoplastic composite or glass mat thermoplastic (GMT) blanks are inserted into the stamping cavity.

Step 2: The beam is formed in the compression moulding process.

Step 3: The mould opens, with the undercuts in the composite (the mechanical interlock) released through tool movement in the cavity. The sliding table locates the injection moulding cavity under the press, while on the core (where the beam remains) slides move around the beam to allow over moulding of the outer layer.

Step 4: The mould again closes. The injection unit comes forward, and the skin is moulded.

Step 5: The part is ejected in the cavity and the cavity moved away from the press.

Simultaneous injection-compression (Fig 10 & 11)

In this technique, the mould is kept open slightly, when, the semi-solid material is injected. Mould is, then, closed to distribute the material by compression.

The advantages of injection-compression moulding are;

1 Lower moulded stress in the part.

2 Thinner walls can be produced.

3 Higher molecular weight components which are not possible by injection moulding can be made.

4 Lamination can be done in the mould itself.

5 Less clamping tonnage required.

Thermo-forming-injection: (In mould decoration-(IMD))

In this process, a film, foil, fabric is inserted into the mould followed by the injection of the plastic material to produce the laminate.

Three types of in mould lamination (In mould decorations-IMD) techniques are available today.

Fig 10

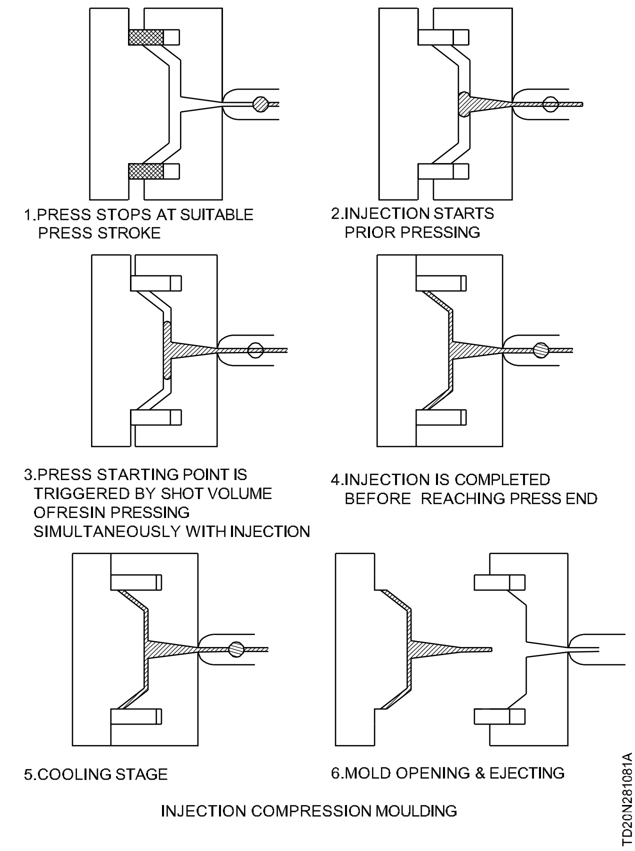
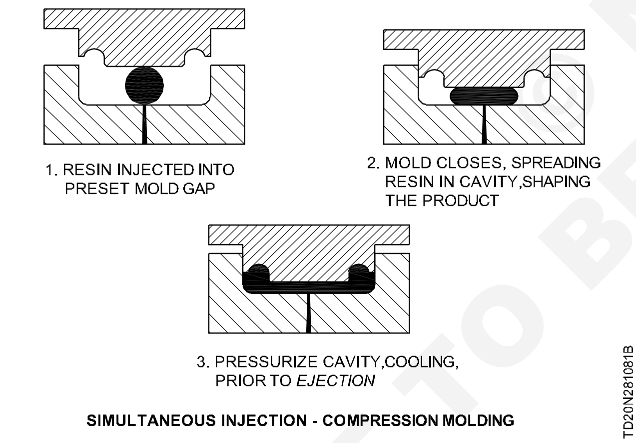


Fig 11



Ink transfer process (Fig 12)

In this process, moulding to decoration is completed in one step. In this system, the film is precisely located. This precision and application of design, allows various graphics, patterns and appearances to be obtained simultaneously.

The advantages are

1 High level accuracy in locating the film available.

2 Trimming process is not required.

3 Moulding to decoration is completed in one process.

4 The processing sequence is as given below:

5 Ejector pins retract.

6 The feeder detects the film-supply sensor and stops the film at the required position.

7 Film is clamped.

8 Suction of film starts.

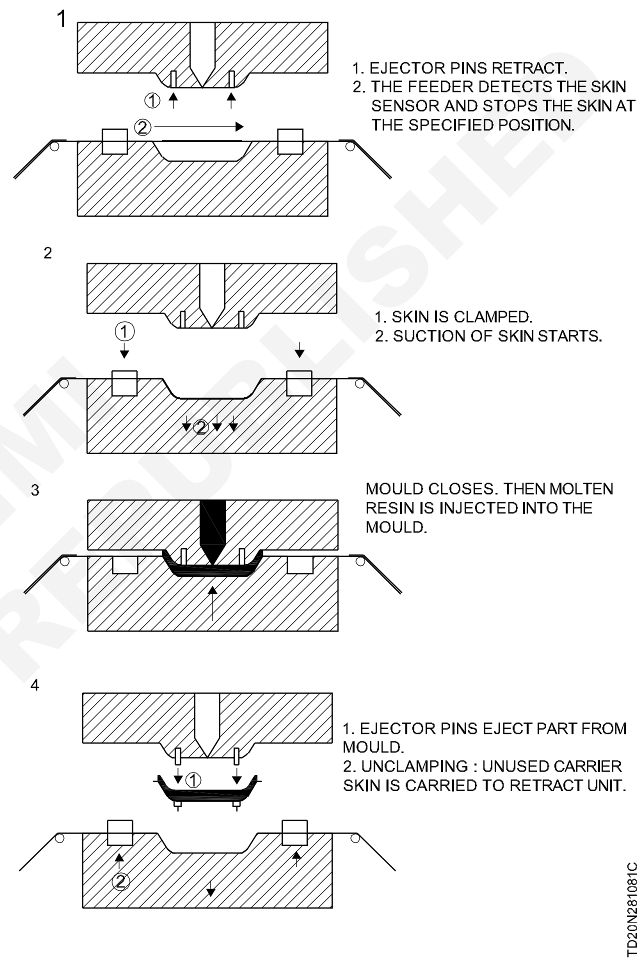
9 Mould closes. Then molten resin is injected into the mould.

10 Mould opens.

11 Robot enters.

12 Ejector pins eject part from mould.

Fig 12



Simultaneous forming process (Fig 13)

In this process in mould decoration is achieved through simultaneous automated overlay of decorated films.

The advantages of process are

1 The decorated film can be applied to the contour of the part with less distortion.

2 High durability and very good appearance.

3 Moulding to decoration is completed in one step.

4 Clamping: heat cutting takes place.

5 Film feeder returns to the standby position.

6 Robot enters and retreats.

7 Robot completes reversing (Film is heated on the film side of the mould, while a completed part is removed from the core of the mould).

8 Suction of film starts.

9 Robot completes departure.

10 Ejector pins retract.

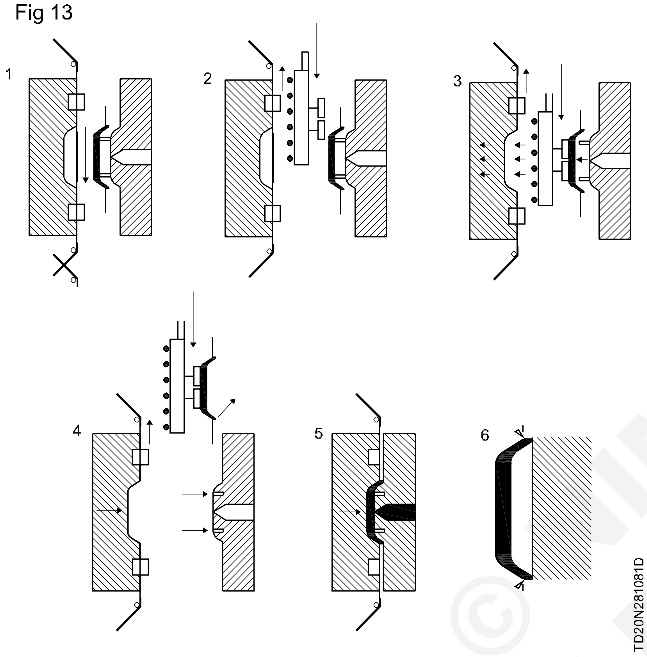
11 Mould begins closing.

12 Mould closes.

Molten resin is injected into the mould.

Trimming occurs after the part is removed from the mould.

Fig 13



Moulding with preforms (Fig 14)

In this process the laminated film is preformed with various graphics and appearances. It is then moulded to a part in an injection mould.

The advantages of this method are:

1 The decorated film can be applied to the contour with little pattern distortion.

2 The depth of draw can be high.

3 High durability and very good appearance.

4 Film is heated.

5 Thermo forming by vacuum or compressed air.

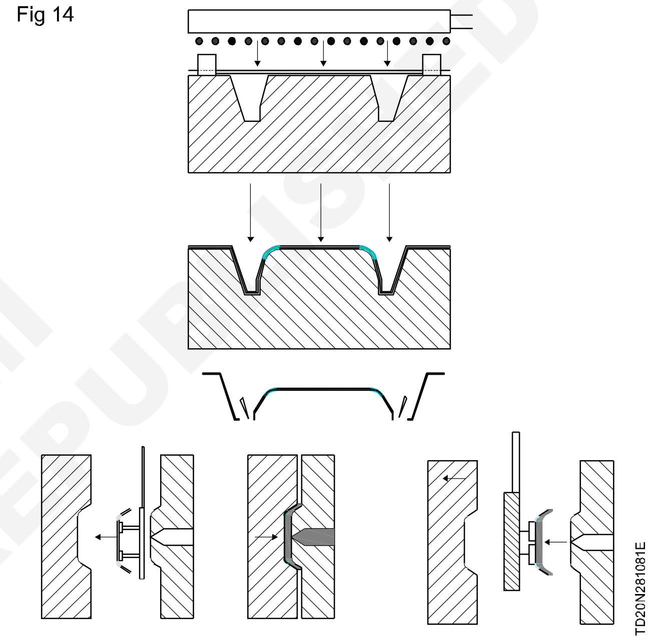
6 Trimming.

7 Form inserted in the cavity.

8 Mould is closed and the material injected.

9 Mould opens and part removed by the robot.

Fig 14



Joining of plastics

Objective: At the end of this lesson you shall be able to
- **explain the various techniques that can be used to assemble plastic parts.**

Variety of techniques that can be used to assemble plastic parts

Most plastics part is a component of a larger product assembly. This part must be combined with one another or with parts produced from other process to produce the final product. The method used to assemble these components must provide mechanical continuity between the various parts. To optimize productivity and quality, the assembly must consist of the minimum number of parts that are feasible, as well as the minimum number of assembly operations.

There are a variety of techniques that can be used to assemble plastic parts to other plastic components or to other materials. The techniques are following:

1 Moulded in assembly

2 Chemical-bonding

3 Thermal welding methods, and

4 Assembly with fasteners

Moulded in assembly

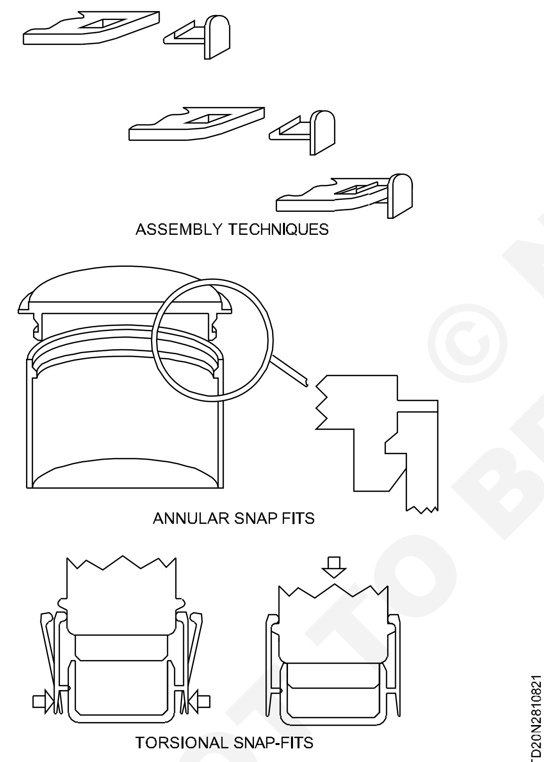
Moulded in assembly systems are generally economical, as no additional steps are required. Examples of moulded in assembly are snap-fits, press- fits, and moulded in threads. Advantages of these methods are that assembly is fast and inexpensive, and does not require additional parts or substances. These, also, minimize the possibility of improper assembly, as the proper fits can be fine tuned in the tooling. The primary disadvantage is that the tooling can be complex and expensive. Some of these methods may not be suitable for parts that need to be disassembled.

Snap- fit assembly (Fig 1)

For high volume production, moulded in snap fit designs provide economical and rapid assembly. In many products, snap fits are designed for, only one, assembly and there is no non-destructive means for disassembly. If disassembly for servicing is anticipated, provision must be made to release the assembly. Other snap fit designs facilitate ease release and reassembly.

All snap fit designs require that potion of the moulded part, such a hook or beam, flex like spring past a designed in interference and quickly return to its inflexed position to create an assembly between two or more parts. The key to successful snap fit design is having sufficient holding power without exceeding the elastic or fatigue limits of the materials. One disadvantage of the snap fit assembly associated with snap fit assemblies is the need for tight tolerance requires maintaining the appropriate amount of interference to maintain required fit.

Fig 1

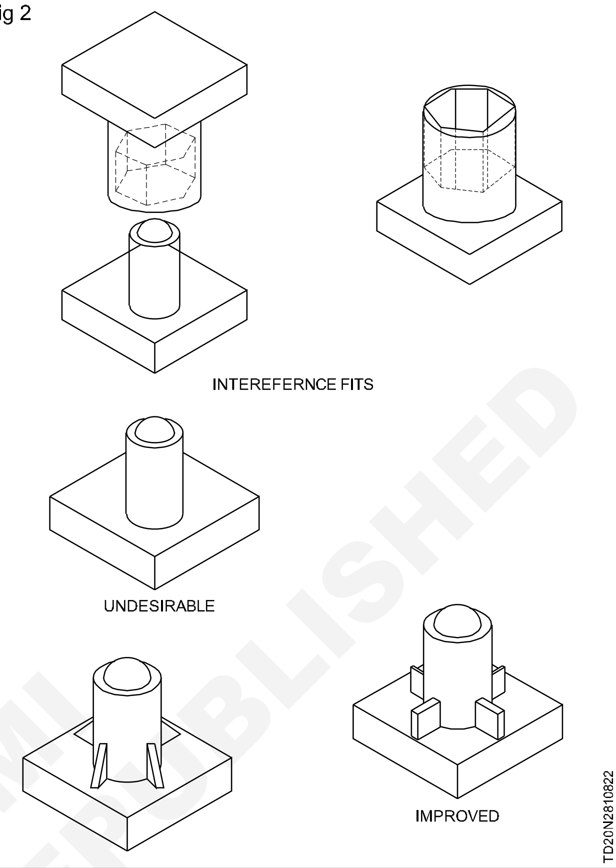


Press fits (Fig 2)

In a press fit assembly, parts or components of a material are assembled to a plastic part, using an interference fit to maintain the assembly. The primary advantage of this system is simplicity. Both, the tooling and assembly are relatively simple. This method is used primarily for cylindrical parts, because, the stresses and strains associated with the interference and distributed uniformly, around the circumference of the part.

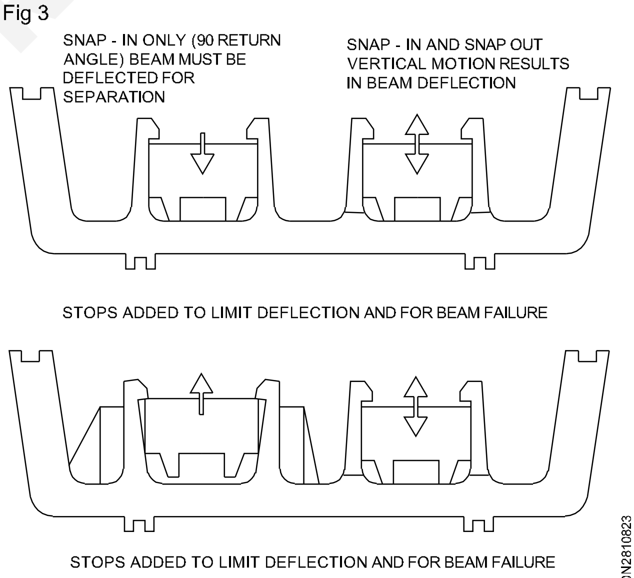
The primary disadvantage of this technique is that, high stress may be created in the plastic part. The consequences of this stress depends upon many factors such as temperature during use, modulus of the mating material, type of stress, and the type of material being used. Some materials creep or stresses relax, while others fracture or craze if the strain is excessive.

Fig 2



Typical snap-fit assembly (Fig 3)

Fig 3

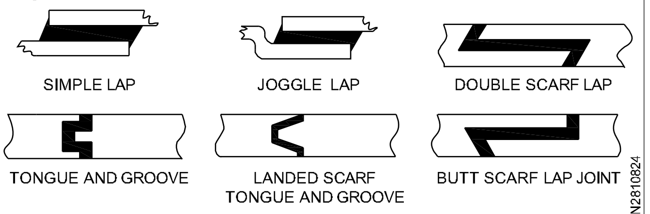


TYPICAL SNAP-FIT ASSEMBLY

Chemical -bonding system (Fig 4)

In this type of assembly, the parts are assembled permanently using reactive bonding chemicals. The strength of the bond depends upon the chemical composition of the bonding material and also, the affinity of the bonded parts.

Fig 4



TYPICAL JOINT DESIGNS FOR SOLVENT AND ADHESIVE BONDING

Solvent bonding

Solvent bonding is a fast and economical method for joining like or similar plastics. The principle involve is to apply a liquid solvent to dissolve the surfaces of the areas to be bonded sufficiently to allow the parts to be joined in a true weld after the solvent evaporates.

The parts are, then, fixture or clamped together, until, the solvent evaporates. Solvent bonding is limited to materials that are compatible and dissolve in the same solvent or combination of solvents. This process is primarily used with amorphous materials as most crystalline thermoplastics have good solvent resistance. The primary advantage of solvent bonding is that, it is inexpensive and requires little or no part preparation or special equipment. Solvent exposure and handling are the issues with solvent bonding. Strict regulation must be observed regarding workers protection, ventilation and solvent recovery.

Adhesive bonding

Adhesive bonding differs from solvent welding, in that, a third substance, which has suitable adhesion to both parts being joined, is introduced at the interference of both parts. This third substance, called an adhesive, is capable of joining together with other plastics, metals, rubber ceramics, glass, wood or any other bondable surface.

Adhesives frequently recommended for use with thermoplastics are epoxies, acrylics, polyurethanes, phenolics, rubbers, polyesters, vinyl and many others, depending upon the plastics being used. Because of their rapid adhesion to many materials, cyanoacrylates have very popular in plastic assemblies. Specific adhesive recommendation can be obtained from either the plastics or adhesive supplier. However, a caution must be noted. It is recommended that the performance of the part be evaluated under end use condition to validate the appropriateness of the adhesive and the joint design.

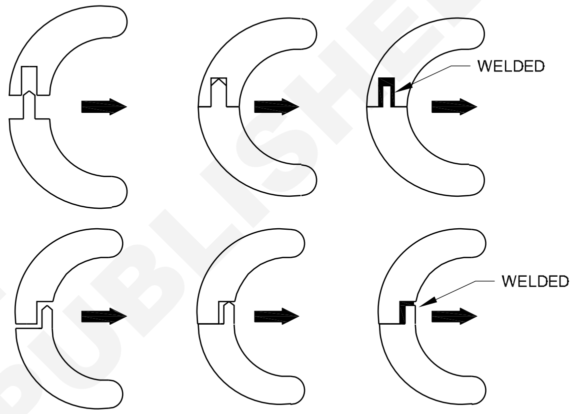
One disadvantage of the adhesive bonding is that, it can take a long time for adhesively bonded joints to achieve their ultimate bond strength. Because. longer clamp times are necessary, more fixtures or special ovens may be required. Surface preparation is critical because any contamination such as grease, oil or mould release can cause bond failure. Surfaces may need to be mechanically roughened or chemically etched to provide improved adhesion. Adhesives are generally used in application where permanent, no serviceable assemblies are required.

Thermal welding methods

Ultra sonic welding (Fig 5)

Ultra sonic welding is economical method for joining small to medium sized parts of the same or similar plastics. This technique is very rapid and can be fully automated. Welding occurs, when high frequency (19-40 kHz) vibration energy is directed to the interface between two parts. This creates localized molecular excitation, causing the plastic to melt. The pressure is maintained between the two parts, after vibration stop. The melted resin solidifies immediately. The entire welding process, normally takes place in less than 2s and has high strength of the base material itself, provided the joint design is appropriate and the equipment is properly set.

Fig 5



SHEAR JOINTS FOR ULTRA SONIC WELDING

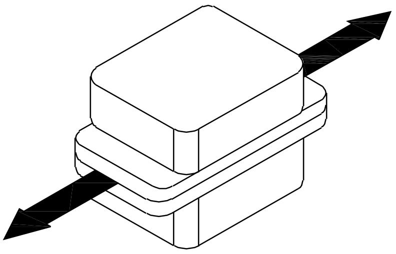
The principle involved in ultrasonic joint design is to concentrate the energy in an initial small contact area, creating rapid melting and melt flow. The melt moves along the joint as the parts are pressed together .The bottom of the plastic part is held rigidly in a specially designed nest. The top part is, aligned with the bottom using the joint design, but, has freedom to couple acoustically when contacted by the horn, which transmits the ultrasonic energy.

Amorphous materials are good candidates for ultrasonic welding. Semi-crystalline polymers, generally, are more difficult to weld using ultra sonic welding. These materials tend to dampen ultra-sonic vibrations more than amorphous materials. Using higher energy levels, these materials can be successfully welded. Because, conditions vary with materials used, equipment, and joint design, it is recommended that the resin suppliers and the ultra-sonic equipment suppliers are consulted during the product design process.

Vibration welding (Fig 6)

In vibration welding, two parts are rubbed together, producing frictional heat, which results in a melt at the interface of two parts. The movement is in the form of high amplitude, low frequency, reciprocating motion. When vibration stops, pressure is maintained, until, the melt cools and the parts become permanently welded in the proper alignment. Typical frequencies are 120 and 240 Hz, and amplitudes range between 2.5 and 5.0 mm (0.10 and 0.20 in) of linear displacement. Vibration welding equipment can also produce angular displacement, when, the geometry or assembly of the product prevents linear displacement.

Fig 6



Vibration welding has a number of advantages over other welding methods. Standard equipment handles part up to 30cm x 60cm (12 in x 24 in).Property designed joints can provide hermetic seals, and internal walls can be welded at the same time as the external walls. Also, the reciprocating motion pushes surface contaminants, such as mould releases, coatings and even paint, out of the weld area. The primary disadvantage of this process is that the weld is generally limited to a single plane. Also, equipment cost is some what greater than for other welding techniques.

Spin welding

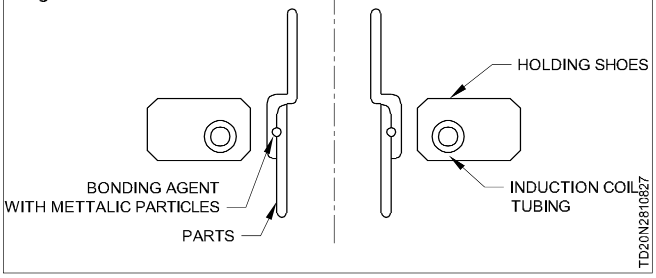
Spin or frictional welding is a relatively simple method of joining thermoplastic parts with circular joint interfaces. As with vibration welding, frictional heat is generated, creating a melt and subsequent weld. While, one part is held fixed, the rotating part contacts it with a specified pressure, generating frictional heating. After the melting occurs, rotation is halted, but, pressure is maintained on the part, until, the joint has solidified.

The entire process,usually,takes less than 3s and can be easily automated compared to ultra-sonic and vibration welding, and the equipment is simple, sometimes, just a property equipped drill press for low pressure runs.

Electromagnetic, or induction, welding (Fig 7)

Electromagnetic, or induction, welding is one of the high technology welding techniques in use today. This process utilises high frequency electromagnetic fields to produce heat and melt at the interface of two thermoplastic parts. The process is fast, efficient, and versatile. It consists of exposing a magnetically active gasket material to an oscillating magnetic field. This gasket material is produced by incorporating minute ferromagnetic particles in a thermoplastic matrix, that is, the same as the materials to be joined.When exposed to a magnetic field; these particles become exited and produce enough heat to cause the gasket materials to flow. Heat generated in the gasket material is conducted to the abutting surfaces, causing them to fuse to the gasket. The process occurs in 3 to 10s and is dependent upon the weld area and the base resins to be welded.

Fig 7



The advantages of electromagnetic welding are numerous. There are few size limitations and weld lines up to 4.6 m (15 f) have been welded. Complex joint configurations are welded parts to be disassembled by reacting the gasket. Induction welding can produce structural, hermetic welds in most thermoplastic materials and can generally be automated for large volume production applications.

One draw back of this technique, is, the need for a third component for the weld. The gasket adds costs the process, particularly, in the case of, custom moulded gasket shapes. Parts containing metal inserts pose a problem, because, the insets may also react to the magnetic field. The cost of supporting equipment can become significant as different applications require custom coils.

Hot platen or hot plate welding

Hot plate welding consists of heating two parts to be joined with a heated insert. The heated platen, at the correct temperature, is placed at the interference of the part to be joined. The parts are, then, brought into contact, or in some cases, very close to the hot surface. Stops on the inserts and the fixtures that hold the part determine the depth of the initial melt. The hold time determines the secondary softening. This initial contact produces a good mating surface as both are made level by the insert. At this point, the insert is removed and the parts are pressed together. The stop on the fixtures, now, determines the amount of materials displaced during mating. The entire assembly is usually completed in 15 to 20s. One of the disadvantages of this method, is, the long cycle times required and material degradation also possible.

Staking

Staking is the process of deforming a plastic stud or boss to capture the part. This can be accomplished using either heat or ultrasonic energy and pressure. Both processes are performed with a contoured tool that transfers heat or energy to the plastics "stake" producing a melt phase and exerting pressure to reform the plastic. Tight assemblies require that pressure be maintained, until, the plastic resolidifies into the new holding contour.

Ultrasonic, are generally chosen over heat staking because, the contact tip of the tool stays relatively cool, during the process and forms a clean head with minimal sticking and stringing. For both, thermal and ultrasonic staking, a slow tool approach is needed to develop even melting and prevent cold forming with its associated stress.

Assembly with fastners

Threaded metal inserts

The most common means of assembling plastic parts are threaded metal inserts. The metal inserts are permanently installed into the plastic part during moulding. These inserts should be prevented from rotation by incorporating certain profiles on the outer surface. E.g.: Undercuts, knurling, hexagon, etc...

Self tapping screws

Screws create their own threads, are, widely used with all thermoplastic parts. They are economical and require no extra operations or special unscrewing cores, because, the thread is cut or formed by the screw during normal assembly operations. A cutting screw actually removes material like a thread cutting tap and works with most of the materials. The forming screw as the name implies actually displaces the materials and must be used with caution as very high hoop stresses can be developed in attempting to form threads in high modulus, low creep materials. These stresses can be additive to other loads and may lead to premature part failure.

Thread cutting screws are not recommended for applications requiring repeated assembly and disassembly for, if, care is not used; subsequent reinsertion may results in formation of a second set of threads.

Rivets

For fast permanent assemblies metal rivets are frequently used. Care should be taken to avoid the high stress concentrations inherent in most riveting techniques. Large diameter heads and reinforcing washers are suggested to distribute stresses, and all corners should have a radius. Aluminium rivets are preferred over steel, because, aluminium will more readily deform under high stress. Use of a shouldered rivet will also, control stresses by limiting the compressive forces that can be applied to the plastic parts.

Moulded in threads

Threads can be moulded into the injection mould parts for use in assembly. A number of different types of threads are commonly used and there is, much special type of threads for specific applications. The type of thread selected will make a large difference in the performance of the part. Design guidelines can be obtained from material suppliers.

Bolted assembly

Standard fasteners, screws, bolts, nuts, lock nuts, lock washers and other specially designed products are used frequently, with the plastic parts. Because, these threaded fasteners are usually made from steel, brass or high strength metal, the plastic parts being assembled can likely to overstress well before these fasteners break or strip. Therefore, good design practice dictates that certain precautions to be taken to prevent excessive stresses when bolted assemblies are used. The most obvious method for preventing a high stress assembly is, to carefully control the tightening of all fasteners with properly adjusted torque limiting drivers. This approach works best when the operations are on a factory assembly line. However, field or customer, assembly prevents a different problem.

Maintenance of moulds

Objectives: At the end of this lesson you shall be able to
- **state the points to be taken care for effective maintenance of the mould**
- **list the types of maintenance and its control**
- **brief frequency of maintenance of the mould.**

Points are to be taken care for an effective maintenance of the mould

Injection moulds play a vital role in production of qualified components. Consistency in the quality would also, depend on the quality of mould. Unreliable mould would need frequent adjustment, repairs etc. It is therefore necessary that mould should be reliable in its function and moulding of qualified components.

One of the prime objectives of any moulder is that the moulds in use should function well to the maximum possible production quantities. A mould properly designed and made may not remain necessarily proper, unless kept and maintained in a proper manner. Therefore, upkeep and maintenance of moulds are important practical aspects of a moulding shop.

The following points are to be taken care for an effective maintenance of the mould:

- Upkeep
- Maintenance
- Specification sheets for all moulds.
- History sheets.
- Instruction manual on upkeep and maintenance

Upkeep

Upkeep means that the mould should be kept well cleaned. Before closing, the two halves of the mould, an anti-rust spray may be used to prevent the rusting of the mould parts. The mould should be stored preferably at a place, where, the humidity is low and ambient temperature is not excessive. Moulds which are stored near a chemical factory or storage, the cavity/core must be chromium plated, otherwise, due to the chemical vapour, the core and cavity will get corroded or eroded. Rusting can be very much minimized, if the moulds are kept covered by PVC sheet covers.

Maintenance

It means that we should keep the moulds and bring them to production in a worthy condition. We can do a preventive maintenance of the moulds and it would cover examining mould for small damages. Check, whether any problem with clearance and the wedge blocks. If it is there, increase in the clearance of the moving parts, the adjusting of the locking wedges. The cleaning of the water channels is a must, otherwise, it will get rusted inside the hole and finally, the flow of water will not be sufficient. Then, check for guide pins. Sometimes, any guide pin or cam pin may found shaky in its holding hole. In that case, as a maintenance action, either a new pin may be put or the existing one may be copper plated at the holding surface.

Specification sheet

Specification may vary from mould to mould. It gives complete information about the mould, which can be stored in a computer. Each sheet may be stored in a computer, allotting a suitable file number. This also, helps to the persons, who are not familiar with the moulds and might have an occasion to handle maintenance and upkeep of moulds.

History Sheets

This pinpoints, the defects, which are occasional / which are repetitive / which are chronic in nature. This helps to take appropriate steps to find the solutions to the defects and take corrective measures. It also, reveals the management a lot of information about the performance and capabilities of the shop floor personnel.

Instruction Manual

Upkeep and maintenance actions are almost similar for all the moulds, but some special precautions may be needed for some moulds. It is therefore, a good practice to prepare a manual on "moulds upkeep and maintenance". Technical personnel involved in upkeep and maintenance can follow it.

Points which affect Physical Mould Life

The following points to be seriously considered which will affect the physical mould life.

- Inadequate physical strength of mould parts to withstand the loads encountered and caused by clamping force, injection pressure, ejection force etc.
- Improper mould making practice
- Improper allowance for fatigue in mould studies in a frequent
- Cause for breakage.
- Improper heat treatment specification.
- Lack of lubrication of sliding faces
- Incorrect machine set ups.
- Poor maintenance practice and rough handling of the moulds

- Moulder not adhering to the suggested start up and maintenance procedure
- The use of dirty and contaminated plastics
- Abrasive, erosion or corrosion fillers such as glass and some other additives.
- Highly humidity
- Dirty and corrosive cooling water

Types of maintenance

There are four levels of maintenance activities and one breakdown maintenance. All these are explained in detail below. The types are

- Routine Maintenance — LEVEL I
- Inspection Maintenance — LEVEL II
- General Maintenance — LEVEL III
- Major Maintenance — LEVEL IV
- Breakdown Maintenance

Routine Maintenance - LEVEL I

This is a very simple preventive maintenance. Before, the mould is removed from the machine, it is to be heated (if it is chilled) so that, all surface condensation is evaporated. With the mould still warm, internal mould surface should be cleaned with shop safety solvent to remove any residual dirt and stain. The ejector system moved fully forward, then, sprayed with approved rust preventive before, the ejector system is retracted and closed. Lenses and highly polished surfaces should be protected by following specific instructions. All water lines should be drained and blown free of all residual water to avoid build up of rust due to standing water. Check and assure all bolts, plates, clamps, etc are in place and tight. Bag the last shot as an example of the typical quality of this run. Any components missing or cavity blocked off should be noted and attended to. Store these parts with the mould.

Inspection Maintenance - LEVEL II

It is done for every 20,000 cycles and also, at the end of production run. Using the check list and visual inspection technique, the mould is looked over. Any minor repairs that are necessary are noted. If not, needed repairs or future required touch up notations are kept with the mould history log for future evaluation. Any components missing or cavity blocked off should be noted and attended to. A sample from blocked cavity should be retained for the mould maker to make repairs. The mould should be washed with safety solvent to remove the varnish and build up from the moulding process. Notations for work to be done during the general maintenance procedure should be noted on the form for future work. Bent, worn or broken ejector elements should be noted. The mould should be removed from the production and elements replaced

General Maintenance - LEVEL III

This is after 1, 00,000 cycles of moulding. Only, competently trained tool room technicians should perform this maintenance. All plates are separated and their faces cleaned. Highly polished surface should not be cleaned with brushes and rags. Clean with facial tissues or cotton balls. All components are checked for wear. Any excessive wear is noted and a determination is made to repair, replace or continue to use. All rough areas outside the cavity detail area are to be worked out. All moving parts are to be lubricated, if required. Vents should be checked for depth, width and land as compared to the tool drawing specifications. "O" rings, seals and gaskets should be checked for integrity. All water lines are to be pressure tested for any leaks and re-certified for flow capacity. Water lines that have built up scales are restricted should be pressure cleaned with de-scaling agent. The ejector system is to be examined for proper alignment. Any damaged hole or pin should be reworked replacing broken return pins, re-plating or re-texturing to be done as a result of the material eroding the mould surface. Replace all springs after 50,000 cycles. Nature of work done should be recorded in the general maintenance file.

Major Maintenance - LEVEL IV

This is after 250,000 cycles. This maintenance should be performed by skilled tool makers. Before maintenance starts, there should be four complete shots delivered with the tool for study. Two shots should be from initial mould qualification. This gives a visual record of what was acceptable when the mould was new and fully functional.

The second two shots should be the most recent parts produced before the tool was pulled for maintenance. Comparison of the both shots will find an excellent indication of wear and abuse the tool was suffered. All components determined and authorized to be replaced should be removed and new components constructed and installed in accordance to the original design if previously certified parts are not there. Worn leader pins, bushings and all bearing moving surfaces should be checked for wear and replaced or repaired. Plates and mould cavity surfaces should be checked for parallelism and ground flat required. Mould cavity surfaces should be cleaned and polished as required to the original surface requirements. All moving components should be checked for easy movements, adjustment to be made as required.

Springs are to be replaced with new springs to avoid fatigue.

Break down Maintenance

It is done unexpectedly when an accident occurs to the mould is in production. The problem may be, water leak, core/cavity damage, cooling line blocked, ejector system failure and heater failure in case of hot runner mould, etc. We have to study the problems along with production engineers and to bag a last shot from the production.

The maintenance of the mould is done with reference to the decision taken in the outcome of the meeting.

Mould removing, cleaning, and storage

Whenever a mould production is over and it is to be stored in a store room, we should do maintenance for the mould. This mould has to be taken for production after sometime. We should heat the mould to room temperature before removing from the moulding machine to evaporate any surface condensation. Using a mild, clean shop safety solvent and soft, clean towels gently clean internal mould surfaces to remove residual dirt and grime. Move the ejector system fully forward and spray both mould halves with shop approved protective rust preventive before retracting the ejector and closing the mould. Drain and blow free all residual water to avoid rust buildup.

No water should be trapped inside the mould and it is critical. Make sure all bolts, plates and clamps are properly placed and the safety straps are secured and tightened.

Remove mould, check and clean it again, if required. Bag the last two or four shots to store with the mould including feed system as examples of the expected run quality.

Store the moulds on storage racks notably out of shop floor. Don't let water lines stick out when storing moulds on racks. Wrap the mould with plastics. Ensure that air is dry when wrapping, or put dissident air absorption bag inside, so that, temperature changes don't cause condensation inside the wrapping that could form rusting.

Maintenance of idle moulds

Basically, a repeat of activities already done during preventive maintenance during production. We have to check the flash faces. Then remove stains and special attention to the given to air vents. You must check the cooling channels. Replace "O" rings if necessary. Check slating ejector pins, normal ejector pins, guide pins, and pins for slide movements on wear and tear and bending. If necessary, replace. Check springs and replace if necessary. Replace them in sets. Check on leakage after assembly.

Check sprue bush on dented marks. Check slide movements. Blow cooling channels before bring the mould down.

Preventive Maintenance

The preventive maintenance of the mould mainly depends on the role of designer and the mould maker. If they do their job very well, then this maintenance will be very easy.

The Role of Designer

The designer must study the part drawing first. He must identify the area of concerns and to be discussed with customer to avoid on doing mould and moulding problems.

He must discuss to customer about the part design which include sharp corners, inadequate draft, and limited areas of ejection etc. which call fragile steel sections in the mould. He should discuss with them to determine and modifications can be made in the part design. Mould designer should incorporate into the mould design provision for mould maintenance. Critical area should be inserted for ease of repairs. Materials should be used which are resistant to galling when slides are required and wear strips and locks should be accessible and adjustable. Steel should be selected for strength and hardness properties that will ensure reliable performance. Mould bases for high production moulds should be fabricated from steel that will not collapse and more under stresses induced by the moulding operation.

The Role of the Mould Maker

Mould maker must follow the mould design, any changes in connection, should be with concurrence of the mould designer. If a mistake in design has been noticed, the mould maker should inform the designer so that the correction can be made & recorded.

Identifying steel for parts is important. After hard machining he should send the steel for stress relieve or send when appropriate. Heat treatment of steel should be documented to ensure that proper hardness has been achieved.

Maintenance control

An effective maintenance system will reduce the risk of needless downtime and poor quality parts in both short and long run moulding condition. Maintenance report is important that individual records are kept for each mould and injection moulding work. These records will document their history and should contain the following

- Machine/mould specifications including accessories.
- Record all modifications
- Maintenance logs
- Downtime logs
- Manufacture services call reports
- All alterations to the machine since installation.

Frequency of maintenance

Frequency of maintenance is determined by four factors. They are

1 Material

2 Complexity

3 Moulding material

4 Abuse

Material: Aluminium or soft tools will suffer wear and tear in a shorter period of time than tools made of conventional tool steel. Mould made of hardened steel will last longer than those made of conventional mould steel.

Complexity: Moulds with intricate mechanism or parts requiring unreasonably high precision will require more maintenance than a simple two plate mould of low precision parts. Slides, lifters, internal cores, hydraulic and mechanical systems, hot runners, complex ejector system, all add to the maintenance schedule.

Moulding material: Materials with high melt temperatures wear out moulds quicker than those with low temperatures. Higher melt temperature will expose the metal to more heat and enhance material wear. Filler materials are abrasive to metal; they will tend to wash the mould steel away after thousands of cycles.

Abuse: Excessive clamp pressures, high injection pressures, jerking the mould, opening and closing the mould, not lubricating the appropriate components, multiple ejection, crashing the mould closed or closing up on the partially ejected parts are a sure road to increased maintenance.

Abuse can also be defined at the work place. A shop that is dirty, open to the outside dust and dirt, machines with grease and absorbent on the floor all creates an atmosphere for mould to wear out quicker. Area exposed to acid vapour, area of high uncontrolled humidity or salt water, dry area with constant wind and dust cause premature wear on the mould. When the mould is clogged or restricted cooling circuit, this also is an abuse which is called Environmental Abuse.

So, the upkeep and maintenance of the mould is very essential to keep the mould to run beyond its life.

Pressure die casting

Objectives: At the end of this lesson you shall be able to
- **state what is die casting**
- **explain the hot and cold chamfer process**
- **list out the process variables of die costing process**
- **brief the gating systems in diecasting**
- **calculate the force related to die to determine machine size**
- **list out the die casting defects, causes and its temedies.**

Introduction

Pressure die casting is a process in which molten metal is forced under high pressure into a metal die in a fraction of a second and allows it to solidify. When the casting has solidified, the die is opened and the die casted component is removed.

The process is rapid and allows complex shapes to be cast as almost finished parts, and many thousands of castings can be produced from a set of dies without significant changes in casting dimensions.

There are two basic die casting processes.

1 Hot chamber process

2 Cold chamber process

The hot chamber process is used for die casting metals with low melting point such as zinc and lead. The cold chamber process is used for metals that melt at higher temperature such as Aluminum, Magnesium and Brass.

Advantages of die casting

1 Die casting provides complex shapes with closer tolerance than many other mass production processes.

2 Die castings are produced at high rate of production. Little or no machining is required.

3 Die casting can be produced with thinner walls than those obtainable by other casting methods.

4 Zinc casting can be easily plated or finished with a minimum surface preparation.

5 Die casting can be produced with surface simulating a wide variety of textures.

6 Holes in the casting can be cored and made to tap drill size

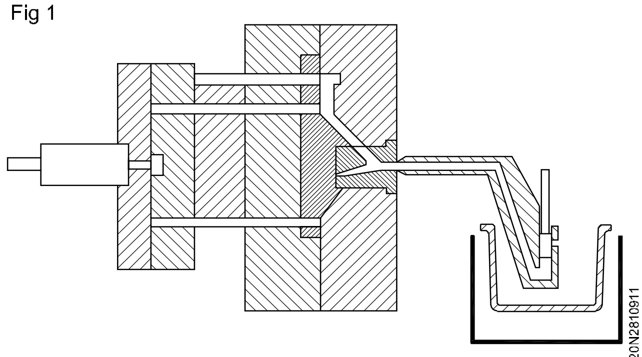
7 Inserts of other metals and some times nonmetals, studs, bosses can be die cast in place.

Hot Chamber Process

The basic components of a hot chamber die casting machine and die are shown in Fig 1. In a hot chamber process the plunger and cylinder are submerged in the molten metal in the holding furnace. The power to pump the material into the cavity is provided by a hydraulic accumulator. Oil is supplied to the accumulator by a rate that will bring the accumulator pressure up to the desired operating level.

When a shot is made the control valve opens allowing the shot cylinder to force the plunger down and force molten metal through the nozzle to the sprue and through the runners and gates into the cavity. The gases that were in the system and some of the molten metal flow through the die cavity and out through the vents or into the overflows. After the cavity is filled the metal is allowed to solidify, the casting is ejected and the cycle is repeated. Since the gooseneck and plunger are submerged in the molten metal, the system refills automatically when the plunger is withdrawn. Since the metal has to pass through the thickness of the back plate a sprue hole is provided. The sprue hole is maintained almost at the same temperature of the die temperature so that the sprue solidifies along with the component.

Fig 1



HOT CHAMBER DIE CASTING MACHINE

Operating sequence of the hot chamber dies casting process

1 Die is closed and hot chamber (i.e. gooseneck) is filled with molten metal.

2 Plunger pushes molten metal through gooseneck and nozzle and into the die cavity. Metal is held under pressure until it solidifies.

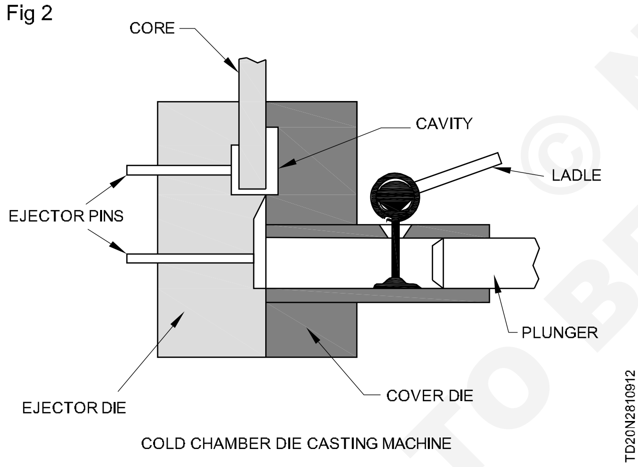
3 Die opens and side cores, if any, retract. Casting stays on the ejector side. Plunger returns, pulling molten metal back through nozzle and gooseneck.

4 Ejector pins push casting out of the core. As plunger uncovers filling hole, molten metal flows through inlet to refill gooseneck.

Cold Chamber Process (Fig 2&3)

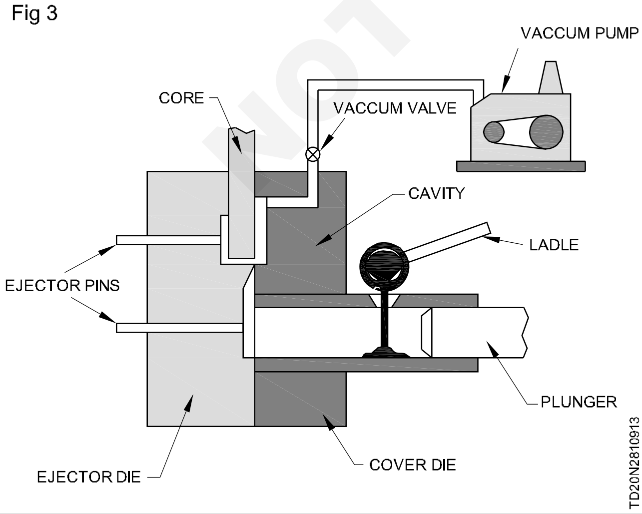
In this process the metal injection mechanism is not submerged in liquid metal. The material is heated outside and poured into the sleeve through which it reaches to the runner and gate. The machine usually has a plunger that is mounted horizontally in a sleeve. The plunger enters part way in to the die pushing the molten material in to the impression. Any excess metal remains in the end of the shot sleeve between the plunger and the parting surface of the die. The solidified remaining metal is called the biscuit. The component along with the biscuit is ejected out of the die.

Fig 2



COLD CHAMBER DIE CASTING MACHINE

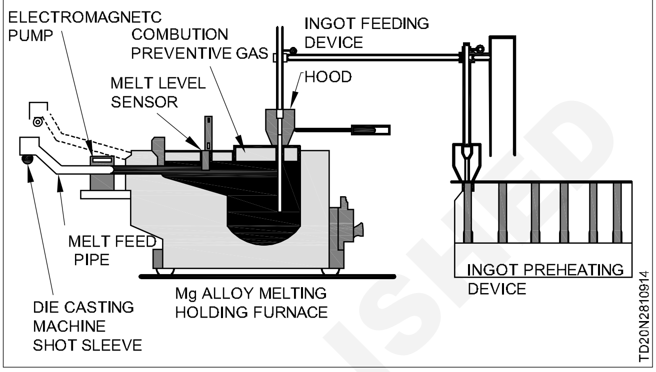
Fig 3



Magnesium die - casting system (Fig 4)

Magnesium alloys, since they are the lightest and the highest in strength-weight ratio among the commonly used structural materials, are attracting attention as energy saving materials for automotive parts. In addition, the needs for reducing the weight, wall thickness and size of cellular information equipment, and the recycling ability make them replace plastic materials. A fully automated magnesium die casting system developed by Toshiba Machine is shown below.

Fig 4



Operating sequences of the cold chamber die casting process

1 Die is closed and molten metal is ladled into the cold chamber

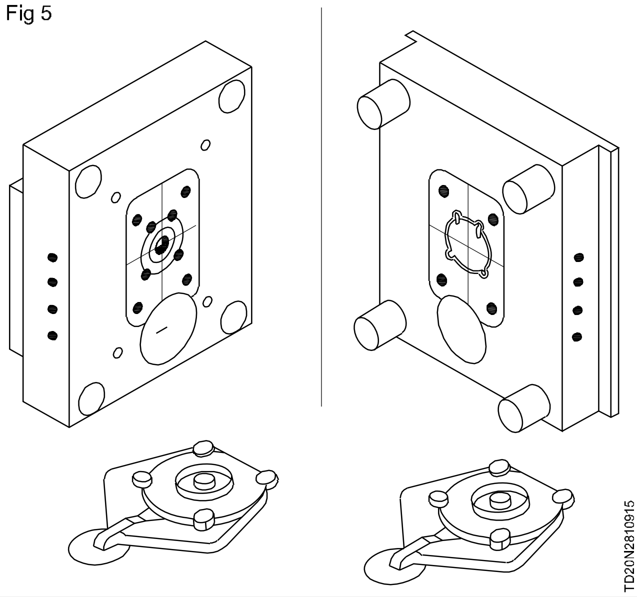
2 Plunger pushes molten metal into die cavity. The metal is held under pressure until it solidifies

3 Die opens and plunger advances to ensure casting stays in the core half side.

4 Ejector pins push casting out of ejector die and plunger returns to original position.

Anatomy of die casting die (Fig 5)

Fig 5



Pressure die-casting process variables

The process variables of the die-casting process are

1 **Metal**

a Alloy composition.

b Metal temperature.

c Shot weight.

2 **Machine**

a Accumulator pressure

b Injection line pressure

c Intensification

d Plunger speed

e Locking force

f Mode of injection

g Plunger diameter

h Timing of stages in the casting cycle

i Lubrication

3 **Die**

a Die temperature

b Filling rate

Process variables (Metal)

1. Alloy composition: A metal having good casting qualities is of value unless it has the properties like dimensional stability and reasonable price to ensure durable casting. If a wrong alloy is taken for making a specified casting, it can be molded and put to use but it will not perform its functions properly in course of time. There are also cases where more than one metal alloy is suitable for a component. In such cases it is decided on the cost of the alloy.

Metal temperature

Aluminum: The temperature for the metal entering into the cavity depends upon the holding furnace temperature and the temperature of the sleeve, the mode of injection and delay between pouring and start of injection. If the temperature is less than 650°C die casting alloys like LM2, LM4, LM6 and LM24 containing silicon, iron (impurity) and manganese which form compounds and settle out as hard spots which might result in considerable difficulties in machining the casting. (Aluminum is moulded by the cold chamber process).

Zinc: If the metal temperature is too high it shortens the life of plunger and goose neck and it has been reported that the magnesium loss increases substantially. The ideal casting temperature for zinc ranges from 410°C - 420°C. if the metal temperature is too low the molding material sticks onto the plunger stem unless it is heated in this area separately. This build up can increase the friction between the plunger and the sleeve (goose neck sleeve) which results in slower injection speed

Shot weight: Variation in shot weight can influence the quality of castings. It affects the volume of gases in the shot sleeve, average heat input by the metal entering in the die and the size of slug which in turn influence the transmission of force from the plunger to the die cavity. If the size is less in the final stages of injection the slug will be thin (biscuit) and the impact on the die will be greater if the size is more. It can be wastage of material. The ideal biscuit thickness can range from 6-50mm. In some cases excessive shot weight can be dangerous because very long slug can burst and can injure the operator as the die opens.

Process variables (Machine)

1 **Accumulator pressure:** Accumulators are used to store up energy from hydraulic pump during the process cycle. If the pressure inside the accumulator is too high it can result in the separator not being lifted with the result that the accumulator will not operate. Too low pre charge will result in a low injection pressure.

2 **Injection line pressure:**This controls the maximum force on the plunger and it influences the maximum injection force.

3 **Intensification:** It is defined as the controlled increase of pressure on the casting alloy at the end of the cavity fill, immediately following impact. It is achieved by increasing the hydraulic pressure over the nominal value by shifting to alternate relief valves opening high pressure accumulators or operating multipliers called cylinder intensifiers. The usual ratio is 3:1 compared to the pressure of filling the cavity. The objective of this large pressure is to compress gas porosity voids that have occurred during cavity filling. Intensifiers enable large changes to be made in the injection force without changing the line pressure. The rate of pressure build up should not be too fast, otherwise it may cause the die to flash. Alternatively too long delay in pressure results in a low pressure being applied during the final stage of injection.

4 **Plunger speed:** A controlling valve sets plunger speed. Small variation in plunger speed can happen due to the fluctuations in the injection line pressure and also with the large variation in the die temperature. If the plunger speed is too high it can result in flashed cast because of the heavy impact and the metal being in the fluid condition even after the completion of the cycle.

5 **Locking force:** Locking force depends upon the locking pressure, tie bar nut setting and temperature of the die and surroundings. As the die heats up, locking force increases and unless an allowance has been made for this increase, the machine will stall when locking. The locking force, injection pressure and projected area of casting together with the die face deviations are the influencing factors in controlling the flash. For the best result, as far as possible the design should be made so that the center of pressure produced in the cavity acts at the center of the area between the tie bars. The impression should be placed in the center of the bars.

6 **Mode of injection:** The use of two stage injections with a slow first stage allows the gases more time to escape from the cavity during the initial fill period. Since the initial speed is less, it avoids the flashing of molten metal through the nozzle and runner. By changing mode of injection the cavity filling speed can be increased or decreased.

7 **Plunger diameter:** To increase the shot capacity (shot capacity is the maximum weight of material which can be pushed out of the nozzle in one stroke) large diameter plunger can be selected. Increasing the plunger diameter also reduces the final force on the metal and tends to reduce plunger speed during injection. Instead of changing sleeves and plungers to suit different shot capacities, and also to reduce the number of spare parts required, it is often convenient to use the same size plunger for the whole range of castings produced on a given machine.

8 **Timing of stages in the casting cycle:** Usually the speed of casting cycle is controlled by the opening and closing time in an automatic machine and in case of a semi automatic machine it is the speed of the operator. Once the tool is loaded, it is better to run continuously till it is unloaded.

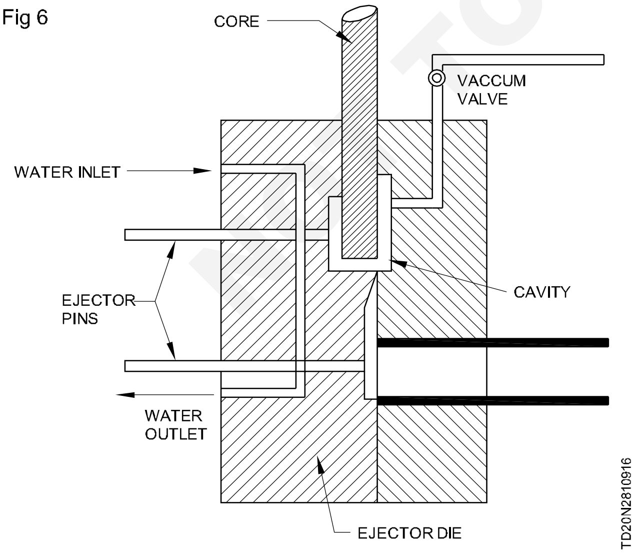
9 **Lubrication:** The correct lubrication for the sleeve and plunger is very important because it affects the quality of casting and the life of the shot sleeve and plunger.

Process variables (Die): The process variables for die are

1 **Temperature:** Die temperature is governed by casting rate, cooling water flow and molten metal temperature. Die temperature is 230-240°C for zinc and 340-355°C for aluminum. Cooling channels in the die do increase the tool cost and the complexity of the die but they are cost effective for high production rate, high volume jobs.

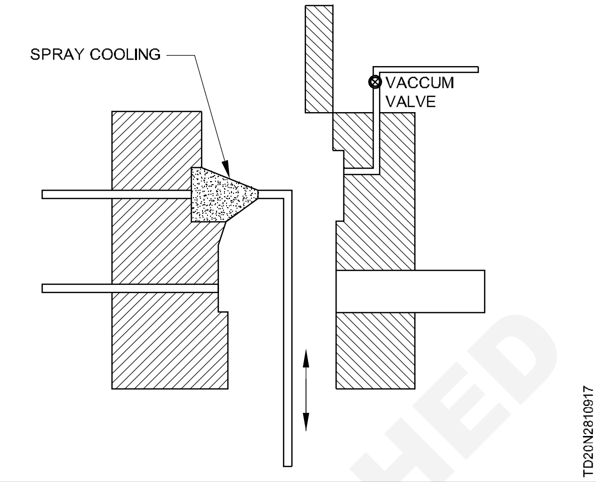
Cooling channels in the die (Fig 6)

Fig 6



Dies are also cooled between shot by spraying cooling solutions on the face of the die. These solutions also act as lubricants for casting separation and ejection. (Fig 7)

Fig 7



2 **Filling rate:** Filling rate can be considered in a terms of

a **Cavity fills time:** It can be raised from 20-60 milliseconds and most favorably 40 milliseconds.

b **Cavity filling rate:** It is expressed in cc/seconds.

c **Gate velocity:** The values of gate velocity can be 28- 55 meters/seconds.

Clamping system (Fig 8): The heart of any die casting machine is the clamping system.The hydraulic die closing cylinder straightens the toggle links to close the die. This arrangement achieves high die clamping force and rapid die opening and closing action. Because of the tremendous pressures used to inject the molten metal into the die, the force required to hold the two die halves together is very high. This holding force is accomplished with the tie bar-platen toggle type machine construction which opens and closes the die rapidly to achieve high production rate.

Sprue spreader: The sprue spreader performs several functions. The first is to spread the incoming stream of the molten metal. The flow is divided. At the same time, the mass of the flow stream is reduced to runner thickness and turned into a direction of travel parallel to the dies parting plane. The next function is to help remove heat to solidify the metal in the sprue. Finally the spreader acts to pull the sprue out of the bushing as the die opens.

Commercial sprue spreader is shown in Fig 9. Spreaders are available with either straight body or tapered body. The cooling water must be directed in to the tip of the spreader. An internal baffle or all around cooling method can be used.

Fig 8

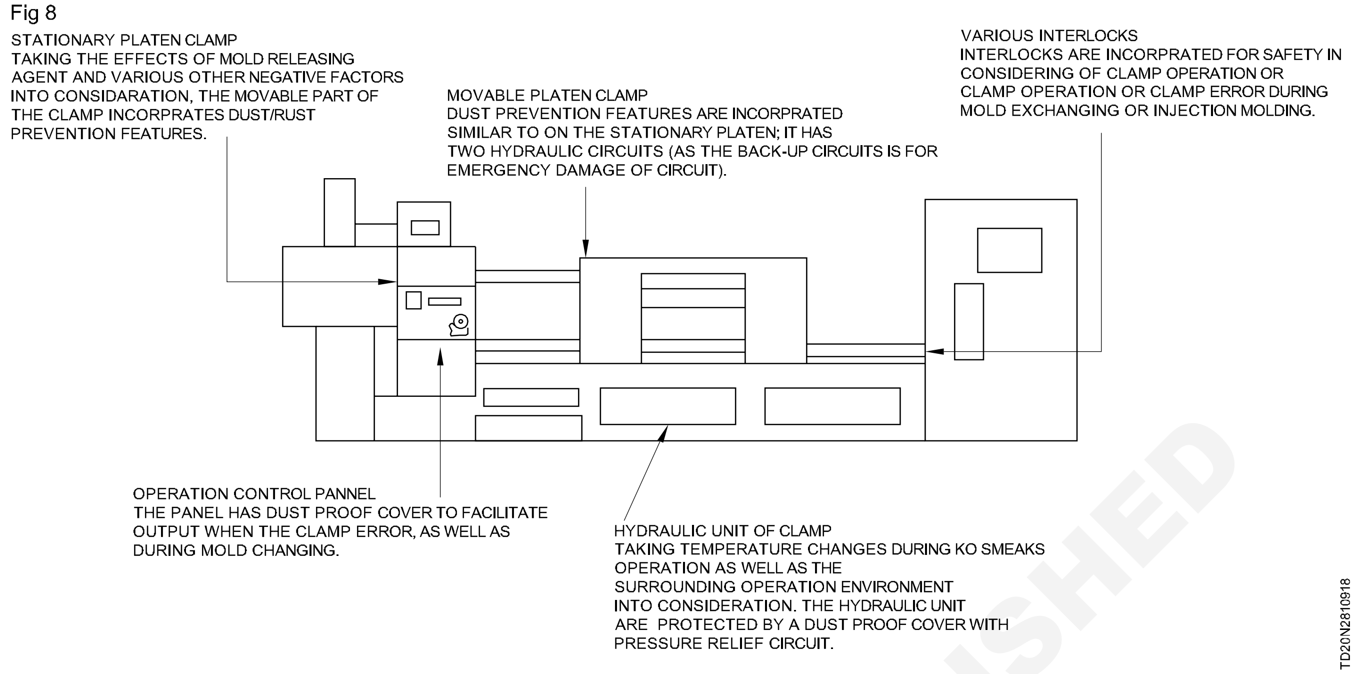
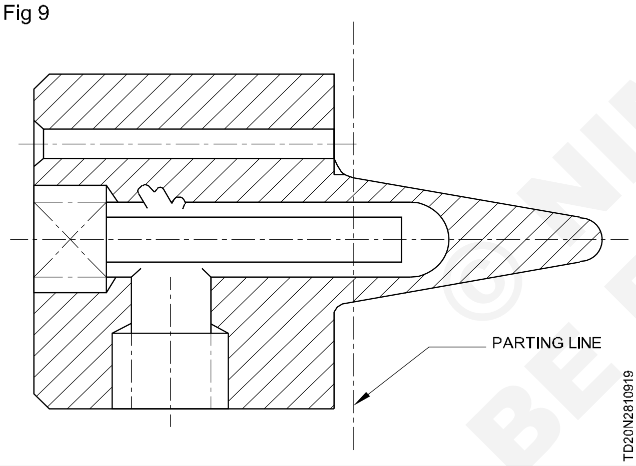
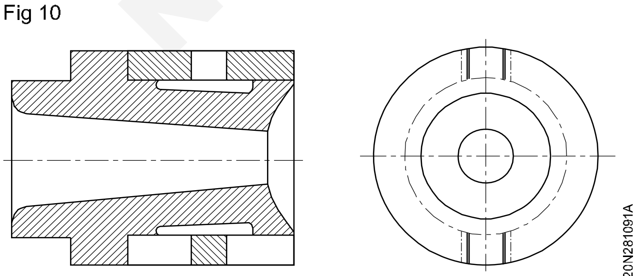


Fig 9



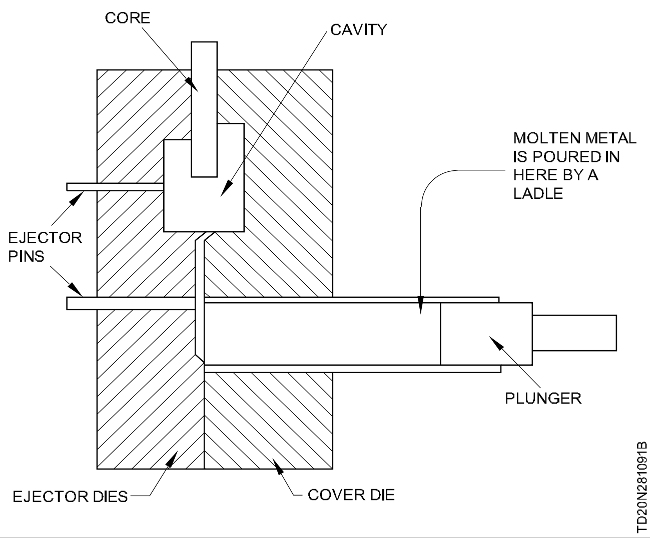
Sprue Bushing (Fig 10): Sprue bushings for Zinc casting are made of heat treated steel. They are made of two piece assembly, brazed all-round for maximum water cooling capacity. The spherical seating is polished and precision ground. The radius is slightly bigger than the nozzle radius. The starting diameter either equal or 1.2 - 1.75 times the area cross section of the gate. The taper is normally between 5-15°/side

Fig 10



Shot sleeves (Fig 11): The cold chamber machines require shot sleeve. Usually the shot sleeve is not considered part of the die. The die is made to fit over it. In those instances, the thickness of the die must be made to accommodate a specified sleeve. The out side diameter of the shot sleeve is ground to specified dimension. That dimension should be given to the die designer. The hole in the die should be a slide fit to the shot sleeve for about 25mm from the parting line. The reminder of the hole is relieved. The shot sleeve is retained by a shoulder that is secured between the die and the machine platen. The shoulder can be seated into the machine platen or into the die or both the machine platen and the die. The die designer will not usually have any choice as to the shot sleeve construction but usually will usually have to accommodate a system that is already in use. Normally a die is designed with a hole to accept the shot sleeve.

Fig 11



Gating system design

Introduction

The gating system is a combination of various channels and pockets machined into the parting surfaces of the die which allow the molten metal to fill the impression. The metal enters the die through sprue, runner and gate. The metal flows through the cavity and out the opposite side through smaller gate and into the pockets called overflows. In all situations, the liquid metal should flow through the cavity in parallel streams. Each flow stream should travel the shortest possible distance in the casting and all should reach the opposite side of the cavity at the same time.

The molten metal in the gating system solidifies along with the part and is removed when the die is opened. The casting along with the part, sprue, runner, gate and overflow is known as shot.

The various steps involved in gating system design are

1 Planning the flow patterns

2 Calculating filling time

3 Selecting gate velocity

4 Determining minimum and maximum gate depth

5 Striking a balance between machine power and gate size

6 Calculating plunger size and speed

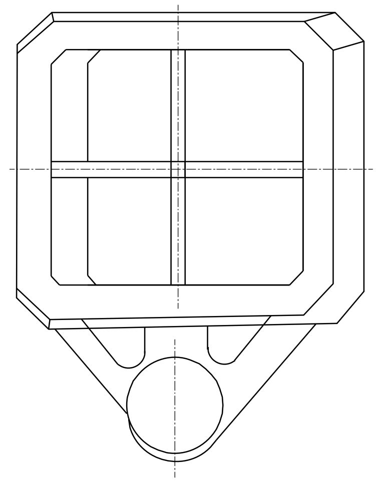
7 Calculating gate and runner area and geometry.

Planning flow patterns: Control of metal flow in the die is a key factor in producing sound castings. Metal must flow uniformly rapidly into the die minimizing turbulence and entrapped air. The first step in planning a gating system is to decide how the metal should flow through the cavity. The molten metal should flow in the direction of the shortest dimension of the cavity. The flow paths should be parallel or diverge from each other. All parts of the component should be filled simultaneously. If the purpose is to disperse shrinkage porosity evenly then the molten metal should enter the cavity where the part has the thinnest portion and flow into thicker regions. This method is known as shrink dispersion process. Most die castings with thin walled sections are filled by this process. If the plunger pressure is to be used for feeding metal into the shrinkage voids, then the molten metal should enter the thickest portion and flow into the thinner regions. This method is known as shrink feed process. Only small and thick castings can be made by this process.

When the molten metal must travel a long distance through the thin cavity it is best practice to add additional runner and gate to fill the cavity. The flow distance through a thin cavity should be less.

Runner system (Fig 12): The runner system consists of a collection of channels which carries the molten metal from the nozzle in to the cavity. A key feature in die design is the positioning of runner and gates.

Fig 12



Sprue spreader (or sprue pin): The first component in the runner system that the molten metal encounters is the sprue spreader. It enters the sprue bush when the mold is closed. It ensures a leak free sealing with the moving and fixed half of the die. The sprue spreader directs the molten metal towards the runner. The spreader is also designed to direct the metal along the centre line of the runner. The sprue pin is also used to pull the sprue out of the sprue bush. When the metal shrinks as it solidifies it holds on to the pin. So it takes away the sprue from the sprue bush.

Runner: A runner is the part of the runner system which begins from the sprue and ends at gate.

Gate (Fig 13&14): A small opening which connects the runner with the cavity is known as gate

Fig 13

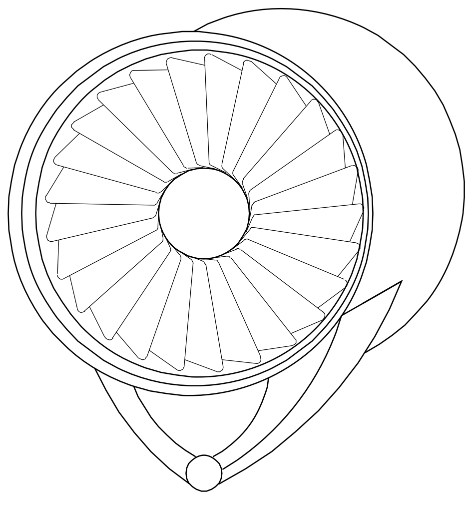
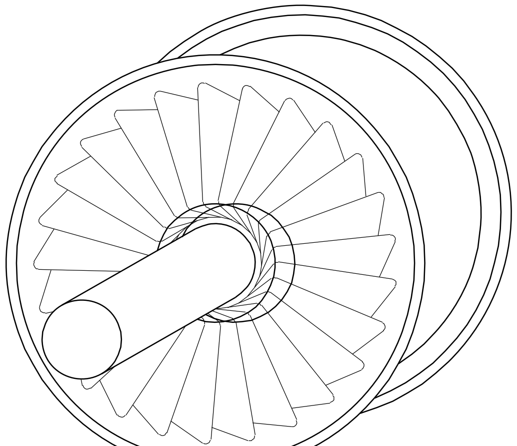


Fig 14



CENTER SHOT GATE

TD20N281091E

Gate Runners: The gate and the adjacent runner together are known as gate runner.

Types of gate runners: Gate runners are of two types namely fan type and tangential type. Both the fan and tangential gate runners match the runner where they connect to it.

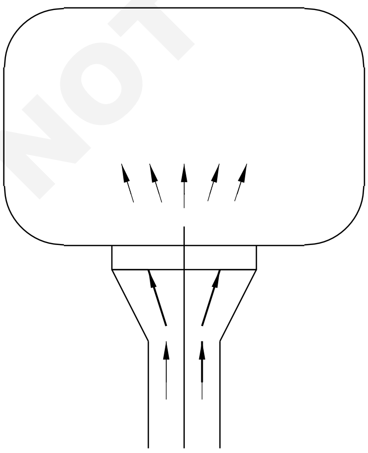
The fan gate runner becomes thinner and wider as it gets close to the cavity until it is as wide as the gate width and as deep as gate depth. The metal flow is spread out such that the flow streams are diverging as they enter the cavity. The flow is neither parallel nor perpendicular to the edge of the cavity. In fact it creates two directional flows. One flow at the center which is perpendicular to the gate and the other on either side is at oblique angle to the cavity edge. The flow speed is high at the center and less at the sides. This means the front moves more quickly at the center than at the sides. Thus the fan causes preferential filling if the part is filled is filled from the centre of the gate.

As a general rule fan gates are used for filling small parts. There are two types of fan gates

1 Flat cross section (Fig 15)

2 Curved cross section (Fig 16)

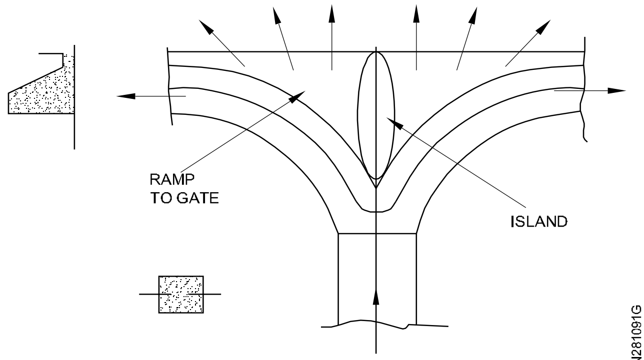
Fig 15



FLAT CROSS SECTION

TD20N281091F

Fig 16



CURVED CROSS SECTION

TD20N281091G

The tangential gate runner carries the flow of molten material along the edges of the cavity. As the metal travels along the edges of the cavity it passes side ways through the gate into the cavity. The result is that the metal can be made to enter the cavity side ways in parallel flow streams. The flow streams will not be perpendicular to the edge of the cavity in this case.

A tangential gate runner must have regularly decreasing cross section (from upstream to down stream). This taper causes increased pressure in the runner and assumes regular flow rate along the length of the gate.

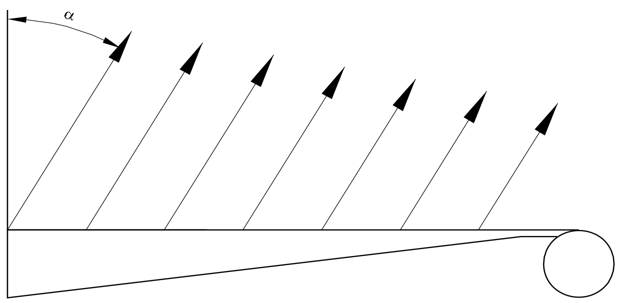
The principle advantage of tangential runner is that they can be circulated to give good control of the flow direction and flow rate of the metal which is difficult with the other form of runner.

There are two types of tangential runners.

1 Single tangential runners (Fig 17)

2 Double tangential runners with delta. (Fig 18)

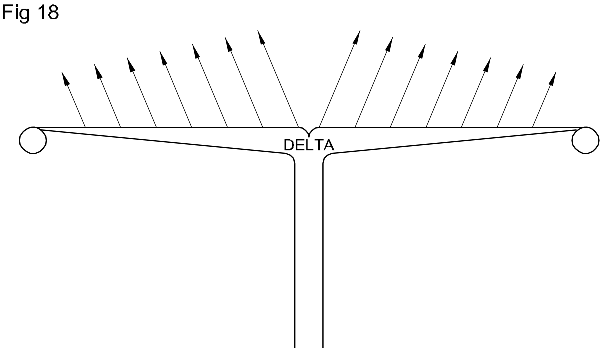
Fig 17



SINGLE TANGENTIAL GATE RUNNER

TD20N281091H

Fig 18



DOUBLE TANGENTIAL RUNNERS WITH DELTA

TDN291351I

Shock absorber: A shock absorber is the part of the runner system provided at the end of the gate runner. Its purpose is to

1 Receive the cold metal and the impurities.

2 Trap air that will compress and thus absorb any impact of the cold metal reaching the runner end.

3 Maintain filling angle of the metal when it reaches the end of the gate runner.

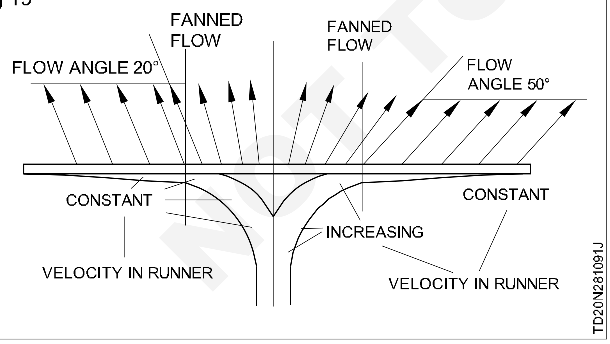
4 Minimize the effect of any large increase in the speed of the metal at end of the runner due to tapered cross section of the tangential runner.

5 Can position an ejector pin

Casting is first divided in to different segments of equal volumes so that all gate length required becomes equal. The metal must flow through the segment and into the overflows. The cavity and the adjacent over flows must be filled at the same time with all other segments of the cavity. The shape of the segment is either trapezoid or parallelograms. When the part can not be divided into these shapes a closer shape should be selected. An over flow can be some times shaped to get the required shape for the segment If the shape is trapezoid or shaped more like it a fan gate is the ideal choice. When segment is shaped more like a parallelogram a tangential gate is likely to be the best. Each segment is supplied with a single gate and the volume including the over flow is calculated to find out the percentage of metal required to fill them. The gate size when calculated should be proportional to the segment volumes.

Flow paths (Fig 19): The route of travel of the material is known as the flow path. The angle between the actual direction of metal flow and a line perpendicular to the edge of the cavity is called the flow angle. The boundaries between segments are some what arbitrary, but it should match with the values of the flow angle.

Fig 19



Overflows (Fig 20&21): Over flows are small pockets machined adjacent to the cavity and connected by a gate which is filled along with the cavity.

Fig 20

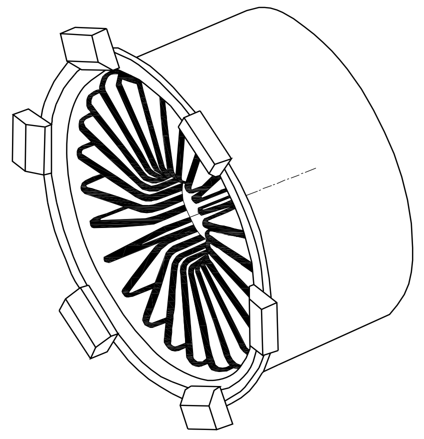
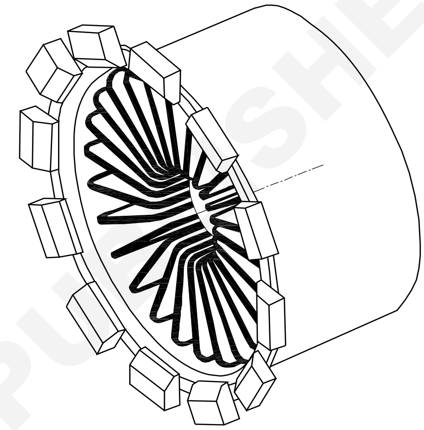


Fig 21



The important functions of over flows are

1 They are reservoirs that receive and hold gases that are flushed out of the cavity.

2 They are reservoirs for cold and oxidized metal that initially flows through the cavity.

3 They provide additional heat input to the die, thereby helping to maintain a satisfactory and stable die temperature.

4 They serve as locations for contact of ejector pins when the ejector pin marks are not permitted on any surface of the casting.

5 They may be provided to balance the demand on the gate so a uniform gate thickness can be used.

6 They are some times shaped and sized to provide handles for holding the cast shot, guide for handling equipment and locators for positioning the shot for secondary operations.

7 They can counter balance the trimming stresses. When over flows are positioned opposite to the gate they allow trimming stress to be counter balanced.

8 Over flow can be used to balance multi cavity dies.

The main function of overflows is to carry first metal past a casting edge and leave on the edge only hot metal capable of making smooth, intimate contact with the die surface. The first metal entering a die cavity carries with it a high concentration of oxides, release residue, and injection turbulence froth. It also carries with it foiled metal that is, thin skins of metal formed by contact with die surfaces. If this foiled metal is permitted to deposit itself in any one area of the cavity, it forms cold shuts causing poor finish. It is desirable and usually necessary to carry this 'first' metal through the cavity and into an overflow. The overflows are connected to the casting by overflow gates, and the location and conformation of these gates are important. The gates usually range in thickness from 0.5-0.8mm. The thickness is determined by the speed with which it is desired to push off first metal. Width of the overflow gate must be enough to give the area of all overflow gates a total of 60 to 75% of the gate area. This width permits exit of first metal, gas and air.

In the event that all overflows are located in the area of first fill of the cavity, additional overflows must be used to provide sufficient vent area to vent the die properly. It usually is good practice to place ejector pins in the overflow area to facilitate clean ejection of the overflow. The overflow also should have generous draft, especially on the casting side, to prevent shrink toward the greater volume of the casting from sticking the overflow.

Overflows often can be used for other purpose, as well. They may be used to eject the casting without ejector pins in the finished casting itself. This practice requires an overflow gate sufficiently strong to eject the casting without distortion. Overflows also often are formed so as to serve as hooks or attachments for carrying the casting on conveyor systems. Another use is for location in secondary operations. It is poor practice to connect the overflow outside the casting. This practice leads to backfill from the overflows into the casting area. As long as it is fluid, metal will of course follow the line of least resistance. When overflows are connected, the metal often will flow from one area of the casting through one overflow, into another overflow, and back into another area of the casting. This backfilling causes two metal fronts to meet and try to weld within the casting area. It creates rough casting surfaces and internal porosity. If overflows must be connected, place chokes in the connecting runner. It must be remembered that overflows are a definite part of the die casting system. Use them according to the rules, not on the basis of 'art'.

Venting: Vents or air escape passages on the die faces usually lead out of overflows, but may lead from any point on the cavity or runner parting line. Total vent cross sectional areas should be at least 50 percent of the gate area. Self cleaning of air escape passages can be insured by keeping the thickness of vents leading from overflows and cavities to approximately 0.5 to 0.6mm. After a distance of 40 to 50 mm, the thickness is decreased to between 0.1 to 0.2 mm to cause the metal to freeze before spitting occurs.

Calculating filling time

The second step in the gating system design is to calculate the filling time. It is the time required to fill the cavity and the overflows. Die casting dies are filled with 10 to 60 milliseconds.

Gating system and machine parameters are designed to meet the required filling time. Large runners, gates and higher plunger speeds give shorter filling time.

The ideal filling time for a given condition can be calculated using the formulae given below.

$$t = \frac{KT(t_i - t_f + SZ)}{t_f - t_d}$$

Where:

t = the ideal filling time (sec)

K = derived constant(s/mm)

t_i = metal injection temperature (°c)

t_f= minimum flow temperature (°c)

t_d= temperature of the cavity (°c)

s = percent solid fraction allowable in the metal at the end of filling (%)

z = units conversion factor (°c/%)

T = Average part thickness

The equation given above gives the best filling time. If the processing conditions are changed the equation gives different value, but the best filling time for those conditions.

An alternate formula for calculating the cavity fill time is

$$t = 0.52 \times \frac{(Tg - Tliq + 46.1°C)}{0.6\,(Tg - Td)} \times T$$

Where

t = Optimum filling time in sec.

Tg = Metal temperature at gate in °c

Tliq = Liquid's temperature of metal in °c

Td = Temperature of die cavity surface before shot in °c

T = Casting wall thickness in mm.

Metal injection temperature: The metal injection temperature (ti) in the gating equation is the temperature of the molten material when it reaches the cavity. In hot chamber process the holding furnace temperature and the metal injection temperature are usually the same. In cold chamber process the injection temperature is considered to be 8 to 10°c less than the holding temperature.

Minimum flow temperature: the minimum flow temperature (t_f) in the gating equation is the flow temperature near the solidus.

Percent solid fraction allowable in the metal: the percent solid fraction allowable(s) in the metal defines the amount of solid particle that the casting can accommodate during cavity filling which will not effect the surface quality of the part being produced. Generally 10 to 15% solids give better surface finish. But greater percentage (50%) will result in less internal porosity. The percent of solid fraction also affect the intensification pressure which is applied to compensate for the shrinkage. However large values of "s" (30 to 40%) will reduce the level of shrinkage porosity. There fore pressure intensification is not required. Thick casting may benefit from the large values of "s" where as a smaller value is good for thin casting.

Units of conversion: The units of conversion "Z" is derived from the thermal properties of the alloy. So its value is a constant.

Die temperature: Die temperature (td) used in gating equation is 30°c more than the surface temperature. With higher temperature filling time can be increased which reduces plunger speeds. Higher die temperature also extends die life. The disadvantage of high die temperature is longer cycle time.

Part thickness: Casting thickness is an important factor in the gating equation. If the part thickness doubles the ideal filling time doubles. Although castings very rarely have uniform thickness its thickness at maximum place is considered. If a casting has a thin or thick region that is far away from the gate then that thickness is considered. Other wise it is best to use the average thickness.

Table value for filling time calculations

Variables for Filling time calculations							
Alloy	**Empirical constant(k)**			**Metal Temp(Ti)**	**Mini. Flow Temp(Tf)**	**Die Cav Temp(Td)**	**Solidus Factor(z)**
	P20	H13/21	W				
	Sec/mm	Sec/mm	Sec/mm	°c	°c	°c	°c
Mg		0.0252	0.124	650	510	340	2.5
Al360,380,384		0.0346	0.124	650	570	340	3.8
Al390		0.0346	0.124	720	595	355	3.8
Zn 12,27	0.0312	0.0346	0.124	565	445	260	3.2
Zn 3,5,7	0.0312	0.0346	0.124	405	382	230	2.5
Fe		0.0346	0.124	1540	1370	980	6.0
Cu 60/40		0.0346	0.124	955	900	510	4.7
Cu 85-5-5-5		0.0346	0.124	1035	930	515	4.7
Pb	0/0156	0.173	0.124	315	280	120	2.1

Problems: A casting is to be made with the following process conditions. Calculate the ideal filling time?

Ti = 405°c

Tf =382°c

Td = 230°c

S =15%

Z = 2.5°c/%

T = 2mm

K = 0.0312

Answer

$$t = \frac{0.03122\ (405-382) + 15\times 2.5)\ 2}{382-230}$$

= 0.02483sec

Problem: A Zn 12 alloy casting has to be made in a die made out of H13 material. Find out the ideal filling time from the following specifications?

Ti = 565°c

Tf =445°c

Td = 260°c

S =20%

Z = 3.2°c/%

T = 2.5mm

Answer

$$t = \frac{0.0346\ (565-445) + 20\times 3.2)2.5}{(445-260)}$$

= 0.086sec

Selecting gate velocities: Selecting gate velocity is the third step in the design process. The gate velocity is the speed at which the molten material enters the cavity. It is expressed in meters per second.

Typical gate velocity for various materials

Alloy	Typical gate velocity m/s
Mg	42.0
Al360,380,384,390	38.7
Zn 12,27	29.0
Zn 3,5,7	25.0
Fe	25.0
Cu60/40	22.50
Cu85/5/5/5	22.0
Pb	20.0

Recommended gate velocity as per wall thickness

Thinnest wall thickness Any where in the casting(mm)	Gate velocity ranges for Zn (m/s)	Gate velocity ranges for Al (m/s)
0.76	46-55	48.7
1.27-1.52	43-52	47.0-45.7
1.90-2.29	40-49	44.0-43.2
2.54-2.80	37-46	42.6-41.7
2.86-3.81	37-46	44.0-43.2
4.56-5.08	31-40	38.0-36.5
6.30	28-35	33.5

Flow rate: The volume of material (cm^3) entering the cavity per second is known as the flow rate. The flow rate (Q) is determined by the Whole volume (V=part+ overflow + vent) divided by the ideal filling time (t).

$$Q = V/t$$

The flow rate "Q" is also equal gate velocity multiplied by the area of cross section of the gate.

$$Q = G_v \times G_a$$

There for the gate area can be found out once the gate velocity has been selected. The factors which influence the gate velocity are atomized jet flow, runner and gate size and the machine power.

Atomized Jet flow: Atomization means converting into fine particles. The molten material as it leaves the gate it should be in the form of very fine particle. The best castings are made with atomized jet flow. To achieve this certain amount of velocity is required. The minimum value depends upon the gate depth.

Runner size: Smaller runners require greater velocities. Runner size is inversely proportional to the square of the velocity. For greater velocity higher machine power is required. Less power machines are provided with larger runners. Larger runners increase the cost of re-melting.

Gate depth: Gate depth depends upon the atomization of metal flow required. For a lesser gate depth greater velocity is required for atomization.

Establishing minimum and maximum gate depth: Selecting gate depth is the fourth step in the design process. The gate depth is some times known as gate thickness. It is one of the critical dimensions that must be established by calculations. The final selection of the gate must satisfy the conditions such as atomization of metal flow, gate vestige, degating easiness etc.

The minimum gate depth (Fig 22): The minimum gate depth depends up on the gate velocity. But it is seldom kept less than 0.4mm. The minimum gate depth can be calculated using the following equation.

$$M_{sg} \times G_d \times G_v^{1.71} = J$$

Where

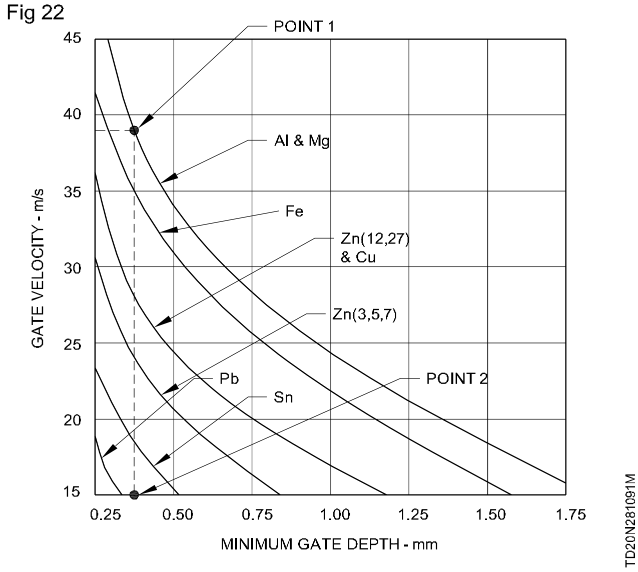
M_{sg} = Density of the alloy

G^d = Gate depth

G_v = Gate velocity

J = Derived constant for the alloy

Fig 22



Gate velocity and J value for different alloys

Alloy	Gate velocity m/s	Constant (J)
Mg	42	360
Al 360,380,324,390	38.7	525
Zn 12, 27	29	624
Zn 3,5,7	25	624
Fe	25	1314
Cu 60/40	22.5	985
Cu 85/5/5/5	22	985
Pb	20	394

Very good casting can be obtained if the metal is made to enter the cavity in the atomized condition. Atomization can be ensured if the intersection of gate velocity and gate depth falls on the right side of the curve.

Maximum gate depth: The maximum gate depth is governed by mechanical constraints. The thickness of the gate can not be more than the thickness of the part. Generally it is kept below 80% of the part thickness. If the gate thickness is excessive trimming becomes difficult. It also leaves a bigger witness mark on the part. If the gate has to be removed by manual bending or twisting it is better to have thin gates.

Matching the power required by the gating system with the available machine power: Matching the gating system to the power available from the machines injection system is the fifth step in the design process. The power required by the gating system can not be more than the power available from the machine's injection system. There are many things that can be done to match the gating system to the power available from the machines injection system. The machine power should be used to give maximum shot yield. Shot yield is the ratio of the weight of the usable casting to the actual weight of the shot. If a die has to be loaded on lesser power machine we have to unnecessarily increase the runner size and over flows. This will reduce the shot yield.

Many a times the machines are underutilized. Die may be provided with too large runners and overflows so that high power is not required at all. Some times the die flashes when the excess power available is used. Flash can be some times controlled just by changing the plunger diameter to bigger size than increasing the runner and overflows.

With the proper gating system built into the die, it becomes the function of the machine to drive the metal through it therefore proper selection of the plunger diameter and the pre-fill stroke are necessary to achieve the desired machine performance.

$$P_{stl} = L - \frac{4v}{\Pi P_d^{\,2}}$$

Where,

Pstl. = pre-fill stroke mm.

L = Total available shot stroke mm.

Vs = Volume of metal ladled cubic mm.

Pd = Plunger diameter mm.

Gating system power: The power required by the gating system is actually the product of flow rate and metal pressure.

Power $(G_p) = QxP_m$

The flow rate $Q = \frac{V}{t}$

Where

Q = flow rate

V = volume of part including overflow and vents

t = ideal filling time as calculated earlier

Metal pressure is related to the gating velocity by the following equation:

$$P_m = \frac{M_{sg}}{2g}\left(\frac{G_v}{C_d}\right)^2$$

Where

P_m = Pressure that must be applied to the molten metal to push the metal through the die's gating system

M_{sg} = Density of the molten metal

g = Acceleration due to gravity

G_v = Gate velocity

C_d = the discharge coefficient or efficiency of the gating system (0.6 to 0.8)

$$\text{Discharge coefficient} = \frac{\text{Actual flow rate}}{\text{Idealflow Rate}}$$

The value for C_d is taken as 0.6(pessimistic value) when one expects that the metal flow will not be smooth. I.e. when there are sharp corners and varying cross sectional areas for the metal to flow though. The value is taken as 0.8 (optimistic value) if one expects the flow will be smooth.

Problem: Calculate the pressure on metal during cavity filling if an Al casting is made with gate velocity 38.7m/s?.

$$P_m = \frac{M_{sg}}{2g}\left(\frac{G_v}{C_d}\right)^2$$

$= 2.7/2x9.8\ (38.7/0.6)^2$

$= 0.138x4160.25$

$= 5.73MP_a$

Machine power: The available machine power is given by the PV^2 diagram (P = Equivalent hydraulic pressure and V= Equivalent plunger velocity). If there is no molten metal in the shot sleeve, there will be no resistance to the motion of the shot cylinder piston other than those resistances of the mechanical elements. In that situation the plunger velocity will be at the maximum because all the pressure available in the accumulator (hydraulic pressure) is used for moving the piston. That means at the maximum velocity of the plunger all the hydraulic pressure is utilized by the machine systems. There is no pressure available to push the molten metal through the die's gates when the system is moving at that velocity. This velocity is called the Dry shot speed (when the fast shot speed control valve is fully open) and is denoted by the symbol P_{vd}. The dry shot speed is readily measurable. Dry velocity and the accumulator pressure (also readily measurable) are the critical parameters in establishing the machine's power performance.

There are two ways to cause the plunger to move slower than the maximum velocity. One method is to partially close the fast shot speed control valve. The other method is to provide resistance to the motion of the plunger. The molten metal being pushed in to the die provides such a resistance to the motion of the plunger. As the cavity is being filled the resistance to the motion of the plunger increases and the plunger velocity comes down. As the load increases, the pressure (P_h) in the shot cylinder must increase to cause a force on the piston equal to the resisting load.

At maximum velocity (P_{vd}) the effective pressure in the shot cylinder is zero. As the velocity of the system decreases the pressure in the shot cylinder increases until the velocity is zero and is equal to accumulator pressure. The relation ship is given by the equation given below.

$$P_h = P_{hs} (1- Pvm/Pvd)^2$$

Where

P_h = Effective pressure in the cylinder. (Required equivalent Hydraulic pressure to fill the cavity)

P_{hs} = Hydraulic system pressure (accumulator)

P_{vm} = Velocity of the plunger when pushing metal through the die

P_{vd} = Dry speed velocity of the plunger.

The PV^2 Diagram pressure line (Fig 23): The PV^2 diagram is a device used to compare the machine power to the power requirement of the gating system. In a die casting machine the system pressure P_{hs} and the dry shot speed P_{vd} are constant once the systems pressure relief valve is set. So the effective pressure in the hydraulic shot cylinder P_h can be calculated for any plunger velocity P_{vm}.

From the above equation it can be seen that if the plunger velocity equals dry speed velocity (P_{vm} / P_{vd} =1) the effective hydraulic pressure becomes zero. Similarly if the plunger velocity is zero the effective hydraulic pressure becomes equal to the system pressure. The PV^2 diagram provides a way to show and use the above equation in the graphical form.

The PV^2 diagram is constructed by plotting the systems hydraulic pressure on the pressure scale and the square of the dry shot velocity on the velocity squared scale. The diagonal line through the graph shows how much hydraulic pressure is available for pushing of the molten metal for any plunger velocity. Each machine has its own PV^2 diagram. It shows the power in terms of the hydraulic pressure available to do work (i.e. push molten material) for any given plunger velocity. The work that is to be done is the displacement of the molten metal. So to have a common basis for comparison the power components of the gating system namely flow rate and metal pressure should now be converted into equivalent plunger velocity and hydraulic pressure respectively. So as a first step the flow rate is converted in to the equivalent plunger velocity by the following equation.

$$P_{vm} = Q/\pi \times (p_d/2)^2$$

Where

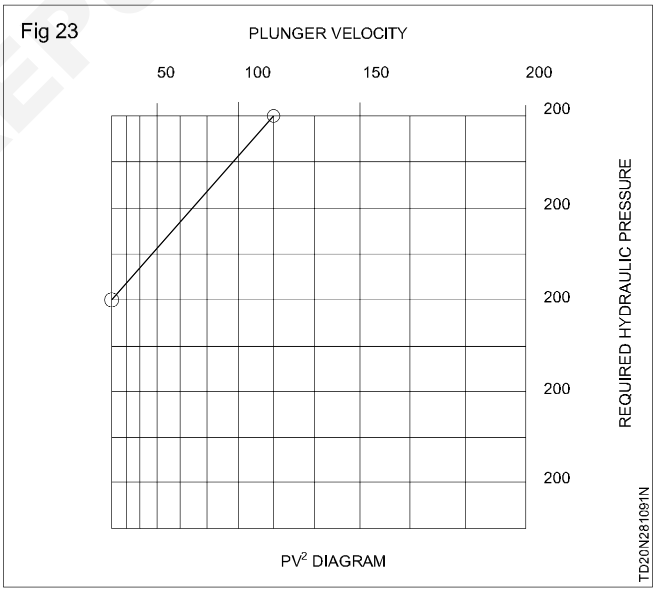
P_{vm} = Plunger velocity (m/s)

Q = Flow rate (cm^3/s)

P_d = Plunger diameter (mm)

The graph below gives an alternative to the above calculation. The flow rate is plotted on y axis. The flow rate is projected horizontally across the graph. The intersection of that projection and the curve for the plunger diameter gives the required plunger velocity.

Fig 23



PV^2 diagram

Problem: Calculate the plunger velocity if the flow rate required is 82000Cm^3 /s and plunger diameter selected is 63.5mm.

$$P_{vm.} = Q/\pi \times (p_d/2)^2$$

$$= \frac{82000X4}{3.14x6.35^2}$$

25.892m/sec

The next step is to convert the second component of the power requirement i.e. metal pressure into equivalent hydraulic pressure. The metal pressure can be converted into the equivalent hydraulic pressure by the following equation.

$P_m = P_{hd} (C_D/P_d)^2$

Where:

P_m = Metal pressure (MP_a)

P_{hd} = Hydraulic pressure acting on the shot piston during cavity filling. (MPa)

P_d = Plunger diameter (mm)

C_D = Hydraulic Cylinder diameter (mm)

Problem: Calculate the hydraulic pressure required if the pressure required on metal is 5.17 MP_a, Cylinder diameter 127mm and plunger diameter is 63.5mm respectively?

$P_{hd} = P_{md} (P_d/C_D)^2$

$= 5.17(63.5/127)^2$

$= 5.17 \times 0.16$

= 1.296 MPa

Once the flow rate and metal pressure have been converted into equivalent plunger velocity and hydraulic pressure it can be compared with the hydraulic pressure available to actually push the molten metal and the plunger velocity that can be achieved with that hydraulic pressure from the PV^2 diagram. PV^2 diagram defines the power curve of the die casting machine's injection system. It shows the power in terms of the hydraulic pressure available to do work (i.e. push molten metal) for any given velocity.

If the required plunger velocity line and the required hydraulic pressure line intersect slightly on the left side of the machines power line then that combination is feasible. If it intersects on the right side of the power line then that combination is not feasible.

PQ^2 Diagram: The PQ^2 is another important tool for die casting design process. It was originally developed by CSIRO Australia in 1977 although its basis can be found in PV^2 which was developed in 1975 by NADCA America. PQ^2 Diagram defines the work that the machine can do in terms of the metal flow (displacement) rate and metal pressure. For any given machine the relationship between displacement rates and metal pressure depends strongly on the plunger diameter. If the plunger diameter is established by the PV^2 diagram then it can be plotted as displacement rather than velocity diagram. The plunger velocity times the plunger area of cross section is the displacement. Once the velocity is converted into displacement rate scale and the effective hydraulic pressure into metal pressure the diagram obtained is known as PQ^2 Diagram. The conversion is given as follows.

$$Q = P_{vm.} \times (\pi \times pd^2)/4$$

Where

Q = Flow rate (cm^3/s)

P_{vm} = Plunger velocity (m/s)

P_d = Plunger diameter (mm)

P_{md} = Phd (CD / Pd)2

Where

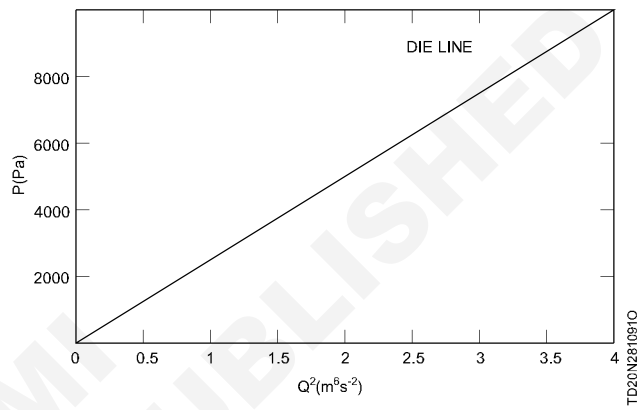
P_{md} = Metal pressure (MP_a)

P_{hd} = Hydraulic pressure (Accumulator pressure area of cylinder) acting on the shot piston during cavity filling. (MP_a)

P_d = Plunger diameter (mm)

PQ^2 Diagram showing the Die line. (Fig 24)

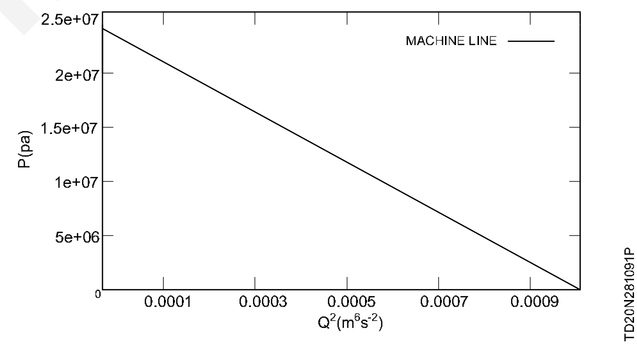
Fig 24



Die Line: A Line which expresses the pressure requirements for the die at any flow requirement.

PQ^2 Diagram showing the machine line. (Fig 25)

Fig 25

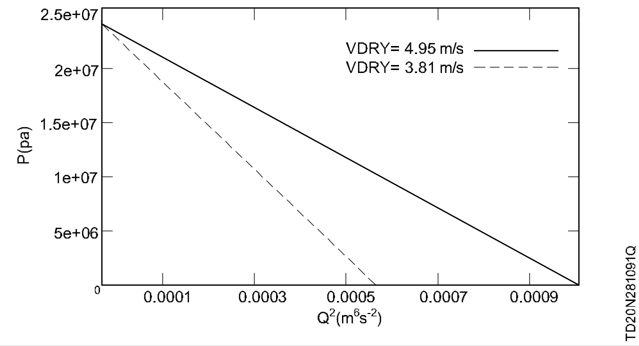


Machine Line: A line which expresses the die casting machines capabilities for any flow rate between zero and dry shot flow rate. The machine line is affected by factors such as the accumulator pressure, the shot speed control valve aperture and the diameter of the plunger.

PQ^2 Diagram showing the effect of different dry shot velocities on the machine line. (Fig 26)

Maximum flow rate: If the die line and machine line are constructed on the same PQ2 Diagram the intersection between these two lines determine the maximum flow rate that can be achieved with this die casting machine in order to produce the casting represented by the die line. The machine line described above is defined for a specific plunger diameter. This approach can be generalized if different machine lines are drawn for the same machine.

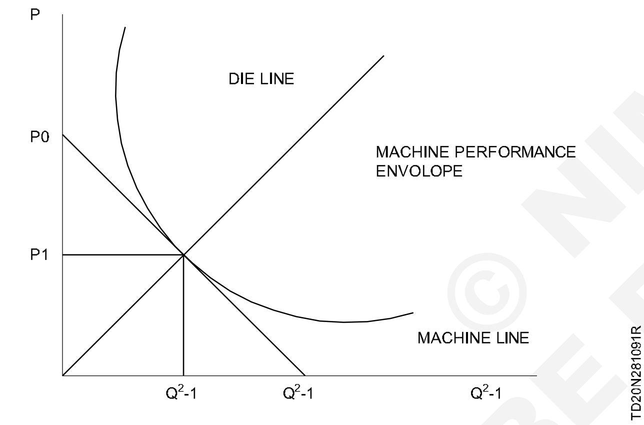
Fig 26



This infinite number of lines defines an envelope curve which has one tangent point for each machine line. A machine performance envelope curve plotted on PQ2 Diagram is shown Fig 27. It shows how the pressure on the plunger face decrease as the flow rate increases.

The Fig 27 shows a tangent point between machine power envelope and the machine line. The point P1 determines the maximum pressure that the die casting machine can provide for a flow rate Q1

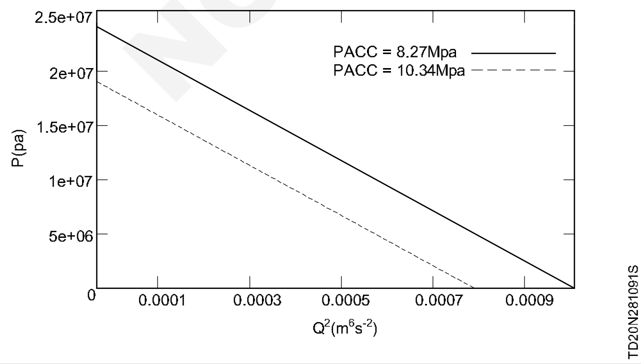
Fig 27



Operational window: The establishment of boundaries on relevant variables in given process defines an operational window (OW). It usually guarantees a known behavior of the process with in its limits and the violation of such boundaries would give unexpected results.

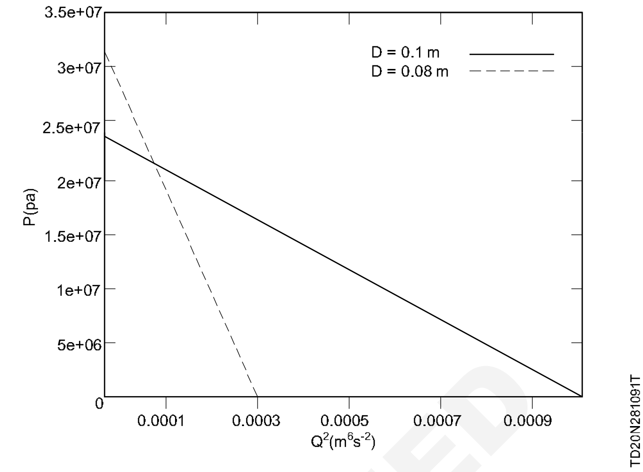
PQ^2 Diagram Fig 28 showing the effect of accumulator pressure on the machine line.

Fig 28



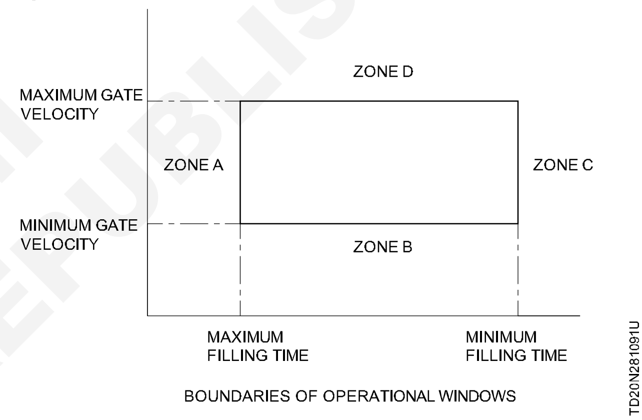
PQ^2 Diagram Fig 29 showing the effect of different plunger diameters on the machine line.

Fig 29



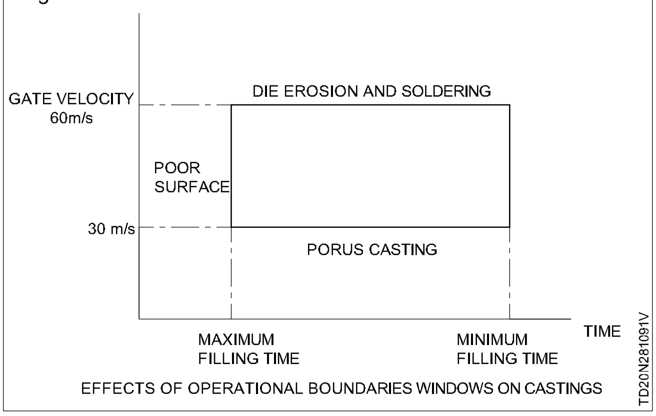
Boundaries of the operational window (Fig 30)

Fig 30



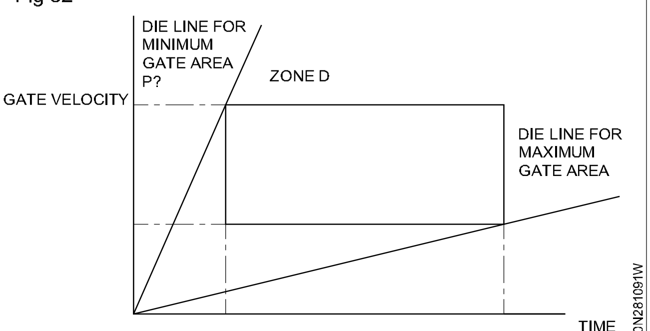
Effect of operational Boundaries windows on castings (Fig 31)

Fig 31



Extreme values for the gate area (Fig 32)

Fig 32



EXTREME VALUES FOR THE GATE AREA

Determining plunger size and velocity: The sixth step in designing the gating system is to select the correct plunger diameter and velocity.

There are many combinations of plunger diameter and plunger velocity that will satisfy the filling time requirement as given by the previous equations.

$Q = P_{vm.} x (\pi x pd^2)/4$

Where

Q = Flow rate (cm^3/s) (Q=v/t)

P_{vm} = Plunger velocity (m/s)

P_d = Plunger diameter (mm)

Some times the machine power also limits the selection of plunger diameter. As already discussed, machine power is the product of flow rate and metal pressure. If the machine power is marginal only one plunger diameter is possible. But usually there is a range of plunger sizes that will work with the available machine power. Metal pressure is related to the hydraulic pressure as given earlier and is given below.

$P_{md} = P_{hd} (C_D/P_d)^2$

Where:

P_{md} = Metal pressure (MP_a)

P_{hd} = Hydraulic pressure acting on the shot piston during cavity filling. (MP_a)

P_d = Plunger diameter (mm)

Once the machine has been selected the maximum hydraulic pressure and the cylinder diameter are fixed. So the metal pressure is inversely proportional to the square of the plunger diameter. If the plunger diameter is doubled the metal pressure is reduced to one fourth.

As the metal pressure is reduced the die flashing is also reduced. The other mechanism by which the large plunger diameter reduces flashing is through the associated reduction in kinetic energy of the shot system. The kinetic energy of the shot system is given by the equation.

$K.E = ½ M P_v^2$

Where K.E = kinetic energy

M = Mass of the injection systems moving parts

P_v = Plunger velocity as cavity is filling

Calculating gate and runner dimensions: The seventh and last step in designing a gating system is to establish all the dimensions of the gate and runner so that they can be built into the actual die.

Gate area of cross section: The gate area is not always placed perpendicular to the flow directions. If the gate area is positioned normal to the flow then it is known as apparent gate area. If the actual gate area is at an angle with the flow direction an apparent gate area is to be calculated. The apparent gate area is the area of cross section if the gate were positioned perpendicular the flow direction. So the actual gate area must usually be larger than the apparent gate area because of the flow angle. But the apparent gate area must be calculated first.

The first step in calculating gate area is to calculate the apparent gate area for each path. The flow rate for single path is calculated as follows.

$Q_1 = V_1/t$

Where

Q_1 = Flow rate for path one

V_1 = Volume of segment one

t = Ideal filling time for the whole part

The flow rate for each path is calculated separately. When all the flow rates have been calculated they should be added to ensure that the value matches with the total flow rate calculated earlier.

The apparent gate area for each flow path can be calculated using the equation given below.

$G_{ap1} = Q_1/G_v$

Where

G_{ap1} = Apparent gate area of section one

Q_1 = Flow rate for path one

G_v = Gate velocity

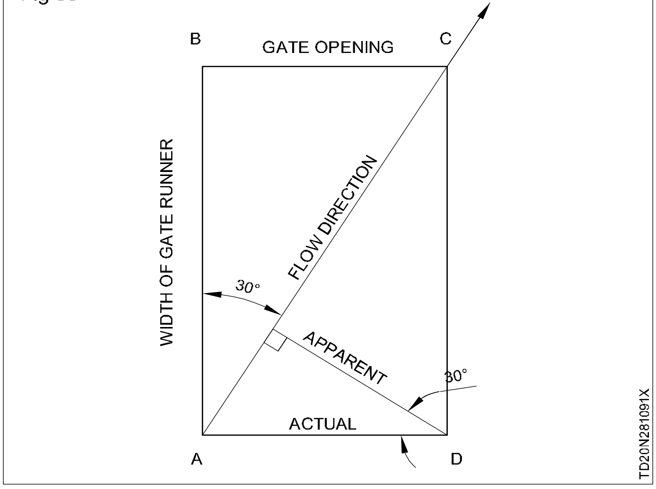
Tangential gate runners: The flow angle of tangential gate runner is same through out the length of the gate opening. The flow angle for different segments was determined in the first step. Segments requiring different flow angle should be filled by different gates. The actual gate area for each segment is equal to its apparent gate area divided by the cosine of its flow angle.

The actual gate area is equal to the apparent gate area divided by the cosine of the flow angle. (Fig 33)

$G_{ac1} = G_{ap1} / \cos\theta$

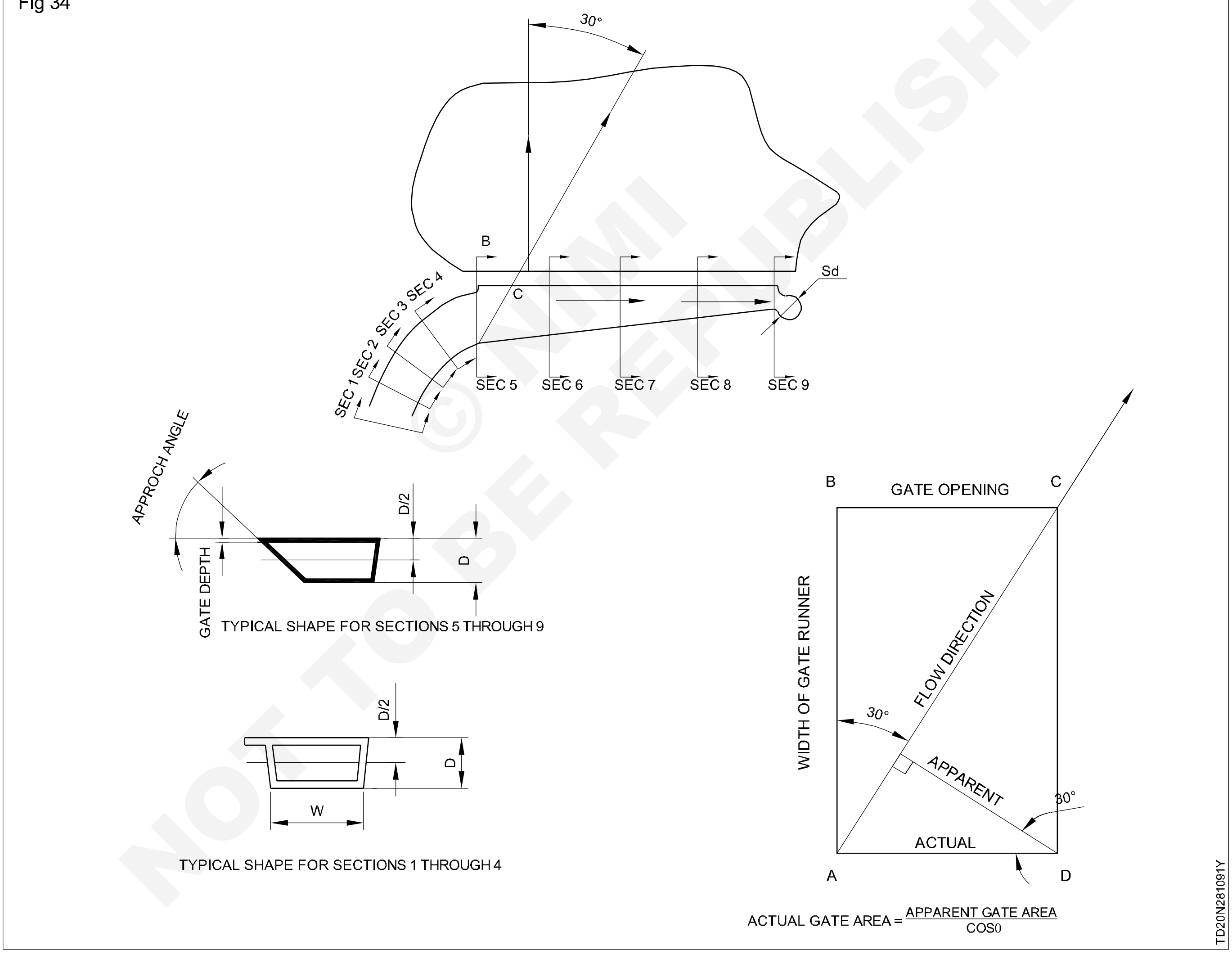
The actual gate area built into the die must be larger than the apparent gate area. The amount is determined by the flow angle. Also the total gate area must always be smaller than the nozzle exit area. But it should not be less than 40% of the nozzle exit area.

Fig 33



Area of cross section (Fig 34): Tangential gate runners are dimensioned by nine characteristics cross sections. Section one (starting of gate runner) through section four are trapezoidal. Section five through sections nine have the approach angle on the side leading to the gate. The cross sectional area of each section is a calculated function of the gate area and the flow angle. The cross sectional area decreases along the length of the gate opening. If the gate cross sections are not built to the proper sizes, the desired flow angle will not be achieved. So to get the desired filling pattern the cross sections have to be properly designed.

Fig 34



Tangential gate area dimensions: The following methods are used to calculate the area of each section.

1 The first step is to calculate the area of cross section at section five.

$$\text{Area of section 5} = \frac{\text{Actual gate area}}{\text{Tangent of flow angle}}$$

2 The second step is to calculate the area of cross section 9.

Area of section 9 = Area of section 5 - Actual gate area

The above equation ensures that the flow angle will be maintained along the entire opening.

3 The third step is to calculate the area of section 7

Area of section 7 =

$$\frac{\text{Area of section 5} + \text{Area of section 9}}{2}$$

4 The fourth step is to calculate the area of section 8

Area of section 8 =

$$\frac{\text{Area of section 7} + \text{Area of section 9}}{2}$$

5 The fifth step is to calculate the area of section 6

Area of section 6 =

$$\frac{\text{Area of section 5} + \text{Area of section 7}}{2}$$

6 The sixth step is to calculate the area of section 1

Area of section one depends up on the design strategy used. The different strategies used are:

a **Decreasing strategy:** In this strategy the area of cross section 1 is more than section5. The decreasing strategy requires the least machine power but requires large runners. The area of section can be 1.05-1.15 times the area of section5. If the area of section 1 is 1.10 times section5 then it provides a ten percentage increase in velocity when the molten metal flows from section1 to section5.

b **Constant strategy:** In this strategy the area of section 1 is same as section 5. This provides constant velocity for the molten metal from section1 to section 5.

c **Minimum shot weight strategy:** In this strategy the area of section 1 is equal to the apparent gate as calculated earlier. This strategy requires maximum machine power but minimum runner size.

7 The seventh step is to calculate the area of section 3.

Area of section 3 =

$$\frac{\text{Area of section 1} + \text{Area of section 5}}{2}$$

8 The eight step is to calculate the area of section 2

Area of section 2 =

$$\frac{\text{Area of section 1} + \text{Area of section 3}}{2}$$

9 The ninth step is to calculate the area of section 4

Area of section 4 =

$$\frac{\text{Area of section 3} + \text{Area of section 5}}{2}$$

Sectional area of the runner depends on the design strategy used. The decreasing area strategy requires the least machine power, but results in the largest runners. The constant area strategy requires a little more machine power but results in some what smaller runners. The minimum short weight strategy requires more power but it considerably reduces the shot weight.

If the decreasing area strategy is used, the area of cross section at the inlet to the gate runner is 1.10

Section depths: The depth of the gate runner is established for each nine sections as shown in the Fig 34. There are two considerations for the gate runner depths. One consideration is width to depth ratio, W/D, denoted by the letter 'r". If the width is equal to the depth (r=1), the molten metal will lose very less heat when flowing through the runner. As the ratio of width to depth increase the metal loses more heat as it flows through. The other consideration is the solidification time. Very thick runners require longer to solidify and can therefore become limiting to the production rate of the die. These two considerations are at conflict. Usually the compromise to design the runner section for a width of two or three times the depth of the runner.

Section widths: The width of the section is measured at one half the depths. In this way the width is independent of the draft and the approach angle. The width must be equal to the area of cross section divided by the depth.

Size of shock absorber: The diameter of the shock absorber should be selected to give the shock absorber a plan view area equal to the sectional area of section 5.

$$\text{Sd} = \sqrt{1.273 \times \text{area of section } 5}$$

The depth of the shock absorber should be at least 1mm, but not more than 2.5mm. Usually it is made one half the depth of section 9. If half the depth of section 9 is less than 1mm then the shock absorber should be the same depth as section 9. If one half the depth of section 9 is greater than 2.5 mm, then the shock absorber should be 2.5 mm deep.

Fan gate runner dimension (directed straight into the cavity) (Fig 35): The flow angle from a fan gate runner ranges from a maximum angle at the left side to zero flow angle at the centre and then to the maximum on the right hand side. The average flow angle is only half the maximum.

The actual gate area (G_a) = G_{app} / cosθ/2)

Fan gate runner dimension (directed at an angle into the cavity) (Fig 36): When the fan gate runner is not directed straight into the cavity edge but enters diagonally the flow angle becomes

$\theta = (\theta_1 + \theta_2)/4 + \theta_3$

Where

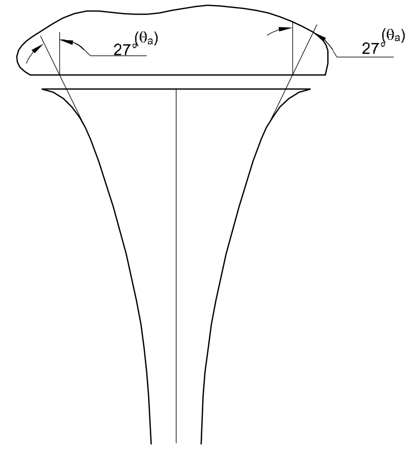
θ = Equivalent flow angle

θ_1 = Flow angle on the left

θ_2 = Flow angle on the right

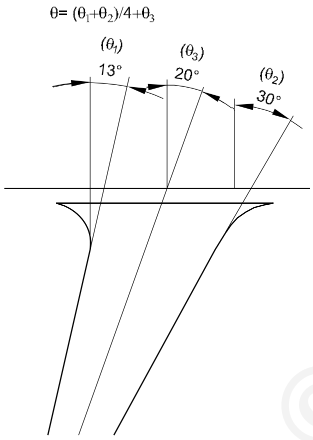
θ_3 = Flow angle at the centre

Fig 35



(FAN GATE STRAIGHT TYPE)

Fig 36

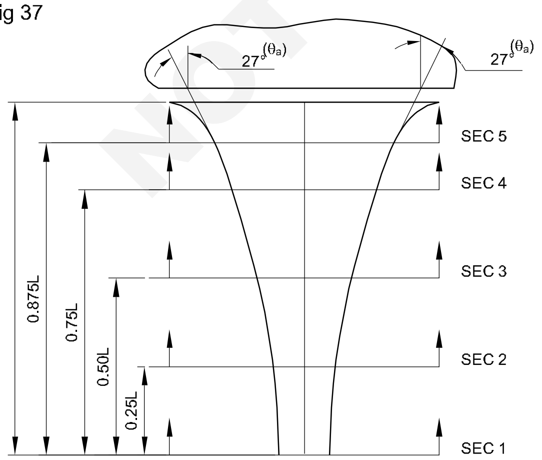


(FAN GATE DIAGONAL TYPE)

Once the equivalent flow angle ( ) has been determined the actual gate area is equal to the apparent gate area divided by the cosine of the equivalent flow angle.

Area of cross section (Fig 37): The fan gate runner is shown to be positioned straight to the cavity. The gate runner is defined by six cross sections.

Fig 37



(FAN GATE STRAIGHT TYPE)

Fan gate runner dimensions: The following methods are used to calculate the area of each section.

1 **Area of section1:** Area of section1 is calculated exactly like tangential gate runner depending up on the design strategy adopted. If the strategy is for decreasing area, the area of cross section1 is 1.1 times the actual gate area. If constant area strategy is to be used, the area of section 1 is same as the actual gate area. If the minimum shot weight is to be used, the area of cross section is equal to the apparent gate area.

2 **Area of section3:** Area of section3 is the average of section 1 and the actual gate area

3 **Area of section 2:** Area of section2 is the average of section 1 and section3

4 **Area of section4:** area of section4 is the average of section 3 and actual gate area.

4 **Area of section 5:** area of section5 is the average of section 4 and the actual area.

Depth of section: The depth of section 1 is determined by the depth to width ratio. A width of twice the depth is generally used. The calculated depth is then compared to the maximum runner depth allowed. If the calculated depth is equal or less than the maximum it is used. However if the calculated depth is greater than the maximum, the maximum depth is used.

Selecting gate depth: The gate depth (G_d) is determined by dividing the actual gate area by the length (G_L).

$$G_d = G_a / G_L$$

Where

G_d = Gate depth

G_a = Gate area

G_L = Gate length

Checking maximum and minimum gate depths: The gate depth obtained with the above calculation must be compared with the minimum gate depth obtained with the earlier equation.

$$M_{sg} \times G_d \times G_v^{1.71} = J$$

If the gate depth obtained is less than the minimum then it is not feasible. If the gate depth calculated is equal or between the maximum and minimum value then it is used as calculated.

Gate and runner volumes: The volume of the fan gate runner is obtained by multiplying the area of section3 by length 1 to 6.

Volume of fan gate runner = (area of section 3 x length 1-6)

Similarly the volume of tangential runner is obtained by adding the volume of the portion between sections 1 and 5 to the volume of the portion between sections 5 and 9.

Volume of tangential gate runner = area of sec3 x length (1-5) + area of sec7x length (5-9)

Runner volumes: The size and shape of the runner depends on the design strategy used. The volume of the each runner is simply the area times the length.

Sprue / Biscuit Volume: The smallest area in the flow path through the sprue is near the end of the spreader. The cross sectional area depends on the strategy used.

a Decreasing strategy: cross sectional area is 10% more than the actual gate area of all gates.

b Constant area Strategy: Cross sectional area is equal to the total of all section 1 areas.

c Minimum shot weight strategy: Cross sectional area is equal to the total of all the apparent gate areas.

Calculation of cross sectional area of sprue (Fig 38)

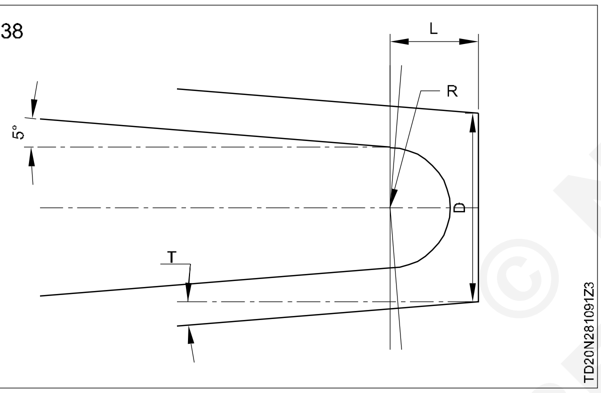
The cross sectional area of sprue can be calculated. The area is calculated by:

$$\text{Area} = \Pi \cos\theta \, (F^2 - R^2)$$

Where:

$$F = L \sin\theta \, \frac{D}{2} + \cos\theta$$

Fig 38



Runner Sprues: The runners are normally continued up to the spreader. The spreader is made to completely fill the busing, except 0.5mm to 1mm clearance, and then the runners are cut into it. The runner sprue eliminates the problem of changing cross sectional area of the molten metal.

Biscuit volume: The volume of the biscuit for a cold chamber die can be calculated directly. The plunger diameter was already calculated.

$$\text{Therefore the biscuit volume} = \frac{\Pi d^2}{4} \times B_1$$

d = Plungerdia meter

B_1 = Biscuit length

The length of the biscuit depends on the die caster. If he uses intensified plunger pressure to feed shrinkage during the solidification of the die casting the biscuit length will be probably kept 25mm long. But such long biscuits will usually increase the solidification time. Shorter biscuits of about 6.5mm will solidify quickly.

Force calculations

Die locking: Early in the analysis of a die casting, die locking requirements should be calculated. These calculations will establish the machine size to be used.

$$F = P_m AI \text{ [Where } (P_m) = P_h \left(\frac{D}{d}\right)^2 \text{]}$$

F = Force required to hold the die closed (tons)

P_h = Hydraulic pressure in injection cylinder

P_m = pressure acting on the molten metal

A = Projected area of all cavities, runners, overflows and sprue and biscuit.

D = Diameter of injection cylinder

d = Diameter of plunger

I = Dimensionless impact/freeze factor

The impact/freeze factor 'I' depends on the injection velocities required. Generally the value may range from approximately 3 for shorter fill time to 1.5 for longer fill times. For very thick casting the value may range from 0.6 to 0.8.

Molten metal pressure will usually be about 13.8MPa for Zinc and 34MPa for Aluminum and Magnesium castings.

If the projected area is not centered between the tie bars the load required for each tie bar must be calculated by the sum of the moments. The required machine size is determined by multiplying the load of the tie bar with the greatest load by the number of tie bars.

The first step is to divide the total projected area in to simple geometrical forms such as rectangles to find the centre of gravity of each. Then both the horizontal and vertical distance from the centre of gravity of each geometric shape from the tie bar are computed. Next the area of each geometric form is calculated and multiplied by the injection pressure.

The next step is to use the sum of the moment's method to calculate how the force of each section of projected area is applied to each of the four tie bars. The calculation is simplified by dividing in to several parts. The first part is to calculate the load on the left side of the machine. (I.e. tie bars A and C as shown in Fig 39).

Using the formula:

$$\text{Load} \quad 1 \text{ on } A + C = F1 \times \frac{X1}{D}$$

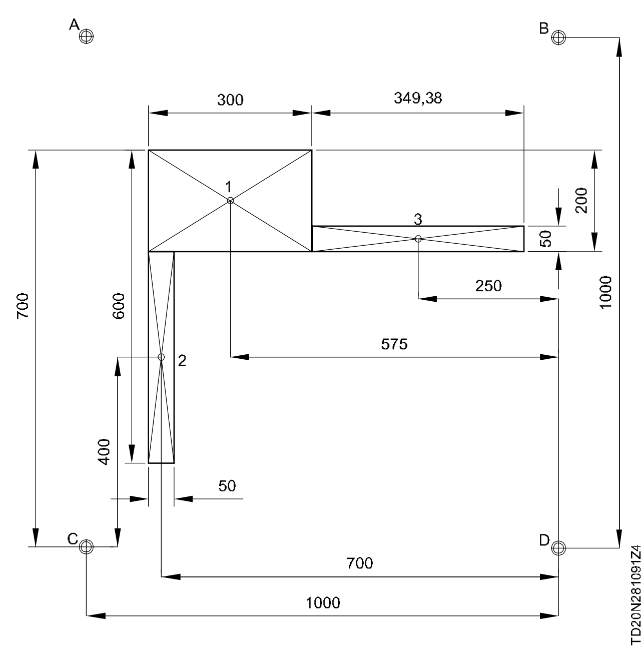
Where:

F1 = Force generate by projected area of section 1 (F1=180 Tones ,i.e. 0.3T/cm^2)

X1 = Distance from centre of gravity of section1 to the centre line of B and D tie bars. (d1=575mm)

D1 = Distance between tie bars, D =1000mm

Fig 39



Therefore load 1 on A + C = $180 \times \frac{575}{1000}$

= 103.5T

Then by using the same general formula, the load acting on the right side of the machine (i.e. tie bars B and D) due to the same section (section 1) is calculated.

The load 1 acting on B + D = $180 \times \frac{1000 - 575}{1000}$

= 76.5T

A check can be made for the calculations by adding the two results (i.e. 103.5+76.5 =180T). The total should be equal to the force generated by section 1 of the projected area. Next it is necessary to determine how the 103.5 is divided between tie bars A and C and tie bars B and D.

The load 1 acting on A = Load on (A + C) x $\frac{Y1}{D}$

Where Y1 = Vertical distance between centre of gravity of section1 to the tie bar C

Therefore load 1 on A = $103.5 \times \frac{700}{1000}$ = 72.45T

Similarly load 1 on c = $103.5 \times \frac{1000 - 700}{1000}$ = 31.05T

Similarly the load 76.5 T on B+D is divided:

load 1 on B = $76.5 \times \frac{700}{1000}$ = 53.55T

load 1 on D = $76.5 \times \frac{1000 - 700}{1000}$ = 22.95T

The procedure is repeated for all section of the projected area.

Therefore load 2 on A + C = $60 \times \frac{700}{1000}$ = 42T

Therefore load 2 on B + D = $60 \times \frac{1000 - 700}{1000}$ = 18T

Therefore load 2 on A = $42 \times \frac{400}{1000}$ = 16. 8T

Therefore load 2 on C = $42 \times \frac{600}{1000}$ = 25. 2T

Therefore load 2 on B = $18 \times \frac{400}{1000}$ = 7.2T

Therefore load 2 on D = $18 \times \frac{600}{1000}$ = 10. 80T

Therefore load 3 on A + C = $52.5 \times \frac{250}{1000}$ = 13.125T

Therefore load 3 on B + D = $52.5 \times \frac{1000 - 250}{1000}$ = 39.375T

Therefore load 3 on A = $13.125 \times \frac{625}{1000}$ = 8.2T

Therefore load 3 on C = $13.125 \times \frac{1000 - 625}{1000}$ = 4.92T

Therefore load 3 on B = $39.375 \times \frac{625}{1000}$ = 24.6T

Therefore load 3 on D = $39.375 \times \frac{1000 - 625}{1000}$ = 14.76T

Lastly, the loads are added for each tie bar. Then find out which tie bar has the maximum load acting on it. This load is multiplied by four to get the tonnage to be set on the machine. So in this case the maximum load is acting on tie bar A which is equal to 97.45 (72.45+16.8+8.2). All tie bars must be locked to 103.3875 tones before the shot is made. There for the minimum machine tonnage required is equal to 389.8tones. The die may be loaded on a 400T machine.

Tie bar loading for off-centre conditions (The injection pressure is assumed to be 0.3T/cm²)						
Geometric Form			**Tie bar load in Tones**			
Section	**Area mm2**	**Force Tones**	**A**	**B**	**C**	**D**
1	60000	180	72.45	53.55	31.05	22.95
2	30000	60	16.8	7.2	25.2	10.8
3	18000	52.5	8.2	24.6	4.92	14.76
Total		292.5	97.45	85.35	61.17	48.51

Die casting defects cause and remedies

Surface defects

Cold flow, cold lap, chill, non-fill, swirls, etc.

Cause: Leading edge of metal flow is too cold, laps together.

Remedies

1 Increase dies temperature, especially at problem location. Check the following causes for cold dies.
  - Slow cycle time
  - Excessive coolant flow for conditions
  - Excessive spray causing cold die
  - Add overflow for more heat

2 Reduce fill time (Caution: Each action listed affects something else besides fill time. A PQ^2 calculation is required).
  - Increase plunger speed
  - Increase plunger size
  - Increase gate area
  - Increase hydraulic pressure

3 Change flow pattern (gating).
  - Direct flow at the problem area by moving the gate
  - Change gate design to direct flow in a different direction
  - Change gate velocity (try to increase first)
  - Add overflows to capture cold metal

4 Check for low metal temperature.
  - Look for delays that cool metal
  - Look for furnace temperature variations (low temp swings)
  - Increase temperature carefully, other problems may appear)

5 Low pressure at end of shot check for
  - Dragging tips - flash around tips
  - Poor sleeve condition
  - Too little or inconsistent sleeve lubrication
  - White (solder) buildup in shot sleeve

6 Accumulator pre-charge pressure too low

7 Thin biscuit

8 Heavy flashing

9 Bubbling and turbulence in hot chamber - bad rings or poor gooseneck

10 Check also:
  - Alloy - for aluminum alloys
  - Silicon at high end of range if possible
  - Metal cleanliness
  - Alloy - for zinc alloys
  - Aluminum content at high end
  - Other constituents in range
  - Vents open, and sized correctly
  - Vacuum working
  - Thin wall section present

Check for minimum wall thickness for the fill time and temperatures being used.

Design errors, designers not aware of casting problems.

Tool making errors (dimensions not as expected)

Uneven wall section (poor part design)

Lamination: layers of metal inside or outside of casting.

Cause: Most common is poor metal flow control, though there are other causes.

Remedies

1 Check injection parameters
  - Fast shot start switch should start fast shot so metal is accelerated before metal gets to gate (start fast shot early).
  - Quick fill time - very important, check with calculations.
  - Proper gate velocity

2 Gating: good flow patterns - no long flow distance and mixing far from gate.

3 Good die temperature: should be consistent over the trouble area best if on the high side.

4 Intensifier action proper and consistent.

5 Die doesn't flex (die may flex from intensifier pressure) - check for adequate support.

6 Check that the lamination is not from flash left on die (often die must be cleaned every shot).

7 Examine lamination to see if it is oxide layer.

Gas porosity

Cause: Gas trapped in the metal flow during die fill.

Remedies

1 Gas comes from trapped air, steam or die lubricant - check for sources of trapped air.

- Consistent pour rates
- Delay after pour - set right to minimize splashing in shot sleeve
- Acceleration to slow shot speed correct
- Use critical slow shot speed
- Accelerate to fast shot speed as late as possible (this depends on situation).

2 Check runner for smooth flow path

- No sharp corners
- No blind ends, pockets
- Ever decreasing areas properly used in runner path

3 Check vents

- Right size (big enough)
- Vents kept open (not full of flash)
- Located at the last point to fill, use short shots or computer predictions to locate last point to fill.
- Vents go to the edge of the die.

4 Vacuum working

- Vacuum channels big enough
- Vacuum channels located at last point to fill
- Filters cleaned and open
- Vacuum valves working
- Vacuum level adequate

5 Check for gas from lubricant

- Look for excessive plunger lubricant, (discolored castings) - be sure to run the minimum possible amount.
- Try to avoid putting lubricant in front of the tip.
- Look for consistent application procedures
- Look for excessive die lubricant or anti solder paste

6 Check for steam (water on die)

- Check that the die is dry when it closes.
- Use lots of air blow off, both with manual and automatic systems.
- Put drain holes in die where die lube (water) could accumulate
- Check for water leaks after die is locked (open die without making a shot, look for moisture).
- Look for leaks from sprayer, hydraulic cylinders, etc.

Blisters

Cause: Gas trapped just under the surface in the metal during die fill.

Remedies

1 Blisters are another version of gas porosity therefore the same correction used for gas porosity will apply for blisters, i.e.

- Reduce trapped air (see gas porosity corrections)
- Reduce spray and plunger lubricant
- Eliminate water on the die
- Correct venting and vacuum problems

2 The most permanent solution to the blister problem is to correct the gas porosity problem. However, as a short-term solution, blisters can be hidden by the following actions.

- Cool the die in the immediate are where the blisters occurs by
- Cooling the blister area with die spray
- Cool the blister area by adjusting water lines
- Cool the whole die by slowing the cycle time
- Adding fountains or baffles to the blister area

3 Cool the casting immediately after ejection by quenching in water (this will keep the skin strong and resist blister formation).

4 Reduce metal temperature (but watch for other problems)

- Keep process consistent
- If blister is associated with metal swirls and captured gas from metal flow, try to correct gating or venting problem, or add vacuum.

Flow porosity

Cause: Metal flow is too slow, too cold or has a poor flow pattern; and leaves space (porosity) between solidified metal flows.

Remedies

1 This is a metal flow problem, so the same correction apply as those previously listed for surface defects

2 The spaces between metal flows may appear on the surface (hole) or inside (porosity).

- The biggest factor by far is the fill time - calculate and measure to be sure it is fast enough - if in doubt, reduce fill time as much as possible

3 Stabilize furnace operation (reduce variation maximum of to +/- 2 C), use the correct metal temperature.

- Metal temperature at the gate is important, in aluminum, watch for heat loss in ladle and shot sleeve - add more superheat.

4 Stabilize die temperature, and run higher die temperature (>205°C).

5 Review and correct metal pressure.

- Review the static metal pressure; it should be above 13.8MPa for zinc and 20.6MPa for aluminum and magnesium.
- Check intensifier operation.
- Consistent (most important)
- Quick enough (measure and evaluate rise time, best time will vary with casting shape)
- Pressure setting high enough (final pressure>41.40MPa is alright, >62MPa is best)

Shrinkage porosity

Cause: Cast material takes up less space when solid than when liquid and this space will appear where there is a hot spot in the casting.

Remedies

1 Increase pressure on the semi solid metal (at the porosity location) during solidification.

2 Check for pressure problems.

- Static metal pressure problems

  >20.70MPa min. for Al and Mg

  >13.80MPa min. for Zn (>10.3MPa may be alright)

- Intensifier pressure correct

  >55.2MPa final metal pressure is very desirable (>41.40MPa may be alright)

- Check intensifier setting

  Use monitor system trace to verify pressures

  Intensifier accumulator charged correctly, intensifier cylinder not bottoming, etc.

  Check rise time on monitor (set a desired standard); be sure intensifier is coming in fast enough check switch settings.

- Shot accumulator pre-charge pressure correct
- Plunger problems that reduce pressure

  Poor tip condition (even if there is no blow by)

  Poor sleeve conditions

  Soldering at end of sleeve

  Check for sleeve contraction from heat (sleeve squeezed by die)

  Plunger cooling not working

  Inadequate lubrication

  Poor sleeve cooling

  Hot chamber - check plunger rings

  Hot chamber - if plunger bottoms and rings are good, change gooseneck

Heat sinks: (shrinkage porosity).

Cause: Shrinkage cracks are just under surfaces.

Remedies

1 See shrinkage porosity, those techniques will work here.

2 Cool hot spot directly where the heat sink occurs - use the following

- Fountain (first choice)
- Die spray

  Thin biscuits, poor plunger conditions (hot or cold chamber).

  Heat opposite side of casting:

- Shut off water
- Stop spray

  Check for uneven temperatures between die halves, especially in the area of the heat sink.

  Use pressure to feed more metal during solidification t area where sink occurs,

- Adjuster intensifier
- Move gate
- Change static pressure
- Look for dragging tip (low pressure)
- Check accumulator pre-charge (low pressure)

Leakers: (Shrinkage porosity).

Cause: Loose dendritic structure inside casting is exposed by an opening in the casting skin that provides leak path (another version of shrinkage porosity).

Remedies

1 Look for sharp corners at the spot where the leak occurs; add as much radius as possible.

2 Cool with spray at the spot where the leak occurs- keep cooling even if the spray makes no difference visually.

3 Thin biscuit, or biscuit size varies too much (can be major cause in many shops.)

4 Try to keep skin intact in leaker area

- Stop drags from solder or other sources
- Reduce machining if possible, keep skin intact whenever possible.

5 Try to keep skin intact in leaker area,

- Check for problems in plunger - sticking and dragging
- Poor pressure control - check static and intensified pressure
- Check intensifier performance
- Add squeeze pins in area of leakers
- Move gate closer so more pressure can be applied during solidification by intensifier.

6 Change temperature balance, cool area where leakers occur, heat areas surrounding leaker location.

Cracks and tears

Cracks, tears, hot cracks

Cause: Many causes from shrinkage cracks on surfaces, casting being stretched in the die, mechanical stress at die opening, ejection, or from trimming etc

Remedies

Determine the most probable cause first.

1 Shrinkage cracks (surface porosity).

- Discolored crack
- Evidence of dendritic structure

2 If shrinkage cracks are the problems.

- Check for good radii at crack location
- Cool the hot spot
- Heat up the adjacent cool spots
- Add pressure in this area during solidification (see shrinkage porosity, the same corrections apply here).

3 For castings that crack in the die from being stretched during cooling (stress cracks during cooling and after solidifying) as shown by:

- Cracks at a weak point in the shape
- Cracks at stress concentration points
- Crack surface in not oxidized.

4 If this stress cracking during cooling is the problem, then:

- Reduce stress riser (add radius as much as possible)
- Shorten hold time
- Look for thin wall castings that cool before runner; in this case cool runner and eject sooner
- Add wall thickness if possible
- If biscuit determines hold time, then add cooling to biscuit
- Change runner shape if necessary to eject sooner

5 Identifying crack problems from mechanical forces or die shift during die opening.

- Identify by cracks at bottom of deep walls or cores - or cores or walls that stick in the cover half.
- Usually have some evidence of drags

6 Corrections for castings for castings that crack from die shift during die opening

- Watch as dies separate, leak for evidence of die shift (ejector die drops forward as dies separate).
- Watch as dies close; if die guide pins carry the load, it is likely there is stress on the casting during opening
- Add die carrier under ejector die
- Reset die to re-align the ejector half better
- Adjust shoes under moveable platen as needed
- Check tie bar stress for even locking
- Check condition of linkage, repair as necessary
- Check condition of tie bar bushing, repair as necessary

7 Identifying crack problems that are caused by ejection

- Drags usually present
- Sticking problems present
- A visual check shows ejection not straight and even
- Slow down ejection process to clearly see what is happening.

8 Corrections for cracks that occur during ejection

- Check that ejection movement is straight, guided, and does not wobble during ejection.
- Check for drags, lack of draft, or undercuts.
- Eject slower and smoother
- Check slides for proper action, (slides not worn out and wobbly)

9 Check metal constituents

- In aluminum, check proportion of Fe, Cu, Si; also check for presence of silicon modifiers.

Inclusions

Inclusions, hard spots

Cause: In aluminum, inclusions are mostly oxides, usually from poor melt-cleaning and furnace cleaning practices: Also can be furnace refractory. In zinc, the iron aluminum inter-metallics can lead to polishing and machining problems.

Remedies

1 Aluminum - the oxides were probably made in the melting furnace; check cleaning procedures.

- Examine melt line and corners for build up

- Scrape bottom for excessive corundum material on the bottom of furnace

2 Check wall cleaning procedures;

- Check for Proper tools?
- Operators trained?
- Is cleaning schedule discipline maintained?

3 Check for a delay after cleaning and before delivering metal - try to get at least half hour, more is better - very important!

4 Check fluxing procedures - once/shift?

5 Check de-gassing procedures - as often as possible

6 Check filter system

- Filters replaces as needed?
- Check Filters leaking around the edges?

7 Review temperature at central melting, is it too high? (Probably best between 730 -760 c) Use one setting; do not allow variations in settings.

Solder

Solder

Cause: Aluminum or magnesium and die steel combine and cast metal sticks to die surface; in zinc, the zinc forms layer on top of the steel.

Corrections

1 Aluminum

- Check temperature in solder area, reduce if at all possible - this is the best solution.
- Add fountain (bubbler) in solder area (even a 3mm diameter fountain will do a good job)
- In solder area use high heat transfer material (TZM, ANVILLOY, and MITECH)
- Be sure water lines are functioning (clean of deposits)
- Increase spray on the solder area
- Reduce speed, increase cycle time.
- Reduce fill time
- Lower the length of time the metal impacts on the solder area
- Check the metal velocity - is it too high?
- Velocity above atomization, but not much (about 25.4m/s to 35.5m/s)
- Check PQ^2 for proper process settings; find best plunger size, speed, pressure, etc.
- Check actual plunger size and speed - are desired conditions really being met?
- Check draft angle at solder point
- Check for undercuts or rough surface on die at solder point.

Carbon build up

Cause: Build up of deposits, usually from lubricant, or water mixed with lubricant.

Corrections

1 Check lubricant application;

- Spray volume is at minimum
- Increase die temperature with cycle time decreases, water flow adjustment, or spray adjustments.
- Even out the die temperature, get rid of cold or hot areas.
- Spray mixture and amount applied should not be varied arbitrarily, especially from shift to shift.
- Use the proper lubricant to match the die temperature, especially when using a cold die (measure die temperature, verify with the supplier that the lubricant will work at the die temperature)
- Do not spray into blind fins and cores and other cold areas
- Extra spray should be carefully removed with air blow-off

2 Do not use hard water for mixing with lubricant.

Die erosion

Die erosion, cavitations, and burn out. Die has worn spots causing raised spots on the casting; can be small deep cavities (cavitation), or larger erosion areas at the gate.

Cause: High metal velocity, bubbles in incoming metal, high oxide or silicon content in metal.

Remedies

1 Check gate velocities

- Aluminum metal velocities should be about 25.4m/s to 40.6m/s
- For zinc, gate velocity should be about 30.5m/s to 50.8m/s
- For magnesium, gate velocity should be about 30.5m/s to 76.2m/s. (Less damages occur from higher velocities in magnesium.)

2 Check metal temperature, should not be high.

3 Check dies temperature in the gate area, reduce with spray if possible.

4 Check metal cleanliness, oxide cleaning procedures should be in place (see hard spots).

5 Check filling times - long fill times accelerate gate erosion.

6 Check alloy - hyper eutectic (high silicon) alloys require smaller process window (lower gate velocities).

Outgassing

Defective surface finish occurring when bubbles appear during a painting or finishing operations.

Cause: Leak path develops through casting skin when casting is heated during finishing, allowing the heated and expanding trapped gas to escape.

Remedies

1 If problem is in overflow gates, then minimize or combine number or overflow gates used, reduce size of overflow.

2 Keep overflows away from edge of castings to minimize heat build up next to casting.

3 Make main gate thinner while still maintaining appropriate gate area for casting quality needs.

4 Reduce metal temperature, but stay above 790 for zinc, and above about 1200 F for aluminum.

5 Reduce die temperature at gate.

6 Reduce trapped gas (see gas porosity corrections)

7 Make sure there is pressure at the end of the stroke

- No thin biscuits or leaking plunger rings
- Change or correct problems before plunger starts to stick or drag
- Check for proper metal pressure, both static pressure and intensified (right size plunger).
- Accumulator pre-charge correct

Edge porosity - porosity at the gates

Cause: Either shrink or gas porosity

Corrections

1 Shrink porosity correction

- The path for the gas is formed by the loose dendritic structure near the gate; this can be reduced by cooling this area more (can overcool for thin gates)
- Use long, flat gate ramps to avoid the hot steel next to the gate
- Make the gate thinner and wider to spread the heat more (don't exceed the proper gate velocity).
- Move the gate to the other half of the die if it would be cooler there
- Reduce fill time as much as possible - this reduces heat left at the gate by the metal stream.

2 Gas porosity

- Gas porosity will contribute to outgassing, but not nearly as much as shrink porosity.
- Use the techniques described in gas porosity corrections to reduce this problem.

Bending warping

Bending warping out of flatness

Cause: Many operational and design issues.

Remedies

1 Design issues

- Too much tolerance allowance for tool construction (save most of the available tolerance for process variations)
- Incorrect shrinkage (one value for all dimensions may not be accurate enough)
- Incorrect estimate of process capabilities

2 Operational corrections

- The most important factor is a consistent ejection temperature; the casting and the die must be at the same temperature each time a casting is ejected
- Control holds time with a thermocouple instead of a timer
- Maintain a very consistent process to keep temperature consistent

  Consistent die spray

  Consistent cycle time

  Consistent cooling water flow rates

Flash: Excessive cast material often accompanied by dimensional deviation.

Cause: High metal temperature and either poor die fit or poor machine locking when high pressure is applied.

Remedies

1 Die fit

- Check die fit at operating temperature - use bluing or other means to check fit with die at operating temperature.
- Review castings for evidence of die deflection (look for differences with high and low pressures); review die design for proper robustness

2 Machine locking conditions

- Strain tie bars for equal load
- Check condition of machine
- Linkage not worn
- Tie bar bushings and movable platen shoes in proper condition
- Platens flat not bent
- Die set up correct
- Tie bar nuts not loose

3 Die opening force centered on machine

- Center of load calculated, and load on each tie bar calculated.

4 Metal pressure considerations

- Static Metal pressure in the 20 to 40 MPa range
- Intensified metal pressure in the 48 to 82 MPa range
- Impact spike high because of accumulator in back or other considerations
- Intensifier control not consistent comes in fast one time, slow the next.

Stained castings

Discolored casting

Cause: Foreign material in the metal, mostly dies lubricant, but can be other material.

Remedies

1 Review lubricant practices:

- Check amount of plunger lubricant
- Consistency of application (this is a major factor in many plants)
- Check amount of die lubricant used
- If possible look for a different lubricant material.

2 Look for other material in liquid metal - possible from scrap.

Waves and lakes

Cause: Usually seen in decorative zinc castings, caused by early metal flows that solidify quick leaving a separate skin that is not remelted; the surface of this area is more fine grained than the rest and has a slightly different appearance.

Correction

1 Correct metal flow:

- Much quicker fill time
- Change flow pattern to minimize splashing and jetting in the area

Drags

Drags

Cause: Deformation of the casting by undercuts encountered during ejection. Undercuts may be caused buildup on the die or by die erosion of solder.

Corrections

1 See corrections for build-up, solder and erosion

2 Make sure die surface is smooth, machining marks have been completely polished out.

3 Check draft angles.

4 Reduce the temperature of the steel that has the drag- this can be done with spray or with high heat transfer die materials.

5 Check metal temperature.

Deformation from ejector pins

Cause: Caused when the casting is still soft and sticks in the die; consequently the ejector pins bend the casting trying to eject.

Remedies

Check the following

1 Undercuts, drags.

2 Casting stays in the die too long or too short (hold or dwell time not correct for casting)

- Short dwell time means the casting may be too soft at ejector pin location and may deform
- Long dwell time for certain shapes may mean that the part has contracted on to the die steel, and thus requires extra ejector force to remove.

3 Machine ejection system is "jerky"' with high impact on the casting

4 Poor die design, check for:

- Too few ejector pins
- Pins in the wrong locations (must be balanced force around cores and other ejection resistant features)
- Use ribs under ejector pins to spread the load
- Ejector pins too small
- Ejection guidance system inadequate or worn out (ejector plate wobbles during ejection)
- Ejection system load not balanced, or if unbalanced ejection load is required, the ejector plate guidance system design was not adequate for the off center loading.

Excessive flux

Cause: Too much flux causes an increase in porosity and surfaces corrosion: this is determined by putting casting in clean water overnight or examining a fracture through the porosity area for white spots.

Remedies

Reduce flux usage

1 Review procedures with experts; determine the correct amount to use, and the correct application procedures.

2 Write down procedures.

3 Train operators carefully about how much flux to use and how to apply it.

Dies and moulds economics

Objectives: At the end of this lesson you shall be able to
- **list the various Technical Terms used in Dies & Moulds Economics**
- **differentiate between estimating & costing**
- **enumetrate material costing**
- **state what is costing & expenses**
- **calculate basic machine hour rate**
- **list the terms related business transaction**
- **calculate cost of components**
- **brief the activity based costing**
- **prepare the estimation of tools.**

Various Technical terms used in Dies & Moulds Economics

1 **Direct labour:** The workers who are actually processing the materials form the direct or productive labour.

2 **Indirect labour:** The workers who are not actually processing the materials form the indirect or non-productive labour.

3 **Direct labour cost:** It is the wages paid to the direct labour.

4 **Indirect labour cost:** it is the wages paid to the indirect labour.

5 **Direct material cost:** It is the cost of those materials which form the final shape of the product.

6 **Indirect material cost:** It is the cost of those materials which are consumed in order to convert the direct material to their final shape.

7 **Overhead charges:** These are business expenses not chargeable to particular part of the work. These are also known as indirect cost, on cost or burden. The term includes all expenses except direct labour, direct material and direct expenses incurred in production and distribution of the product to the consumers.

8 **Factory over heads:** These are known as expenses chargeable as indirect labour, indirect material etc.

9 **Administrative over heads:** These include salaries of office staff, high rank officials, telephone charges etc.

10 **Prime cost:** It is the sum of the direct labour, direct material and other direct expenses if any. It is also known as direct cost.

11 **Factory cost:** It is the sum of prime cost and factory over heads.

12 **Manufacturing cost:** It is the sum of the factory cost and administrative expenses.

13 **Total cost:** It is the sum of the manufacturing cost, selling and distributing expenses.

14 **Selling price:** It is the sum of the total cost and profit of the company.

15 **Man-hour rate:** It is the method of distributing factory over head over each job by considering the total working hours of all workers. It is calculated by dividing total factory overheads by number of total working hours.

16 **Machine hour rate:** It is the method distributing factory over heads over each job by considering the total machining hours. It is calculated by dividing total factory overheads by total hours for which the machine runs.

17 **Allowances:** This is the time allowed to the workers during their working hours. These may be:

a Set up time.

b Operation time.

c Personal allowance.

d Fatigue allowance.

e Tool changing allowance.

f Tool grinding time.

g Checking time.

18 **Fixed cost:** A cost which remains constant irrespective of the volume of out put. These costs mainly depend on the time spend and do not vary directly with the rate of out put.

19 **Variable cost:** A cost which varies directly with the quantity produced.

20 **Depreciation:** The reduction of value of a fixed asset due to use or laps of time.

21 **Price:** The cost to a buyer of any product expresses in terms of money.

22 **Break Even Point:** The point at which neither profit nor loss is made

23 **Quotation:** This is a document containing the details of rates and other terms and conditions at which the jobs are performed. It is a formal statement of the estimated cost of a job.

Aims of estimating

1 The main aim of estimating is to help the factory owner to decide about the manufacturing and selling policies.

2 It helps in preparing quotation.

3 It helps in deciding the expenses in advance.

4 It gives a reference for setting the price.

5 It helps in taking certain favorable decisions.

Aims of costing

1 To determine the cost of each operation.

2 It helps in controlling the cost of production.

3 It suggests changes in operation or design when the cost is higher.

4 It provides information for adding new machines.

5 It helps in deciding workers wages.

6 It helps in determining the price.

7 It helps to locate the source of raise or reduction in profits.

8 It helps to compare the actual cost with the estimated cost.

Difference between estimating and costing

Estimating

1 It gives an expected cost.

2 It is carried out before production.

3 It needs a qualified technical person

Costing

1 It gives the actual cost.

2 It usually starts with the order for production and ends after the commitments for sale is over.

3 It can be done by an accountant.

Estimating Procedure

1 Engineering department decides the requirements and specifications of the product.

2 Engineering department makes out the drawing, decides the type of material and its quantity and other requirements.

3 The planning department lays down the methods and sequence of operations, decides machines to be used and schedule the operations. Estimating department is generally attached to the planning department.

4 To decide accuracy and finish required.

5 To prepare a list of the components of the product.

6 To decide which component can be made in-house and which should be procured from outside.

7 Determine the material cost.

8 Determine the machining cost.

9 Determine the labour cost.

10 Determine the prime cost.

11 Determine the factory overheads.

12 Determine the administrative overheads.

13 Determine the packing and delivery charges.

14 Calculate the total cost.

15 Decide on the profit.

16 Fix the sale price.

17 Decide on the delivery time.

Costing procedure

1 Collect actual material cost.

2 Collect actual labour cost.

3 Collect factory overheads.

4 Collect administrative over heads.

5 Collect selling overheads.

6 Calculate the total cost.

The total cost is made up of three main elements. They are

1 Material cost

2 Labour cost.

3 Expenses (over heads).

The various components of cost are

1 **Prime cost** = Direct material cost+ Direct labour cost+ Direct expenses.

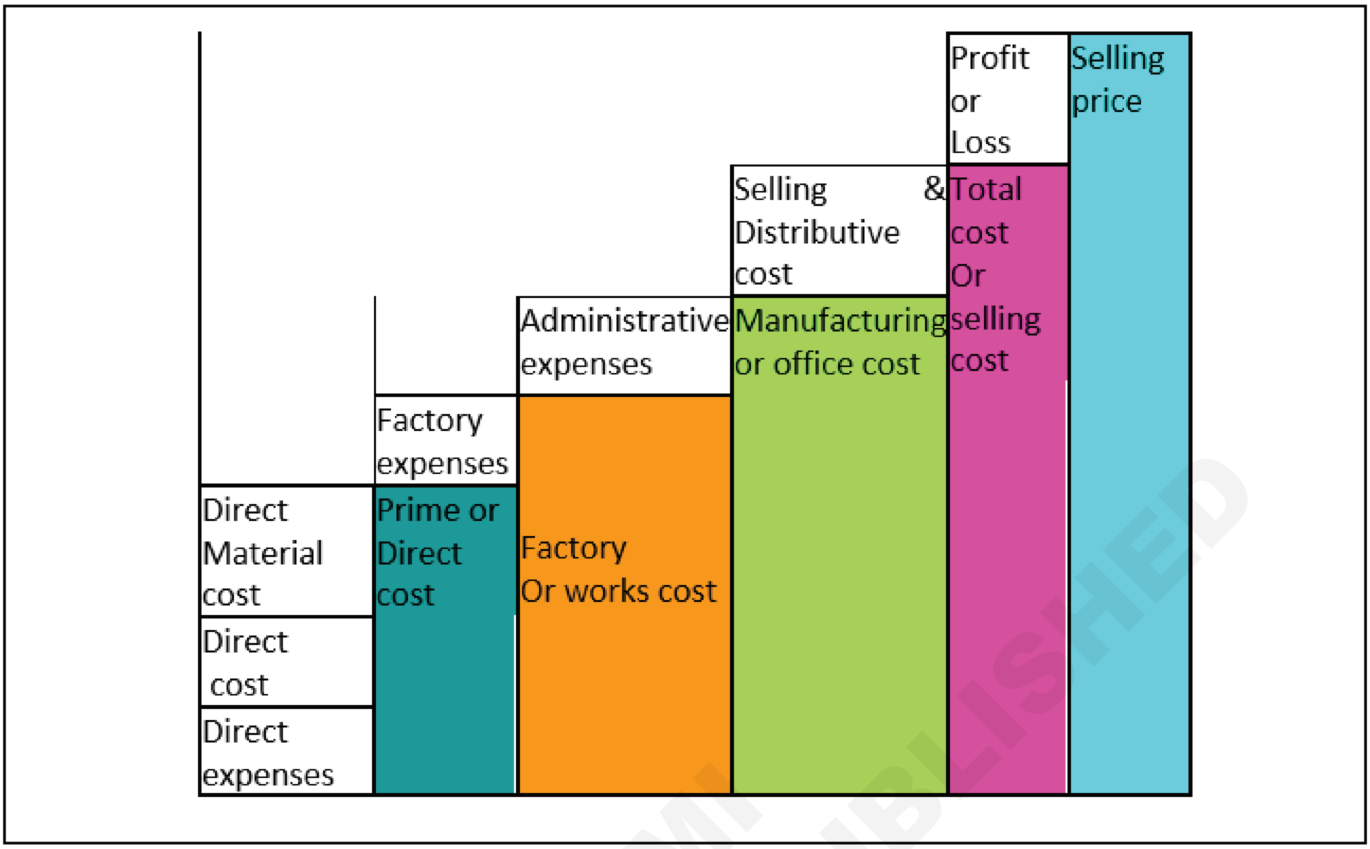
2 **Factory cost** = Prime cost + Factory Expenses.

3 **Manufacturing cost**= Factory cost+ Administrative expenses.

4 **Total cost** = Manufacturing cost+ Selling and Distribution cost.

5 **Selling price** = Total cost + Profit

Element of cost block diagram



General estimation form

Estimate Form

Description:---------------- Quantity:-------------------- Drawing No:------------ Date :--------------- Enquiry No:---------- Customer :-----------		
Item	Total cost	Cost of each item
1 Material (@--------------) Quantity (-------------)		
2 Operation Labour Overhead (a) (b) (c)		
Total : (Factory cost)		

3 Office and administrative Expenses Total: (Manufacturing cost)		
4 Selling expenses (a) Packing and carriage (b) Advertisement and publicity (c) Other allied expenses Total : (Total cost)		
5 Profit Total: (selling price)		
Delivery date:-------------------- Estimated by:-----------------------		

Worked out examples 01

Example1: The expenditure for manufacturing a turret lathe is as given below:

Material consumed	=	Rs.55, 000.00
Indirect wages	=	Rs.8, 000.00
Consultant charges	=	Rs.3, 000.00
Advertisement	=	Rs.10, 000.00
Set profit	=	Rs.12, 500.00
Depreciation on sales department	=	Rs.1, 100.00
Printing & stationary	=	Rs.250.00
Depreciation of plant	=	Rs.4, 500.00
Direct wages	=	Rs.65, 000.00
Factory rent	=	Rs.6, 000.00
Telephone & postage charges	=	Rs.150.00
Plant electricity charges	=	Rs.500.00
Office salary	=	Rs.2, 100.00
Office rent	=	Rs.500.00
Showroom rent	=	Rs.1, 500.00
Sales man concession	=	Rs.2, 650.00
Sales deportment expenses	=	Rs.1, 500.00

Find out

a Direct cost

b Factory cost

c Office cost

d Cost of sales

e Selling price

Solution

a Direct Cost

Direct cost = direct material + direct + direct expenses

= 55,000 + 65,000

= Rs.1, 20,000.

b Factory cost

Factory cost = Direct cost + factory overheads (factory expenses)

= 1,20,000 + 8,000 + 4,000 + 6,000 + 50

= Rs.1, 39,000.

c Office cost

Office cost = Factory cost + Admin Exp.

= 1,39,000 + 3,000 + 250 + 150 + 2,100 + 500

= Rs.1, 45,000.

d Cost of sales (selling cost)

Selling cost = Office cost + Sales overheads (selling & distribution cost)

= 1,45,000 + 10,000 + 1,100 + 1,500 + 2,560 + 1,500

= Rs.1, 65,750.

e Selling price

Selling price = total cost + profit

= 1, 65,750+ 12,500

= Rs.1,74,250.

Example 02

A firm in batches of 100 produces a certain piece of work. The direct material cost for those 100 pieces is Rs.200 and the direct cost is Rs.260. factory on cost is 30% of the total material and cost. Overhead charges are 25% of the factory cost. Calculate prime cost, factory cost. The management wants to make a profit of 15% on the gross cost. Calculate the selling price of each article.

Solution

Given data

Direct material cost = Rs.200

No. of products = 100

Direct cost = Rs.200

Prime cost = Direct cost + Direct material cost + Direct expenses.

= 200 + 260

= Rs.460.

Factory on cost = 30% of Prime cost = 30 % of 460 = Rs.138

Factory cost = Prime cost + Factory on cost

= 460 + 138

= Rs.598.

Overheads = 25% of Factory cost = 25% of 598 = Rs.149.5.

Total cost = Factory cost + Over heads.

= 598 + 149.5

= Rs.747.5

Profit = 15% of Total cost = 15% of 747.5 = Rs.112.12.

Selling price = Total cost + Profit

= 747.5 + 112.12

= Rs.859.625

For each piece = 859.625 / 100 = Rs.8.59 = Rs.8.59.

Selling price for each piece = Rs.8.59.

Example 03

The market price of a lathe is Rs.8000 and the discount allowed to the distributor is 15% of the market price. It is found that the selling expenses cost is ¼ of the factory cost and if the material cost, cost and factory over head charges are in the ratio of 1:4:2. The factory makes what profit on each lathe. The material cost is Rs.600. Neglect other overheads.

Solution

Market price = Rs.8000

Discount allowed 15%

Amount of discount = 8000x15 / 100

= Rs.1200.

(Selling cost) selling price at factory

= Market price - Discount.

= 8000 - 1200

= Rs.6800.

Selling cost = factory cost + selling expenses + admin exp. + profits

Selling expenses = ¼ (factory cost)

Factory cost = direct materials + cost + overhead

Material cost = Rs.600.

cost = 4 (ratio) x 600

= Rs.2400.

Overheads = 2(ratio) x 600

= Rs.1200.

Factory cost = 600 + 2400 + 1200

=Rs.4200.

Selling expenses = ¼ of Factory cost

= ¼ of 4200

= Rs.1050.

Selling price at factory = Factory cost + Selling expenses + profit

6800 = 4200 + 1050 + profit

Profit = 6800-4200-1050

Profit on each lathe = Rs.1550

Example 04

A factory is producing 2000components per hour on a machine. Its material cost is Rs.750, cost is Rs.490 and the direct expense is Rs.150. the factory on cost is 150% of the total cost and office on cost is 30% of the total factory cost. If the selling price of each component is Rs.1.30. calculate whether the management is going in loss or gain and by what amount.

Solution

Material cost = Rs.750.

Cost = Rs.490.

Direct expenses = Rs.160.

Factory on cost (exp.) = 150% of cost

= 150 x 490 / 100

= Rs.735.

Total factory cost = 750 + 490 + 160 + 735

= Rs.2135.

Office on cost = 30% of factory cost

= 30 x 2135 / 100

= Rs.640.5

Total cost for 2000 components = 2135 + 640.5

=2775.50

Cost For each component = 2775.50/2000

= Rs.1.38.

Selling price = 1.30

Loss = cost for each component - selling price of each component = 1.38 - 1.30

= Rs.0.08 / piece

Loss = Rs.0.08/piece

Break even analysis

Break even analysis is concerned with the finding the point at which income and cost are equal. One widely used method of analyzing cost is to break them down into fixed and variable costs. Fixed costs are incurred no matter the volume of sales. Examples are equipment depreciation, taxes, insurance and administrative overheads. Variable costs are those directly associated with the amount produced or sold. Examples are direct , direct material and commissions of the sales force.

Total cost = fixed cost+ (variable cost) (number of items)

$$\text{Unit (averagecost)} = \frac{\text{totalcost}}{\text{number of units}}$$

$$= \frac{\text{fixed cost}}{\text{number of units}} + \text{variable cost per unit}$$

Break even point: The point at which neither profit nor loss is made is known as the break even point.

$$\text{B.E.P} = \frac{\text{Fixedcost}}{\text{Selling price} - \text{Variable cost}}$$

Worked out examples

Example1: To make a particular component require an over head (fixed) cost of Rs 5000 and variable unit cost of Rs 10 per unit. What is the toatal cost and the average cost of producing a lot of 1000? If the selling price is Rs 25 per unit, what is the break even point?

Solution:

Total cost = Rs 5000+ (10x 1000) = Rs15000

Average cost= Rs 15000/1000= Rs 15 per unit

Let X be the break even point

Therfore 25x = Rs 5000+Rs10x

15x = Rs 5000

X = 5000/15

= 333.3 units

Break even point occurs when 333.3 units are produced and sold.

Alternatively

$$\text{B.E.P} = \frac{\text{Fixed cost}}{\text{Selling price} - \text{Variable cost}}$$

$$= \frac{5000}{25 - 10}$$

= 333.3 units

Material costing

All the expenses incurred on materials, starting from the purchase to the time till the material is ready for use, constitute the material cost. Material cost constitutes 25-65% of total production cost. Therefore all efforts should be made to control the material cost. In order to exercise control over material cost big companies have a separate department known as Materials Management department.

Material management department

The following are the functions of the Material Management department:

1 To reduce material cost.

2 To control inventories.

3 To ensure uniform flow of material for production.

4 To ensure right quantity at right time.

5 To find substitute for imported items and to justify its use.

Classification of materials

These can be further classified into

- Direct materials.
- Indirect materials.

Direct Materials

These are those materials which when operated or processed in factory shops through various stages form the final useful shape of the main product or component of part of the main product. These are also known as 'Productive Materials'. Examples of 'Direct Materials' are: Wood for furniture, Steel for bolts.

Indirect Materials

These are those materials which are essentially needed in various shops for helping the materials to be converted into the final useful shapes. Difference between direct and indirect forms of materials can be easily understood by the following example:

Suppose a person continuously working in Milling Machine Shop is cutting gear teeth on cast iron blanks. Now the cast iron blank, of which the gear is made, will be the 'direct material' while the coolant required for cooling the cutter, grease and lubricating oil need for lubricating the machine, kerosene oil and cotton waste etc. needed for cleaning the machine are known as 'Indirect Materials'.

Calculation of material cost

For the calculation of material cost followed procedure may be adopted:

1 Calculate the volume of each component by applying the measurement. For the calculation of volume, necessary machining allowance must be added on the sides, which are required to be machined.

2 Add the volume of all components to get the total volume of the product.

3 Multiply this volume by the density to get the weight of the material.

4 Multiply the cost per unit weight to the total weight of the material to get the cost of the materials.

Elements of material cost

The following are the elements of material cost

1 Cost of material purchased: It is the amount paid to the supplier including tax, duty etc.

2 Procurement cost: It is the cost of getting quotation, processing them, placing orders, receiving, inspection and payment of bill etc.

3 Inventory carrying cost: This includes the expenditure on insurance, storage, depreciation, interest on the value of stores.

4 Material handling cost: It is the handling cost involved in receiving, storing and issuing of materials.

5 Material loss: It is the cost of material lost in storage, in transit or in operation.

6 Indirect expenses: These are the expenses like the salary of purchase department, stores, rent, repair and maintenance of stores etc.

7 Scrap and surplus: It is the cost of materials which have not been used for very long time.

Valuation of material issued from stores

To find out the cost of materials issued from stores several methods are used. The choice depends on the nature of material and type of business.

The various methods are:

a First in first out method.

b Last in first method.

c Average price method.

d Fixed price method.

e Actual cost method.

f Current value method.

g Inflated price method.

a First in first out method

First in first out method

In this system the materials first received are issued first. Materials from second lot are issued only after the first lot is over. In this system the materials received in stores are entered at actual cost. The prices are charged at the cost at which that lot was purchased.

1 Advantages and disadvantages of first in first out method.

Advantages

1 Only actual cost of the material is considered.

2 Calculation is very simple.

Disadvantage

1 Current market price is not considered.

Last in first out method

- In this system the last lot of a particular item is issued first. The price of the material issued is charged at the rate of the last material received. When the last lot is finished then the price of the second last lot is considered.
- Advantage: current price is considered.
- Disadvantage: Old stock always remains in stock.

Average price method

- In this method average cost of the material is considered.
- Two methods in use for calculating averages are:

1 Simple average method: It means the average cost of material in hand at the time of issue.

2 Month end average method: In this method average cost of material is calculated at the end of each month and is considered for the following month.

Fixed price method

- In this method the issued material is charged at a predetermined rate, for a fixed period. Mostly one year rate is charged.

Actual cost method

- In this method actual cost is charged for the material to be issued from time to time.

Current value method

- In this method the material is issued at the existing market price.

Inflated price method

- In this method the cost is hiked by some percentage.

Example: First in first out

- In this method the charged cost is slightly more than the purchased price.

Date	Receipts			Issues			Balance		
	Qty	@Rs	Amt.Rs	Qty	@Rs	Amt.Rs	Qty	@Rs	Amt.Rs
Jan1	-	-	-	-	-	-	20	1.00	20.00
Jan9	100	1.0	100.00	-	-	-	120	1.00	120
Jan13	-	-	-	80	1.00	80.00	40	1.00	40.00
Jan14	300	1.10	330	-	-	-	40	1.00	40.00
							300	1.10	330.00
Jan18				40	1.00	140.00	140	1.10	154.00
				160	1.10	176.00			
Jan20	200	0.90	180.00	-	-	-	140	1.00	154.00
							200	0.90	180.00
Jan24	-	-	-	40	1.00	154.00	90	0.90	81
				110	0.90	99.00			

Example: Last in First out:

Date	Receipts			Issues			Balance		
	Qty	@Rs	Amt.Rs	Qty	@Rs	Amt.Rs	Qty	@Rs	Amt.Rs
Jan1	-	-	-	-	-	-	40	1.00	40.00
Jan6	300	1.2	360.00	-	-	-	40	1.00	40.00
							300	1.20	360.00
Jan8	-	-	-	200	1.20	240.00	40	1.00	40.00
							100	1.2	120.00
Jan12	300	1.10	330	-	-	-	40	1.00	40.00
							100	1.20	120.00
							300	1.10	330.00
Jan16				200	1.10	220.0	40	1.00	40.00
							100	1.2	120.00
							100	1.10	110.00
Jan20	200	0.90	180.00	-	-	-	40	1.00	40.00
							100	1.2	120.00
							100	1.10	110.00
							200	0.90	180.00
Jan24	-	-	-	200	0.90	180.00	40	1.00	40.00
				100	1.10	110.0	50	1.2	60.00
				50	1.2	60.00			

Costing & expenses

Labour cost is the cost incurred by the employer in the employment of the labour. Labour cost comprises of the salary paid, bonuses, gratuities, cost of food, HRA, LTA, CCA, Medical claim, insurance etc.

Objective of costing

1 To detect undesirable labour costs and to take remedial measures.

2 To provide data for formulating labour policies and help the management while negotiating with the employees in case of labour demand for higher wages arises.

3 To know the expenditure incurred on labour for different jobs.

Wages and incentives

- **Wages:** These are the payments made to the employee by the employer for the work done.
- **Incentives:** An incentive is any factor (financial or non-financial) that provides a motive for a particular course of action. These are the extra things given over and above the normal wages to influence or encourage the worker.

Wages

- **Nominal Wages:** It is the amount of money paid to the worker for the efforts put in by him with out any other benefits.
- **Real wages:** It refers to the amount of necessaries, comforts, luxuries and cash payments which a worker can get in return for his effort and work.
- **Living wages:** When the wages can meet the requirements of the family and some insurance against misfortunes etc is known as living wages.
- **Fair wages:** It is actually the wages which can meet other necessities of life in addition to food for his family.
- **Minimum wages:** It is the minimum amount required for the worker to maintain himself according the needs.

Fair wages

The fair wages will depend upon:

1 Production capacity of the worker.

2 Prevailing market rates.

3 The level of the national income and distribution.

4 The place of the industry in the economy of the country.

5 The bargaining power of employer and employee.

Minimum wages

The objectives of the minimum wages are:

1 To protect those sections of the working force whose wages are very low?

2 To prevent exploitation of workers and secure a wage according to the values of the work done.

3 To promote peace in industry.

4 To improve the normal standard of life.

Calculation of direct cost

For the purpose of calculation of cost, estimator must have knowledge of all operations, which are carried out for the manufacture of the product, the tools and the machines, used for production. Estimator should also take the advice of production department about the correct estimated time for each operation. He should also consider various allowances like.

- Set up time
- Operation time,
  - a Handling time
  - b Machining time
- The removal time
- Miscellaneous allowance
  - a Personal allowance
  - b Fatigue allowance
  - c Tool changing and grinding allowance
  - d Measurement checking allowance
  - e Other allowance for cleaning, oiling, getting stocks etc.

Expenses

In each factory there are several other expenditures, such as cost of advertisement, building rent, depreciation charges of plant and factory building, cost of packing, cost of transportation, salaries and commission to salesmen etc. All these expenditure are known as 'Expenses'. So we can say that except direct material and direct cost, all other expenses, which are incurred in the factory, are known as "Expenses".

The cost of indirect material and indirect is also included in the expenses. Expenses may be of two classes,

- Direct or Chargeable Expenses
- Indirect Expenses.

Direct expenses

These are those expenses, which can be charged directly to a particular job and are incurred for that specific job only. For example, cost of special jigs and fixtures, cost of some special patterns and cost of experimental work on a particular job etc.

Indirect expenses

These are also known as overhead charges, on cost, burden or indirect charges. These can be further classified as:

- Factory expenses
- Administrative expenses
- Selling expenses and
- Distribution expenses

Factory Expenses

These overheads include all the expenditures made on the actual operation of the product in the plant. Such as indirect materials. It is also named as works on cost. The various factory over heads are:

1 Indirect material cost: Oils, greases, coolant, cotton waste etc.

2 Indirect labour cost: Salaries of supervisors, Managers, store keepers etc.

3 Repair and maintenance charges: Machine and parts

4 Insurance premium: Building and machines

5 Power charges: Electricity, steam, diesel ,gas etc

6 Depreciation charges: Building, machine, furniture etc

Administrative Expenses

These overheads include all the expenditures made on the salaries of general office staff and executive staff, telegraph telex, fax, computer and telephone charges, depreciation of office building and equipment etc.

This is also known as establishment on cost or office expenses.

Selling Expenses

These overheads include all the expenditure made on the salaries of persons working sales department, advertising expenses and agency expenses etc.

Distribution Expenses

These overheads include all the expenses made on holding finished stock, dispatching them to the customer and packing cost etc.

Allocation of overheads

The following are the main methods of over head allocation:

1 Percentage on Direct labour cost.

2 Percentage on Direct material cost.

3 Percentage on prime cost.

4 Man hour rate.

5 Machine hour rate.

6 Combination of man and machine hour rate.

Percentage on prime cost

$$\text{Percentage Over heads} = \frac{\text{Total over heads}}{\text{Prime cost}} \times 100$$

Percentage on direct cost

$$\text{Percentage Over heads} = \frac{\text{Total Over heads}}{\text{Direct Labour Cost}} \times 100$$

Percentage on direct material cost

$$\text{Percentage Over heads} = \frac{\text{Total Over heads}}{\text{Direct Material Cost}} \times 100$$

Percentage on quantity of production

$$\text{Over head/unit} = \frac{\text{Total over heads}}{\text{Quality of production}}$$

Hourly rate

$$\text{Machine Hour Rate} = \frac{\text{Overhead Expenses for Specific Machine}}{\text{Total Productive Machine Hours}}$$

$$\text{Man Hour Rate} = \frac{\text{Total Overhead Costs}}{\text{Total productive Hours}}$$

Basic machine hour rate

Various factors which come into machine hour rate calculation are:

1 Depreciation.

2 Interest on capital cost.

3 Energy cost.

4 Space cost.

5 Maintenance and service cost.

6 Scrap value.

Depreciation

Over time, machinery values decrease due to use, age, and obsolescence. This loss in value is known as depreciation. So some money must be set aside every year from the profits so that when the machine becomes uneconomical it can be replaced by another one.

The reasons for fall in value of the machines are:

- Wear and tear.
- Physical decay.
- Improper maintenance.
- Accidents.
- Inadequacy.
- Obsolescence.

Cause of Depreciation

1 **By wear and tear:** The depreciation in wear and tear is the cost of replacement of the worn out parts

2 **By physical Decay:** The machines decay due to climatic conditions even if sufficient care is taken. Machinery exposed to weather will deteriorate and subsequently depreciate in value at a faster rate than covered equipment.

3 **By improper maintenance:** The machine value is reduced if proper maintenance work is not done.

4 **By Accidents:** Loss in value due to accidents is one of the causes of depreciation.

5 **By Inadequacy:** It is a fact that even after proper maintenance the efficiency of the machines falls down. Secondly even if the machine is working satisfactorily the capacity of the machine is below the increasing requirements. This type of depreciations where a machine is to be replaced while functioning is known as depreciation by inadequacy.

6 **By obsolescence:** This type of depreciation results when a functioning machine is to be sold out or scrapped because of new invention of a high performance machine.

Calculation of depreciation

There are different ways to calculate depreciation. They are:

1 Straight line method.

2 Annuity method.

3 Sinking fund method.

4 Reducing balance method.

5 Sum of years digit method.

6 Production unit method.

7 Machine hour method

8 Insurance policy method

9 Revaluation method.

10 Repair provision method.

Straight line method

- This method of depreciation distributes the annual depreciation cost equally over the life of a machine. It means that one should deduct the scrap value from the original value and divide the remaining value by the number of years of use full life. This method is also known as fixed installment method because every year fixed amount is deducted and no consideration is made about the maintenance and repair charges for the machine.

If 'C' is the initial cost of the machine, 'S' is the scrap value and 'N' is the number years of life of machine then the depreciation amount per year.

$$D = \frac{C - S}{N}$$

Worked out example

A machine is purchased for RS. 15,000 and its expected life is 9 years with a scrap value of RS. 3050. What is the rate of depreciation by straight line method and the book value at the end of 3 years?

Given

Initial value C = RS.15, 000

Scrap value S = RS.3, 050

No. of years N = 9 years

Solution

Depreciation rate,

$$D = \frac{C - S}{N}$$

$$= \frac{(15000 - 3050)}{9}$$

= 1327.77

Depreciation fund at the end of 3 years

= 3 X 1327.77

= RS.3983.31

Book value at the end of 4 years

= 15000-3983.31

= RS.11, 016.69

Production unit method

- This method is for providing depreciation by fixed rate per unit of production calculated by dividing the value of the asset by the estimated number of units to be produced during its life.

Machine hour method

- In this method the rate of depreciation is calculated by considering the total number of hours a machine runs in a year. The life of the machine is estimated on the basis of its service period. Total number of hours are calculated and the rate per hour is fixed that will reduce the machine or unit to its residual value at the end of its life.

Insurance policy method

- In this method an insurance policy is made for the amount required to replace the machine or unit at the end of its usual life. Premiums are paid on the insurance policy. When the policy matures the insurance company provides sufficient sum to replace the machine at the end of expiry of its life.

Revaluation method

- This method is for providing depreciation by means periodic charges each of which is equivalent to the difference between the values assigned to the asset at the beginning and at the end of the period.

Repair provision method

- In this method depreciation is provided by means of periodic charges each of which is a constant proportion of aggregate of the cost during its life.

Interest on capital cost

While preparing cost accounts, interest on capital is also considered. The rate of interest is that which would have been available. If that capital is deposited in bank. By charging interest, cost of product increases and thus profit seems to be lesser.

Interest should be charged on the products produced because;

1 Real profit is not received until interest on capital is charged.

2 The stocks of finished or raw parts cost more because of the rent and the interest on blocked money.

3 If the capital is borrowed for business, the interest has to be paid. Similarly, if the manufacturer himself provides capital he should be credited by sum equal to the interest.

(Note: 50% of capital value is taken for computation of interest.)

Worked out Example 1

Find out the interest on capital.

The total capital cost	= Rs.1, 60,000/-
Bank interest	= Rs.15%
50% of capital value	= Rs.80, 000/-
Interest per year	= (80,000 X 15)/100
	= Rs.12, 000/-

Energy cost

The energy cost is usually charged monthly. The total consumption of all motors in the machine should be taken. All these motors will not work at the same time and also will not work for the full time of machine operation. Hence it will be sufficient that we take 50% of the total kW rating.

Worked out Example1

A Surface Grinding machine has 4 motors. Motor for main spindle is 1kw. Motor for hydraulic system is 1kw. Coolant and dust collector motors are 0.5kw and 0.3kw. If one unit of energy costs Rs.2. find out the energy cost.

Total power	= Rs.2.8kw
Energy consumed/hour (50%)	= 2.8/2
	= 1.4kW

Energy cost = 1.4 X 2 = Rs.2.80/hr.

Space cost

Space cost is the rent/sq. meter multiplied by no. of sq. meter utilized by the machine. The area utilized by the machine is not merely the area occupied by the machine. It also consists of area set apart for the operator, a portion of gang ways, passage for men to move about, the area needed by the supervisory staff, etc.

Worked out Example1

Find the space cost of the machine which occupies 10 sq. meters on further used 5% of 1000 Sq. meter building rented at Es.500/p.m.

Area utilized by the machine = area occupied + extra space used

= 10m2 + 5/100 X 1000 m^2

= 10 + 50

= 60 m^2

Space charges	= 60 X 500/1000
	= Rs.30/p.m.
Space charges/year	= Rs.360/year.

Maintenance cost

These costs are routing work like oiling, clearing, replacement of worn out parts.

Maintenance and servicing is taken on actual amount with past experience.

Machine hour calculation is considered under following conditions;

1 Single shift operation.

2 Objective of utilization - for production/training/tending

3 Place of utilization - near an area where vibration is more.

Scrap value

Though the book value of the machine at the end of its life is Rs.1/-. The machine in this form can be used and if condemned few spare parts still has got some value. This value is known as scrap value.

Working hours

A factory generally has to work 48 hr./week. We have 52 weeks/year.

National festivals, holidays and leave facility provided to an employee accounts for six weeks in a year.

Therefore hours available/year	= 48 X (52-6)
	= 2208 hrs.

Considering machine breakdown, power failure, operator absenteeism and non-productive activity (strikes and lay off). The hours lost in these accounts, amounts to as high as 30% of 2208.

Therefore balance hours available

= 2208 - 0.3 X 2208

= 2208 - 662.4

= 1545.6 hrs.

(Note: For calculation we conider 1500 hr. of productive hour in a year.)

Worked out examples

Example 01

Find the machine hour rate for a machine, with the following details.

Cost of the machine	= Rs. 80,000/-
Bank interest	= 8%
Installation & erection charges	= Rs. 1,500/-
Energy charges / year	= Rs. 650/-
Space cost / year	= Rs. 480/-

Maintenance cost / year = Rs. 1,005/-

Assumed life of the machine = 10 years

Solution

Capital cost = Rs. 80,000/-

Installation & Erection cost = Rs. 1,500/-

Total Capital cost = Rs. 81,500/-

i Depreciation

$$= \frac{C - S}{N}$$

$$= \frac{81,500 - 0}{10}$$

= Rs. 8,150/-

ii Interest on Capital cost

$$\text{Average Capital} = \frac{81,500}{2}$$

= Rs. 40,750/-

$$\text{Bank interest rate 8\%} = \frac{40,750 \times 8}{100}$$

= Rs. 3,260/-

iii Total cost

Depreciation = 8,150/-

Interest = 3,260/-

Energy cost = 650/-

Space cost = 480/-

Maintenance cost = 1,005/-

Total = 13,545/-

$$\text{Machine hour rate} = \frac{13,545}{1,500}$$

= Rs. 9.03/hr.

Example 2

Calculate the machine hour rate with the following data.

Cost of the machine = Rs. 1,20,000/-

Cost of a accessories = Rs. 15,000/-

Erection charges = Rs. 1,800/-

Installation charges = Rs. 1,200/-

Bank interest = 8%

Machine has 4 motors of following capacity

1 kW, 1 kW, 0.2 kW, 0.5 kW

Cost per unit = 40 paise

Space cost per month = Rs. 100/-.

Maintenance cost = 8% of depreciation

Assumed life of the machine = 10 years

Solution

Cost of Machine = 1,20,000/-

Cost of accessories = 15,000/-

Erection charges = 1,800/-

Installation charges = 1,200/-

Total Capital cost = 1,38,000/-

i Depreciation

Depreciation

$$= \frac{C - S}{N}$$

$$= \frac{1,38,000 - 0}{10}$$

= Rs. 13,800/-

ii Interest

$$\text{50\% of capital} = \frac{1,38,000}{2}$$

Bank interest @ 8% = 69,000 x 0.08

= Rs. 5,520/-

iii Energy charges

Average = 1.0 + 1.0 + 0.2 + 0.5 = 2.7 kW

= 2.7/2

= 1.35 kW

$$\text{Charges per year} = \frac{1.35 \times 1500 \times 40}{100}$$

= Rs. 810/-

iv Space cost

Space cost/year = 100 x 12

= Rs. 1,200/-

v Maintenance cost

Maintenance cost = 8% depreciation

= 13,800 x 0.08

= Rs. 1,104/-

Total cost

Depreciation = 13,800/-

Interest on Capital cost = 5,520/-

Energy charges = 810/-

Space charges = 1,200/-

Maintenance charges = 1,104/-

Total = 22,434/

Machine hour rate $= \frac{22,434}{1,500}$

= Rs. 14.95/-

Example 3

Calculate the machine hour rate with the given details.

Cost of the machine = Rs. 95,000/-

Cost of accessories = Rs. 8,600/-

Friction charges = Rs. 1,200/-

Installation charges = Rs.1,000/-

Bank interest = 7%

Machine has 3 motors of capacity

1.5 kw, 0.5 kw, 0.3 kw Cost per unit = 45 paise

Machine occupies 20m² area.

Area utilized by the machine = 5% of 1000 cm2 building rented at Rs. 1,500/- p.m.

Maintenance cost = 8% of depreciation

Assumed life of the machine = 10 years

Scrap value of machine after 10 years = Rs. 15,800/-

Solution

Cost of machine = 95,000/-

Cost of accessories = 8,600/-

Erection charges = 1,200/-

Installation charges = 1,000/-

Total = 1, 05,800/-

i Depreciation

$$= \frac{C - S}{N}$$

$$= \frac{1,05,800 - 15,800}{10}$$

$$= \frac{90,000}{10}$$

= Rs. 9,000/-

ii Interest on Capital

Average $= \frac{1,05,000}{2}$

Interest @7% = 52,900 x 0.07

= Rs. 3,703/-

iii Energy cost

Total capacity of motors = 1.5 + 0.5 + 0.3 = 2.3 kw

50 % for calculation purpose $= \frac{2.3}{2}$

= 1.15 kw

Energy cost per year $= \frac{1.15 \times 1,500 \times 45}{100}$

= Rs. 776.25

iv Space Charges

Machine occupies = 10 sq. meters

Machine utilizing area $= \frac{1000 \times 5}{100}$

= 50 m²

Total area = 10 + 50 = 60 m²

Rent per month = Rs. 1,500/-

Per year = 1,500 x 12 = Rs. 18,000/-

Rent per m2/year $= \frac{18000}{1000}$

= Rs. 18/-

∴ Space cost = 60 x 18

= Rs. 1,080/-

v Maintenance cost

Maintenance cost = 8% of depreciation

= 9,000 x 0.08

= Rs. 720/-

Depreciation = 9,000

Interest on Capital cost = 3,703

Energy cost = 776.25

Space charges = 1,080

Maintenance cost = 720

Total = 15,279.25

∴ Machine hour rate $= \frac{15,279.25}{1500}$

= Rs. 10.18

Example 4

Calculate the machine hour rate with the given details.

Cost of machine = Rs. 1,30,000/-

Cost of the accessories = Rs. 22,000/-

Cost of the accessories likely to increase over the 10 years to 60%

Erection charges = Rs. 2,000/-

Installation charges = Rs. 1,500/-

Bank interest = 9%

Machine has 4 motors with total 4.2 kw = 40 paise/unit

Maintenance charges = 5% of depreciation

Scrap value of the end of 10 years = Rs. 26,000/-

Life of the machine = 10 years

Machine occupies 20 m^2 in a building of 1200 m^2, rented at Rs. 1,800/- p.m. Machine utilizes 6% of the total area.

Solution

Cost of Machine = 1,30,000/-

Cost of accessories = 22,000/-

Total = 1,52,000/-

Future worth of machine

Replacement cost = 1,52,000 x 1.6

= 2,43,200/-

Replacement cost = 2,43,200/-

Erection charges = 2,000/-

Installation charges = 1,500/-

Total = 2,46,700/-

i Depreciation

$$\text{Depreciation} = \frac{C - S}{N}$$

$$= \frac{2,46,700 - 26,000}{10}$$

= Rs. 22,070/-

ii Interest on Capital

$$\text{50\% of capital for Calculation purpose} = \frac{2,46,700}{2}$$

= 1,23,350 x 0.09

= Rs. 11,101.50/-

iii Energy cost

$$= \frac{4.2}{2} = 2.1 \text{ kw}$$

$$= \frac{2.1 \times 1,500 \times 40}{100}$$

= Rs. 1,260/-

iv Space Charges for 1 year

Machine occupies = 20 m^2

Machine utilizes 6% of 1200 m^2 area

$$= \frac{1200 \times 6}{100} = 72 \text{ m}^2$$

Total area = 20 + 72 = 92 m^2

Rate per year

1200 m^2 = 1800 x 12

$$= \frac{92 \times 1,800 \times 12}{1200}$$

= Rs. 1,656/-

v Maintenance cost

= 5% of depreciation

= 22,070 x 0.05

= Rs. 1,103.5/-

Depreciation = Rs. 22,070/-

Interest on Capital = Rs. 11,101.50/-

Energy cost = Rs. 1,260/-

Space cost = Rs. 1,656/-

Maintenance = Rs. 1,103.50/-

Total = Rs. 37,191/-

$$\text{Machine hour rate} = \frac{37,191}{1500}$$

= Rs. 24.79/-

Business transactions

The main aim of doing business is to earn profit. The businessman will either purchase goods in one market and sell it at a higher price in another market or produce goods of his own and sell with a profit. But this is not true in all cases. He may be able to sell the goods at a loss or with a low margin. However he will be anxious at the end of the year to know whether he has made profit or loss and what is the financial situation of the company on a particular date. More over in big business information is required for planning, control, evaluation of performance and decision making. This information can be provided only if proper records are maintained.

In order to achieve the above purpose it would be necessary to record business transactions according to certain standards. Book keeping (in elementary stage) and Accounting (in advanced stage) is the name given to such a system.

Basic terms

1 **Creditor:** An account is creditor when it gives. Debtor is a person who owes money to the business. The group of debtor is known as sundry debtors.

2 **Debtor:** An account is debtor when it receives. Creditor is a person to whom the business owes money. The group of creditor is known as sundry creditors.

3 **Goods:** This includes all articles, commodities or merchandise in which the business deals.

4 **Assets:** The properties owned by a business are called assets. Any physical thing or right owned that has a money value is an asset.

5 **Equity:** The rights to properties are called equities. Equities are of two types: (a): liabilities (b): capital.

6 **Liabilities:** The equity of the creditor is known as liabilities (what is belonging to others).

7 **Capital:** The equity of the owners is called capital

8 **Income:** A favorable change in owner's equity.

9 **Expenditure:** Expenditure takes place when an asset or service is acquired. E.g. purchase of goods, construction of building ,garden etc

10 **Expense:** An expenditure whose benefit is enjoyed or finished. E.g. salary, rent, travel, advertising, cost of goods sold etc.

11 **Loss:** Expenditure with out any benefit to the company.

12 **Drawings:** Any amount drawn by the owner for personal use.

13 **Voucher:** Any written document in support of a business transaction.

14 **Journal:** Each transaction in a business is recorded on the date and time of occurrence in a book called journal.

15 **Ledger:** Ledger is a book in which separate pages are allotted for each record we wish to keep. The entries from the journal for different accounts are posted in the pages allotted.

16 **Subsidiary registers:** In big business, transactions are many and to reduce the work of journal some subsidiary registers like cash, sales and purchases etc are maintained. Posting in ledger is made directly from these registers.

17 **To foot:** To foot is to total the amounts in a column, such as a column in a journal or a ledger.

18 **Note receivable:** The written promise for a future collection of cash within a specified period. Usually includes the requirement for interest to be paid.

19 **Notes Payable:** A written promise by the business for a future payment to a creditor within a specified period.

Book keeping

Book keeping: Book keeping is the art of recording business transactions systematically. It is the art of reducing each transaction to its simplest form.

Transaction: Transaction is the exchange of money or money's worth between one party another.

System of Book keeping

The purpose of doing business is to make profit. Therefore it is important that the financial implications of each transaction are correctly recorded.

There are two different systems of maintaining accounts:

1 **Single entry system:** Rather than dealing with debits and credits, some businesses just record one side of the transaction. This is a single-entry accounting system.

2 **Double entry system:** The double-entry system provides checks and balances to ensure that your books are always in balance. In double entry accounting every transaction has two journal entries:a debit and a credit. Debits must always equal credits. Because debits equal credits, double-entry accounting prevents some common bookkeeping errors.

The double entry book keeping is that system which recognizes and records both the aspects of transactions. Every transaction has two aspects viz. (1) the receiving of some benefit called the receiving aspect or gaining aspect or debit aspect and (2) the giving of some other benefit called the giving aspect, losing aspect or the credit aspects.

There are three different stages in the double entry system of book keeping.

They are:

1 All transactions are first recorded in the order in which they take place in a book called journal.

2 Afterwards the entries in the journal are posted to the appropriate accounts in a book called 'Ledger', periodically, say weekly, monthly, etc. This helps us to ascertain the exact position of each account on a particular date.

3 Finally, at the end of the accounting period, all ledger balances are taken into a statement called trial balance to check the arithmetical accuracy of books of accounts and then final accounts (i.e. Trading and profit and loss account and balance sheet) are prepared. This is to ascertain the result of the business (either profit or loss) for the accounting period and the financial position of the business as on the last day of the accounting period.

Double entry system

You provide consulting services, on account, to one of your regular customers, GTTI, for Rs1, 500. When you write up the invoice, you would make the following book keeping entry in your sales journal.

	Debit	Credit
Accounts receivable	1500	
Consulting revenue		1500

- These entries show that your accounts receivable (a balance sheet account) has increased by Rs1, 500, and your consulting revenue (an income statement account) has also increased by Rs1, 500.
- Upon receipt of the invoice, your customer sends you a check for Rs1, 500 in payment of her account. When you receive the check, make the following entries in your sales journal.

	Debit	Credit
Cash	1500	
Accounts receivable		1500

- These entries show that your cash has increased by Rs1, 500 and your accounts receivable have decreased by RS1, 500.

Single entry system

In the above example, you would simply record the revenue amount of Rs1, 500 in your sales journal. However, you would also want to make a separate entry in your accounts receivable ledger, so that you keep track of all customers that owe you money.

Single vs. double entry system

A double-entry accounting system will result in more accurate financial records. Because debits must always equal credits, a double-entry system will help you find common bookkeeping errors. Such errors include an amount entered incorrectly, forgetting to record a transaction, improperly copying an amount from one page to another, and transposition errors. If your accounts don't balance (total debits don't equal total credits), you know you've made an error that must be investigated.

Why Book keeping?

Book keeping tells us:

1 The financial effect of each transaction.

2 Combined effect of all transactions.

3 Correct financial status of the firm.

4 All purchases and sales.

5 Quantity and value of goods available.

6 Assets and liability of the firm.

7 Cash availability.

Books of original entry

These are the books of accounts in which transactions are recorded originally from a source document. The following are the main books of original entry.

1 Journal

2 Cash book

3 Purchases book

4 Sales book

5 Purchases return book

6 Sales return book

7 Bills receivable book

8 Bills payable book

Journal

Journal is a book or page where accounting entries are made. Journals are sometimes referred to as books of original entry. The chronological, day-to-day transactions of a business are recorded in sales, cash receipts, and cash disbursements journals. A general journal is used to enter period end adjusting and closing entries and other special transactions not entered in the other journals.

Trading account

The trading account is an account which shows the result of buying and selling of goods or services. It summarizes transactions occurring during a trading period which have a direct relation to the goods dealt in. The result trading may be either a gross profit or a gross loss and it is the difference between sales and cost of goods sold or cost of services rendered.

Profit and Loss account

Profit and loss account is an account designed to ascertain the net profit earned or net loss incurred by a business concern during an accounting period. It is debited with all operating expenses and losses and credited with all revenue incomes and gain and profits.

In practice, the profit and loss account is prepared as a continuation of the trading account and not as a separate account. The account will be headed as Trading and profit and loss account for the period ended.

Balance sheet

A balance sheet is a statement of assets and liabilities of a business prepared with a view to ascertain the financial position of a business as on a particular date. It is prepared at end of the accounting period immediately after the preparation of the trading and profit and loss account. It is called a balance sheet because it is a sheet of balances of accounts which still remain in the trial balance after the preparation of trading and profit and loss account.

The accounting equation is expressed in financial position is called balance sheet. The balance sheet of a business shows the assets and liabilities. A business takes cash from the owner which is called capital. Loans are taken for business. Credit purchases are made for business. Business is supposed to pay to them and there for the balance at the end of the period is liability and is shown on the liability side of the balance sheet.

Lands, building, plant, machinery, furniture, balance of finished goods, debtors and cash balance are assets and are shown on the assets side of the balance sheet.

Rules to remember

1 All debits except expenses enter on the asset side. (All expenses go to profit / loss account as debits and the net loss will be entered on asset side)

2 All credits except income enter on the liabilities side. (All incomes go to profit / loss account as credits and the net profit will be entered on liabilities side)

Cost of components

Component cost calculation

The cost of a component can be calculated from the following costs.

1 Raw material cost

2 Labour cost

3 Amortization cost

4 Inspection cost

5 Tool maintenance cost

6 Secondary operation cost

7 Packing and delivery cost

Raw material cost

Raw material cost: In case of sheet metal items, first an economical strip layout is to be drawn and no. of components that can be produced out of a standard size of sheet metal is to be found out. The cost of the entire sheet is to be distributed over total number of component that can be produced. This gives the raw material cost for each component. (Suppose the scrap material yields more money then this money should be deducted from the cost of total number of components and fix a new cost for each component).

Total weight of the sheet in tons

Wt (in gms) =

$$\frac{\text{Total weight of the sheet in tons}}{\text{Total no. of components that can be produced from the sheet}}$$

In case of plastic item, the shot weight of the material is decided by the no. of cavities in the mould. Knowing the market rate of raw material, the rate per component can be calculated.

Labour cost

In order to find out the labour cost, we have to list out the various processes adopted to bring out the final component.

Labour cost is to be found out in each of the process and the sum total should be taken as labour cost.

This is calculated by the formula,

Labour cost (one process) =

$$\frac{\text{Shift rate}}{\text{No. of components produced per shifts}}$$

Amortization cost

Cost of the tool spread over no. of components is called amortization cost. This can be done over a period of time, or over number of components.

Amortization cost **=**

$$\frac{\text{Value of the tool}}{\text{no of components over which amortisation to be done}}$$

Cleaning and Inspection charges

Cleaning means de-burring, de-flashing, tumbling, etc. This cost is difficult to charge over each component especially if the components are very smell. Hence this can be charged on batches so also the inspection charges.

Shearing cost

This is the cost incurred on shearing the sheet metal into strips. This cost is also to be spread over number of components that are produced in a standard sheet.

i Surface finishing cost

Some times the components are plated to protect the surfaces. After the production of components, in presses if the customer needs the components with protective surfaces, then the expenses incurred towards this process must also be charged on the components.

This is charged in two ways, 1) On area basis 2) On weight basis.

ii Tool maintenance cost

Some components are produced in large quantities continuously for a very large period. In such cases the tools producing those components need periodic maintenance or sometimes replacements of major parts. The expenditure incurred towards this maintenance is to be charge on the component.

Sheet metal components

worked out example 1

Calculate the cost of a component given the following.

Operation	**Press**	**Shift rate**	**Press strokes per minutes**	**Strokes Utilized**	**Tool cost**
Blank & Draw	63T	Rs.125	60	50%	15,000
Piercing	25T	Rs.55	60	30%	4,500

1mm thick 45 mm wide steel strips purchased Rs.11, 900/ton; indicate that 22 components are produced for every meter length tool cost has to be recovered with in a year on assumption that the tools work on three 8 h. shift system for 270 days in a year.

1 Raw material cost

Weight of the raw material in Kg.

$$= \frac{1000 \times 45 \times 1 \times 8}{10^6} = 0.36 \text{ kg or 360 g}$$

No. of components produced in the above strip = 22 nos.

$$\text{Norm} = \frac{360}{22} = 16.36\text{g}$$

Raw material rate = Rs11.9/- per kg or Rs.0.0119/ g or 1.19 paisa/g

Raw material cost per components: 16.26 1.19 = 19.46 paisa.

2 Cost

There are two processes

a Blank and draw

b Piercing

Blank and Draw

$$\text{Cost} = \frac{\text{Shift rate}}{\text{No. of components produced per shifts}}$$

Blank & Draw stroke/min. 50% utilized.

$$\text{Component per h} = 60 \times 60 \times \frac{50}{100} = 1800.$$

No. of components per shift = 1800 x 8 = 14,400

$$\text{cost} = \frac{125}{14400} = \text{Rs.0.0086 or 0.86 paisa}$$

Piercing

60 strokes/min. 30% utilized

$$\text{Comp. per/h} = 60 \times 60 \times \frac{30}{100} = 1080$$

Components per shift = 1080 x 8 = 8640

$$\text{cost} = \frac{55}{8640} = \text{Rs.0.0063/} - \text{ or 0.63 paisa}$$

3 Amortization cost

Blank and draw

$$\text{Amortization} = \frac{\text{tool cost}}{\text{No. of components produced / year}}$$

No. of component produced per shift = 14, 400

No. of component produced per day = 14400 x 3 = 43200

No. of component produced per year = 43200 x 270 = 11664000 components

$$\text{Amortization} = \frac{15000}{11664000} = \text{Rs.0.001236/ or 0.13 paisa.}$$

Piercing tool No. of components produced per shift = 8640

No. of components produced per day = 8640 x 3

No. of components produced per year = 25920 x 270 = 6998400

Amortization cost =

$$\frac{\text{Cost of tool}}{\text{No. of components produced per year}}$$

$$= \frac{4500}{6998400} \times 100 = 0.06\text{paisa}$$

Total cost of the component

1	Raw material cost	= 19.46 paisa
2	cost (blank and draw)	= 0.86 paisa
3	cost (piercing)	= 0.63 paisa
4	Amortize (blank and draw)	= 0.13 paisa
5	Amortize (pierce)	= 0.06 paisa
		= 21.14 paisa

Cost of the component = 21.14 paisa.

Worked out example 2

Estimate the cost of the component shown in Fig 1. Give the raw material rate Rs.9.5/kg. Shift rate of the press producing the above components is Rs.200/- tool cost of Rs.9100/- is to be amortized over 4 lakh components. Press runs at 60 stroke per minute and is utilized at 30%.

1 Raw material rate

Assume the length of strip to be 1m long

No. of components that can be produced in the strip

$$= \frac{1000}{41} = 24 \text{ nos}$$

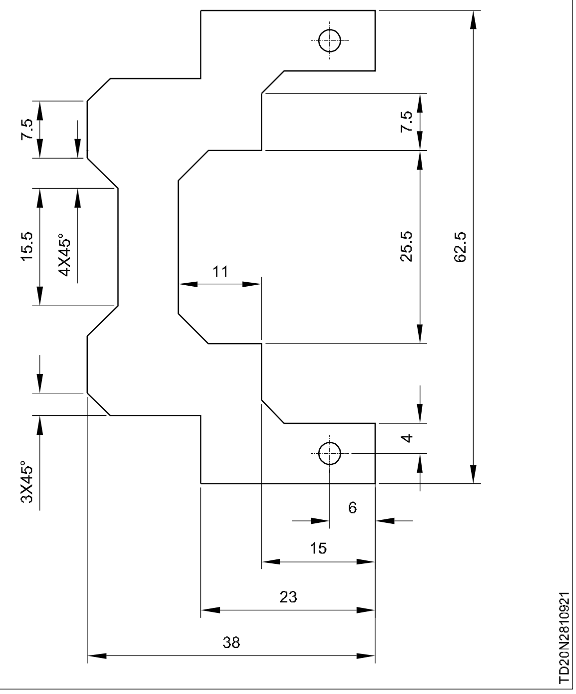
$$\text{Weight of the strip} = \frac{1000 \times 41 \times 1.6 \times 8}{106} = 0.5248 \text{ kg}$$

$$\text{Weight norm} = \frac{0.5248}{24} = 32.8 \text{ gm}$$

Raw material cost = 0.0218 9.5 = Rs.0.2071/-

Raw material rate = 31 paisa

Fig 1



2 Cost

No. of components produced per shift

60 strokes per minute 30% utilization

$$\text{Component/h} = 60 \times 60 \times \frac{30}{100} = 1080$$

Per shift = 1080 x 8 = 8640 nos.

$$\text{cost} = \frac{\text{Shift rate}}{\text{No. of components/shift}}$$

$$= \frac{200}{3640} = \text{Rs.0.23/- or 2.3 paisa}$$

3 Amortization charges

$$= \frac{\text{Cost of the tool}}{\text{No. of components over which amortization to be made}}$$

$$= \frac{9100}{400000} = \text{Rs.0.0227/-}$$

= 2.27 paisa

Cost per component

Material cost = 20.00 paisa

cost = 23.00 paisa

Tool amortization = 2.27 paisa

Total **= 45.27 paisa**

Molded components

Worked out example

Calculate the cost of molded component given the following

No. of cavities = 2

Each component weight = 20.5 g

Sprue weight = 3g

Runner weight = 1.5 g

Raw material rate = Rs.27/ kg

Raw material = polystyrene clear

Cycle time = 35 seconds

Machine capacity = 80

Machine shift rate = Rs.300/-

Mould cost = Rs.26000/-, to be amortized in one year.

Solution

Circle time = 35 seconds

$$\text{No. of shots/h} = \frac{60 \times 60}{36} = 102$$

No. of shots/shift =102 x 8 = 816

No. of cavities = 2

No. of components/shift = 316 x 2 = 1632

Machine shift rate = Rs.300/-

$$\text{cost/component} = \frac{\text{shiftrate}}{\text{No.ofcomp/shift}} = \frac{300}{1632}$$

= Rs.0.1838 or 18.4 paisa.

Raw material cost per component in paisa

$$= \frac{\text{shotwt.}}{\text{No.ofcavities}} \times \frac{\text{R.M.rate/kg}}{1000}$$

$$= \frac{45.5}{2} \times \frac{27}{1000} = 0.614 \text{ R.s} = 61.4\text{paisa}$$

Mould amortization cost

No. of components/shift = 1632

No. of components/3 shifts = 1632 x 3

No. of component 270 days of one year = 1632 x 3 x 270

= 1321920 Nos.

Mould cost = Rs.26000/-

$$\text{Amortization cost} = \frac{26000}{1321920} = 1.97 \text{ paisa=R.s 0.0197}$$

Total cost per components

1. Raw material cost	= 61.4 paisa
2. cost	= 18.4 paisa
3. Amortization cost	= 1.97 paisa
	81.77 paisa

Worked out example 2

Calculate the component cost based on the following data (amortization of the tool is not required).

Weight of the component = 30.7 g/component

Shot weight	= 35 g/components
Specific pressure	= 600 kg/cm2
Projected area	= 16.5 cm2
Machine capacity	= 80t
Number of cavities	= 6 nos.
Cycle time	= 3 min 30 seconds
Machine rate	= Rs.325/ shift
Raw material	= bakelite (filled)

Raw material cost Rs.23.5/- kg.

Solution

No. of components / shift

Cycle time = 3 min. 30 seconds

In one hour $= \frac{60}{0.5} = 17$ shafts

No. of component per shift = 17 x 6 x 8 = 816 nos.

$$\text{cost / component} = \frac{\text{Shift rate}}{\text{No. of components/shift}}$$

$= \frac{325}{816} =$ Rs.0.39 or 39 paisa.

Raw material cost / component

= Shot weight per component x raw material rate

$= 35 \times \frac{23.6}{1000} =$ Rs.0.8225/- or 82.25 paisa say 82 paisa

Total cost of the component =

Material cost	= 82
cost	= 39
	Rs.1.21

Cost of the component :Rs.1.21/-

Worked out Example 3

Calculate the component cost in the following case

Component weight	= 30g
Shot weight	= 45g

Raw material - polystyrene clear

Rate of raw material	= Rs.50kg
Machine used	= 80T

Shift rate of the machine = Rs.350/-

Cycle time = 35 seconds

No. of cavities = 2

Neglect amortization

Consider 15% profit on

1 Raw material rate =

$$\frac{\text{Shot weight in kg}}{\text{No.of cavities}} \times \text{Raw material rate}$$

$= \frac{45.5}{1000} \times \frac{1}{2} \times 30.42 =$ Rs.0.692 or 69.2 paisa

2 Charges/components =

$$\frac{\text{Shift rate}}{\text{No. of components per shifts}}$$

Cycle time = 35 seconds

No. of shots/h $= \frac{3600}{35} = 102.85$say103

No. of cavities = 2, 103 x 2 = 206 component/h.

No. of components/shift = 206 x 8 = 1648

cost = 300/1648 = Rs.0.182 or 18.2 paisa

Profit 15% on = 18 x 88888880.15 = 2.7 paisa

Total cost = 69.2 + 18.2 + 2.7 **= 90.1 paisa (or) Rs.0.9**

Activity based costing (abc)

Methods of costing

Ways to determine product cost:

1 Feeling

2 Educated guessing

3 Traditional Cost Accounting (TCA)

4 Activity Based Costing (ABC)

Activity-based costing (ABC) is a system to identify activities and assign costs to those activities-regardless of the nature or size of a business.

Traditional Methods versus ABC

Traditional cost systems (such as standard costing or job order costing) determine the value of inventory and cost of sales, but they can lead to inaccurate measurements of the total processing of products or services. Traditional systems also can lead management to make inaccurate business decisions. A management team needs the best information possible, and with ABC, accurate cost information is provided in real time.

Activity-based costing (ABC) is a method of allocating costs to products and services. It is generally used as a tool for planning and control.

ABC is a cost management method. ABC handles the overhead cost and is more accurate than Traditional Cost Accounting (TCA).

Traditional Cost Accounting (TCA)

These traditional costing systems are often unable to determine accurately the actual costs of production and of the costs of related services. Consequently managers were making decisions based on inaccurate data especially where there are multiple products.

Instead of using broad arbitrary percentages to allocate costs, ABC seeks to identify cause and affect relationships to objectively assign costs. Once costs of the activities have been identified, the cost of each activity is attributed to each product to the extent that the product uses the activity. In this way ABC often identifies areas of high overhead costs per unit and so directs attention to finding ways to reduce the costs or to charge more for costly products

Why activity-based costing preferred?

In today's manufacturing industry increasing technology and productivity improvements have reduced the relative proportion of the direct costs of labour and materials, but have increased relative proportion of indirect costs. For example, increased automation has reduced labour, which is a direct cost, but has increased depreciation, which is an indirect cost.

Traditionally cost accountants had arbitrarily added a broad percentage onto the direct costs to allow for the indirect costs. However as the percentages of overhead costs had risen, this technique became increasingly inaccurate because the indirect costs were not caused equally by all the products.

For example, one product might take more time in one expensive machine than another product, but since the amount of direct labour and materials might be the same, the additional cost for the use of the machine would not be recognized when the same broad 'on-cost' percentage is added to all products. Consequently, when multiple products share common costs, there is a danger of one product subsidizing another.

Limitations of ABC

Even in activity-based costing, some overhead costs are difficult to assign to products and customers, for example the chief executive's salary. These costs are termed 'business sustaining' and are not assigned to products and customers because there is no meaningful method. This lump of unallocated overhead costs must nevertheless be met by contributions from each of the products, but it is not as large as the overhead costs before ABC is employed.

Although some may argue that costs untraceable to activities should be "arbitrarily allocated" to products, it is important to realize that the only purpose of ABC is to provide information to management. Therefore, there is no reason to assign any cost in an arbitrary manner. Management accountants can be creative in finding other ways to represent these costs on internal reporting statements.

A typical situation in manufacturing industry:

1 Technically good product or services

2 Products or services delivered on time

3 Satisfied customers

4 Productivity around or above industry average

5 Very successful growth in the initial years

6 Profitably comes down over period of time

Common perception

1 Not enough sales to be profitable

2 Market is not conducive

3 Products are sold at very low price

Some Facts

1 An increase in sales does not necessarily increase profit

2 Some products are money makers and some are money losers

3 There are too many money losers

4 No body is sure where money is being lost

Why is a new cost management system needed?

To determine the true cost of a product or service

Why is the knowledge of the true cost of a product so important?

1 To identify money makers or money losers

2 To find an economic break even point

3 To compare different options

4 To discover the opportunities for improvement

5 To prepare and actualize the business plan

6 To improve strategic decision making

Major factors for determination of market price:

1 Customer Value

2 Competition

Note: Total manufacturing cost has only a negligible importance in final price of the product.

Total cost of a product

Total cost of a product = Prime cost+ Over head cost

Allocation of overheads by traditional cost Accounting (TCA) method

1 Arbitrarily allocates overhead to cost of products

2 Total companies overhead is allocated to the products based on the volume based measure e.g. labour hours, machine hours, material cost etc

3 Assumption: Relation between Overhead and the volume based measure

Two products: Product A and Product B

Product A:

1 hour of direct

Direct labuor cost for @20/ h = Rs 20

Demand = 100 Numbers

Product B:

2 hous of direct

Direct cost @Rs 20/ h = Rs40

Demad = 950 Numbers

Total overhead: Rs100000

Total direct = 2000h

$$\text{overhead / h} = \frac{100000}{2000} = 50$$

Product A : 1 hour of direct

Product B : 2 hours of direct

TCA over head allocation:

A: Rs50/unit

B: Rs100/unit

ABC basic principle

1 Cost objects consume activities

2 Activities consume resources

3 This consumption of resources is what drives costs

4 Understanding this relationship is critical to successfully managing over head

When to use ABC

1 Overhead is high

2 Products are diverse

3 Cost of errors are high

4 Competition is stiff

ABC steps

1 Identify activities

2 Determine cost for each activity

3 Determine cost drivers

4 Collect activity data

5 Calculate product cost

Identifying activities for ABC

1 Set up

2 Machining

3 Receiving

4 Packing

5 Engineering

Determine cost for each activity (Example)

set up = Rs 10000

Machining = Rs 40000

Receiving = Rs 10000

Packing = Rs 10000

Engineering = Rs 30000

Determining cost drivers (Example)

set-up: Number of steps

Machining : Machining hours

Receiving :Number of receipts

Packing : Number of deliveries

Engineering : Engineering hours

Activity data (Example)

Activity	Totalcost (Rs)	Product A	Cost/unit (Rs)	Product B	Rs.
Setup	10000	1	2500	3	7500
Machining	40000	100	2000	1900	38000
Receiving	10000	1	2500	3	7500
Packing	10000	1	2500	3	7500
Engineering	30000	500	15000	500	15000
			24500		75500

Overhead allocation by (abc) method

Overhead for product A = 24500/100 = Rs245

Overhead for product B = 75500/950 = RsRs79.50

Production cost TCA vs. ABC

Cost for Product A

Over head cost:	TCA: Rs50	ABC: Rs 245
Direct cost:	Rs 20	Rs20
Total:	Rs 70	Rs 265

Cost for Product B

Over head cost:	TCA: Rs100	ABC: Rs 79.50
Direct cost:	Rs 40	Rs 40
Total:	Rs 140	Rs 119.50

Advantages of (ABC)

1 More accurate cost management methodology

2 Focuses on indirect costs (Overheads)

3 Traces rather than allocates each expenses category to the particular cost object

4 Makes "indirect" expenses "direct"

Estimation of tools

Machine hour rates

1. Power saw	Rs.15/- per hour
2. Engine lathe	Rs.30/- per hour
3. Bench lathe	Rs.20/- per hour
4. Milling machine	Rs.75/- per hour
5. Coordinate drilling	Rs.75/- per hour
6. Ordinary drilling	Rs.15/- per hour
7. Surface grinding	Rs.80/- per hour
8. Cylindrical grinding	Rs.100/- per hour
9. Engraving	Rs.20/- per hour
10. Jig grinding	Rs.250/- per hour
11. Bed saw	Rs.20/- per hour
12. Spark erosion	Rs.80/- per hour
13. Wire erosion	Rs.250/- per hour
14 CNC milling	Rs.300/- per hour
15. CNC lathe	Rs.180/- per hour

Miscellaneous costs

1 Hand time skilled	Rs.50/- per hour
2 Hand time semi skilled	Rs.20/- per hour
3 Welding (arc & gas)	Rs.5/- per cm
4 Plain hardening	Rs.20/- per kilo
5 Case hardening	Rs.40/- per kilo
6 Blackening	Rs.30/- per kilo
7 Polishing (Fine finish _0.1μ)	Rs 150/per 25mm2 (5-30%of mold cost)
8 Polishing (Mirror finish_0.02 μ)	RS900/ per 25mm2

Approximate cost of standard die sets (refer suppliers price list for actual)

No.	Dies set sizes	Cost (Rupees)
1	80 x 100	1, 800.00
2	125 160	2, 400.00
3	160 200	4, 200.00
4	200 250	4, 500.00
5	250 350	7, 200.00
6	300 450	10, 200.00

No.	Mould base sizes	Cost (Rupees)
1	100 x100 x 80	6,240.00
2	120 x 100 x 80	6,960.00
3	250 x 200 x 160	10,200.00
4	250 x 200 x 260	12,120.00
5	400 x 300 x 200	20,040.00
6	400 x 300 x 225	17,700.00
7	550 x 400 x 150	30,960.00
8	550 x 400 x 425	35,520.00

Approximate cost of standard mould base (refer supplier's price list for actual)

Approximate cost of standard item (refer suppliers price list for actual)

Packing and forwarding cost.

Inland	Rs.50/kilo
International	Rs.100/kilo

Activity centers

1 Power saw

2 Turning

3 CNC turning

4 Milling

5 CNC milling

6 Surface grinding

7 Cylindrical grinding

8 Jig grinding and profile grinding

9 Spark Erosion

10 Wire EDM

11 Inspection

12 Heat treatment

13 Hand time (skilled)

14 Hand time (semiskilled)

Estimation moulds and dies

ESTIMATION SHEET FOR MOULDS & DIES		COMPONENT SKETCH	
Enq. From			
Enq. Ref			
Enq. For			
Component mater			
M/c			
No. of cavities			
Mould type		**Quote**	

Part Description		ACTVITY CENTERS														TOTAL
		1	2	3	4	5	6	7	8	9	10	11	12	13	14	
1	CORE															
2	CORE INSERTS															
3	ELECTRODE															
4	CAVITY															
5	CAVITY INSERTS															
6	ELECTRODE															
7	EJECTOR															
8	COOLING/HEAT															
9	SIDE CORE 1															
10	SIDE CORE 2															
11	SIDE CORE 3															
12	SLIDE ACTUATION															
13	LIFTER															
14	SPRUE/SPREDER															
15	TRANSFER PLUNGER POT															
16	LATCH /LOCKS															
17	SPECIAL MECHANISM															
18																
19																

REMARKS		
	Total cost	
	Labour cost	
	Overhead cost	
	STD +Raw matl+Dir cost	
	Mold base cost	
	Programming Charges	
	Design charges	
	Polishing charges	
	Trial	
	Grand total	

Documentation - 1

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

Work organisation

Work organisation is to arrange and distribute the work between the work teams in such a way that the best use is made of the available labour, materials, tools and equipment.

- to order the operations and activities of the work should follow each other.
- to decide the various work teams.
- to instruct and communicate correctly in order to avoid misunderstandings.

Organisation of work

The organisation of work includes many aspects, such as:

- Pace of work (speed of an assembly line, quotas).
- Work load.
- Number of people performing a job (staffing levels).
- Hours and days on the job.
- Length and number of rest breaks and days away from work.
- Layout of the work.
- Skill mix of those workers on the job.
- Assignment of tasks and responsibilities and
- Training for the tasks being performed.

Some common terms technically used in industry are as follows:

- Lean production
- Continuous improvement
- Just-in-time production
- Work teams
- Total productive maintenance
- Total quality management
- Outsourcing/ contracting out

Lean production

An overall approach to work organisation that focuses on elimination of any "waste" in the production / service delivery process. It often includes the following elements: "continuous improvement", "just-in-time production" and work teams.

Continuous improvement

A process for continually increasing productivity and efficiency, often relying on information provided by employee involvement groups or teams. Generally involves standardizing the work process and eliminating micro-breaks or any "wasted" time spent not producing/ serving.

Just-in-time production

Limiting or eliminating inventories, including work-in-progress inventories, using single piece production techniques often linked with efforts to eliminate "waste" in the production process, including any activity that does not add value to the product.

Work teams

Work teams operate within a production or service delivery process, taking responsibility for completing whole segments of work product. Another type of team meets separately from the production process to "harvest" the knowledge of the workforce and generate develop and implement ideas on how to improve quality, production and efficiency.

Total productive maintenance

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

Total quality management

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

Outsourcing/ Contracting out

Transfer of work formerly done by employees to outside organisations. In many workplaces undergoing restructuring, worker knowledge about the productive/ service process is gathered through "employee involvement" and then used by management to "lean out" and standardize the work process, thereby reducing reliance on worker skill and creativity. This restructuring has resulted in job loss for some underperformed, while increasing the work load and work pace for those who remain on the job. The result of these changes in work organisation is that it is no longer just machines that are wearing out-it is the workers themselves.

Different types of documentation as per industrial needs

Objectives : At the end of this lesson you shall be able to
- **state the purpose of documentation**
- **list the different types of documentation**
- **explain the documents format - batch processing, BOM, cycle time, productivity report, manufacturing inspection report.**

Documentation

Documentation and records are used throughout the manufacturing process as well as supporting processes (quality control) must meet the basic requirements. Documentation is a set of documents provided on paper, or online, or on digital or analog media, such as audio tape or CDs. Examples are user guides, white papers, online help, quick reference guides.

The stages of recording the documents is to

- prepare, review, update and approve documents.
- identify changes and current revision status of documents.
- use of applicable documents available at points of use with the control documents of external origin
- identify and distribute relevant versions to be identifiable and remain legible.
- prevent unintended use of obsolete documents and archiving.

The different types of documentation as per industrial needs includes

- Processing charts
- Bill of materials (BOM)
- Production cycle time format
- Productivity reports
- Manufacturing stage inspection report
- Job cards format
- Work activity log
- Batch production record format
- Estimation of work
- Maintenace log format

Process chart

A process chart is a graphical representation of the activities performed during manufacturing or servicing jobs. Graphical representation of the sequence of operations (workflow) constituting a process, from raw materials to finished product.

Process charts are used for examining the process in detail to identify areas of possible improvements.

The different types of process charts they are

- Operation process chart
- Flow process chart (man/ material/ equipment type)
- Operator chart (also called two handed process chart)
- Multiple activity chart
- Simo chart

The following symbol set derived from Gilbreth's original work as the standard for process charts.

Symbol	Letter	Description	Examples
O	O	Operation	Saw cut, paint, solder, package
→	M	Transport	Conveyor / Fork lift / OTR truck
☐	I	Inspection	Visual/dimension
D	D	Delay	WIP/Hold/ Queue
∇	S	Storage	Warehouse/tracked storage location

The application of symbols on a flow process chart is shown in the figure

Flow process chart(Machines)

Industry : ______________________________

Product : ______________________________

Summary				
Function	**Present**		**Proposed**	
	*	**Time**	*	**Time**
Operation				
Inspection				
Transport				
Delays				
Storage				

Details	**O→□ D∇**	**Qty**	**Time (in mins)**	**Analysis**	**Actions recommended**
Raw material from stores	O→□ D ∇				
To cutting machine	O→□ D ∇				
Cutting of material to size	O→□ D ∇				
Filling, Finishing	O→□ D ∇				
To inspection for finished size	O→□ D ∇				
To stores (Finished job)	O→□ D ∇				

Batch record forms

The documents used and prepared by the manufacturing department provide step-by-step instructions for production-related tasks and activities, besides including areas on the batch record itself for documenting such tasks.

Batch production record is prepared for each batch should include information on the production and control of each batch. The batch production record should confirm that it is the correct with standard operating procedure.

These records should be numbered with a unique batch or identification number and dated and signed when issued.

The batch number should be immediately recorded in data processing system. The record should include date of allocation, product identity and size of batch.

Documentation of completion of each significant step in the batch production records (batch produciton and control records) should include :

- Dates and, when appropriate time
- Major equipment used machinery and specific batch numbers of raw materials, reprocessed materials used during manufacturing.
- Critical process parameters records.
- Trial product or sample (if required).
- Signatures of staff for sequence of operation.
- Laboratory test results and line inspection notes.
- Achieved production against target.
- Packaging and label (if any) details.

Batch processing record : (Sample format - 1)

The format 1 used in documentation of batch processing record has the description of the job, necessarily mentioned with part number and name of the part.

A predetermined batch quantity with batch number alloted and identified with batch record number is documentation. The product reference is made with purchase order number.

The production process is descriptively written about the sequence of operation to be carried out on the product.

The manufacturer organization name, period of manufacture preferably the year with starting date of manufacture and end date of manufacturer and number of pages of document according to batch quantity processed, and total number of pages of document, inclusive of inserted pages and manufacturing facilities is provided with.

The batch processing record is signed with date mentioning name of person responsible and their designation.

The remarks if any on the process should be also mentioned then and there.

BATCH PROCESSING RECORD - FORMAT - 1

Batch Processing Record			
Description of job		Batch no. :	
Part no. : Name of part :		Batch quantity : Batch record no. : Purchase order no. :	
Description of process :			
Manufacturing Organisation :			
Period of manufacture (Year - Qtr):	Start date of manufacture:	End date of manufacture:	
Number of pages according to batch:	Inserted pages:	Manufacturing facilities:	
Total number of pages			
1. Operator / Technician	Date		Name and signature
2. Production in-charge:	Date		Name and signature
3. Section manager	Date		Name and signature
4. Plant in-charge:	Date		Name and signature
5. Production in-charge:	Date		Name and signature
Remarks (if any)			

Bill of materials (BOM) format - 2

The list of parts involved in manufacturing of an assembly hierarchially is given in this format.

The format shown is as per bureau of Indian Standards IS:11666-1985 as example for Engineering Component drawings.

The BOM in the form of tabular columns has the component marked with item number, and its name is given under description and number of is mentioned under quantity, with reference drawing ie., sub assembly/part drawing number.

The material designation as per code of practice or standards is mentioned, and any other specific notes are given under remarks column.

The BOM is placed on the manufacturing drawing containing with assembly and parts in standard sheet sizes of engineering drawing.

BILL OF MATERIAL (BOM) - FORMAT - 2
as per IS: 11666-1985

S.No	Item No.	Description	Quantity	Reference dwg no.	Material as per standard	Remark

Cycle time

Cycle time is the total time from the beginning to the end of the process. Cycle time includes process time, during which a raw material worked with to bring it closer to required form output, and delay time, during which the workpiece waiting for next operation.

The time taken to perform one operation repeatedly measured from “Start to Start” the starting point of one product’s processing in a specified machine or operation until the start of another similar product’s processing in the same machine or process. Cycle time is commonly categorized into same machine/process.

Machine cycle time

The processing time of the machine working on a part.

Auto cycle time

The time a machine runs un-aided (automatically) without manual intervention.

Overall cycle time

The complete time it takes to produce a single unit. This term is generally used when speaking of a single machine or process.

Total cycle time

This includes all machines, processes, and classes of cycle time through which a product must pass to become a finished product. This is not lead time, but it does help in determining it.

Production cycle time (Format - 3)

This format 3 should contain mentioning the organization name department / section name. The process which is being observed for analysing the cycle time is mentioned with line in charge name and the date/time of the operations, with operator name is indicated.

The time observation on each operation, sequence noted in the column, and lowest repeatable is also mentioned for each operation. The times observation for machine cycle time is also noted, with any notes be recorded in respective operations in sequence.

PRODUCTION CYCLE TIME - FORMAT - 3

Organisation Name: Department / Section :	Process:	Line Incharge:	Date/Time:

Operator :										Machine Cycle Time	Notes
Operator Sequence	Observed Times								Lowest Repeatable		

Productivity report

Productivity report to measure and review the efficiency of a person, machine, factory, system, etc., in converting inputs into useful outputs. Productivity report is computed by dividing average output per period by the total costs incurred or resources (capital, energy, material, personnel) consumed in that period.

The base document daily production report which reveals the actual output against the target plan and on investment cost incurred as mentioned above decides the cost efficiency.

Daily production report (Format 4)

The output of production is shown in the format, referring the job order no quantity, material and size, every process involved, to produce a component, quality control, packing should contain the details of planned quantity and produced quantity is recorded in the document. This is the base details for arriving the productivity report.The incurred cost is worked out considering infrastructure, raw materials and facilities.

DAILY PRODUCTION REPORT - FORMAT - 4

Date:	Daily Production Report Department: Section:	Organisation Name:

	Process - I		Process-II		Process-III		Process-IV		Quality Control		Packing	
	Planned	Completed	Planned	Completed	Planned	Completed	Planned	Completed	Planned	Completed	Planned	Completed
Job Order No. Quantity Material & Size												
Job Order No. Quantity Material & Size												
Job Order No. Quantity Material & Size												
Job Order No. Quantity Material & Size												
Job Order No. Quantity Material & Size												

Signature of section Incharge

Manufacturing stage inspection report (Format 5)

The format 5 is to monitor the production in various stages for which manufacturing stage inspection conducted for documentation to review the productivity. The format gives the details of product being inspected showing the details of customer reference by purchase order (PO) number and date, job order number and date, process involved in manufacture of product, the quality submitted for inspection. The accepted and rejected quality recorded with inspection record review date and the inspection person signature who conducted the stage inspection is recorded date wise for mentioned /specified period with start and end dates.

MANUFACTURING STAGE INSPECTION REPORT - FORMAT - 5

Organisation Name :

Status: From Date .../.../..... To Date .../.../.....

Date	Product ID/ Code	Customer	P.O No. & Date	Job Order No.	J.O Date	Process	Qty	Accepted	Rejected	Inspection Record No.	Inspection conducted by

Documentation - 2

Objectives : At the end of this lesson you shall be able to
- **state the purpose of job card and its format details**
- **explain work activity log format details**
- **state the details of batch production format.**

Job card

A job card is a document showing the details of a job to be performed in a production shop. It is used to authorize and instruct the work team to take up the production work.

Job card format - 1

Job card has the details of commencing the job, customer name, work order no, document number, reference number and date.

The details which have to be recorded about the product line description showing the operations each into recording of start time and total time of operation. The location time recorded is to track if any delay/ reasons and necessary actions if taken with remarks.

If the product has to be completed with any of the further operations in sequence, this card will travel along with job for next workstations for further operations if any to complete the requirement of job, and recorded till finishing of the job.

JOB CARD - FORMAT-1

Job Card	Doc No.	
	Rev No.	
	Date	

Order Starting Date	
Customer	
Work Order No.	

Details								
S.No.	Date	Production Line Description	Time (Minutes)			Location Time	Remarks
			Start Time	End Time	Total Time		

Work activity log format - 2

This document is to record the activity/operations performed by the operator from time to time (format) shows time duration as one hour (For whole day shift). The operator has to record every hour, activity description, equipment/ machinery/instrument used to perform the job.

Any remarks may noted by the operator to complete this record.

<u>**WORK ACTIVITY LOG - FORMAT-2**</u>

Organisation Name:

Department:

Section:

Employee Name:

Supervisor Name:

Date:

Start / Stop	Operations performed	Equipment / Machinery/ Instruments used	Remarks
8.00 to 9.00 a.m.			
9.00 to 10.00 a.m.			
10.00 to 11.00 a.m.			
11.00 to 12.00 noon			
12.00 to 1.00 p.m.			
1.00 to 2.00 p.m.			
2.00 to 3.00 p.m.			
3.00 to 4.00 p.m.			
4.00 to 5.00 p.m.			
5.00 to 6.00 p.m.			

Batch production record format - 3

This document is for recording the details of production covering the processing steps with documented page number with deviation against each in short description.

This document is to be prepared under heading description of job part number, batch number, name of the part. The processing steps number serially for each process with sequential operations in logical order with documented page number. The description of deviation are noted against each operations in sequence gives the detail of batch production record for every part.

BATCH PRODUCTION RECORD - FORMAT-3

Batch Production Record in accordance with batch processing record

Manufacturing Organisation Name: ________________________

Description of job: ______________________

Name of part: ___________________________

Batch No.: _____________________________

The following deviations have appeared (continued)

No. process step	Name of processing step	Documented page no.	Short description of deviation
1	Raw material preparation: Operation 1: Descaling Operation 2: Degreasing Operation 3: Wire brushing		1. ____________ 2. ____________ 3. ____________ 4. ____________
2	Sizing of material: Operation 1: Shearing Operation 2: Deburring		1. ____________ 2. ____________ 3. ____________

Estimation and maintenance records

Objectives : At the end of this lesson you shall be able to
- **state the purpose of estimation**
- **explain the details of formats for estimation sheet**
- **explain the details of formats for maintenance log, history sheet of machinery and equipment and checklist for preventive maintenance.**

Estimation is the method of calculating the various quantities and the expenditure to be incurred on a particular job or process.

In case the funds available are less than the estimated cost the work is done in part or by reducing it or specifications are altered,

The following essential details are required for preparing an estimate.

Drawings like plan, elevation and sections of important parts.

Detailed specifications about workmanship & properties of materials, etc.

Standard schedule of rates of the current year.

Estimating is the process of preparing an approximation of quantities which is a value used as input data and it is derived from the best information available.

An estimate that turns out to be incorrect will be an overestimate if the estimate exceeded the actual result, and an underestimate if the estimate fell short of the actual result.

A cost estimate contains approximate cost of a product process or operation. The cost estimate has a single total value and it is inclusive of identifiable component values.

ESTIMATION SHEET - FORMAT-4

Part Name: Base plate Assembly: Shearing machine Assembly No.: MA2WOAO1	Part No.: 1 Material: Fe310.0 Stock size: 305 x 227 x 20	Insert Part Drawing

Operation No.	Operation description	Machine	Estimated time	Rate / piece per hr.	Tools
01	Setting and aligning job on table	Milling	10 min		
02	Mount arbor and cutter	Milling	10 min		
03	Set speed and feed	Milling	2 min		
04	Align cutter in position	Milling	2 min		
05	Mill four sides	Milling	50 min		
06	Mark 45º angle corner	-	8 min		vernier bevel protractor vernier height gauge
07	Set and clamp the job	-	10 min		
08	Mill 45º on opposite sides	-	10 min		
09	Set clamp on other sides	-	20 min		
10	Mill 45º on other sides	-	20 min		
11	Deburr and mark drill position	-	10 min		
12	Set and align for drilling	Drilling	10 min		
13	Mount drill chuck and drill	Drilling	03 min		
14	Set drill rpm	Drilling	02 min		
15	Drill pilot and holes	Drilling	30 min		
16	Counter bore holes	Drilling	15 min		
17	Place job on magnetic chuck on surface grinder	Surface grinder	03 min		
18	Grind the surface as per drawing	Surface grinder	10 min		
19	Deburr sharp edges	-	02 min		Abrasive stick

Maintenance log - Format 5

This format is made with details of maintenance activities performed machinewise,

MAINTENANCE LOG - FORMAT - 5

Organisation Name :

Department:

Section:

Name of the machine:

S.No	Date	Nature of fault	Details of rectification done	Signature of in-charge

History sheet of machinery equipment - Format 6

The document recorded with historical data about the machinery and equipment, contains all details about supplier address, order no., date of receipt, installed and placed, Date of commissioning and machine dimensions, weight, cost, particulars of drive motor, spare parts details, belt specification, lubrication details, major repair/ overhauls done with dates recorded then and there for analysing the functional and frequency of breakdown etc.,

MACHINERY AND EQUIPMENT RECORD FORMAT - 6

Organisation Name :	
Department:	
Section:	
History sheet of machinery & Equipement	
Description of equipment	
Manufacturer's address	
Supplier's address	
Order No. and date	
Date on which received	
Date on which installed and place	
Size : Length x Width x Height	
Weight	
Cost	
Motor particular	Watts/ H.P./ r.p.m: phase: Volts:
Bearings/ spares/ record	
Belt specification	
Lubrication details	
Major repairs and overhauls carried out with dates	

Checklist for preventive maintenance inspection - Format 7

The very essential document required to observe, the functional aspects of each parts, defects and the remedial measures taken is recorded.

This format enables to program the frequency of maintenance schedules so as to minimise frequent breakdown of machinery/equipments.

PREVENTIVE MAINTENANCE RECORD - FORMAT 7

Organisation Name :				
Department :				
Section :				
Name of the Machine :				
Machine Number :				
Model No & Make :				
Check list for machine inspection				
Inspect the following items and tick in the appropriate column and list the remedial measures for the defective items.				
Items to be checked	**Good working**	**Satisfactory**	**Defective**	**Remedial measures**
Level of the machine				
Belt/chain and its tension				
Bearing condition (Look, feel, Listen noise)				
Driving clutch and brake				
Exposed gears				
Working in all the speeds				
Working in all feeds				
Lubrication and its system				
Carriage & its travel				
Cross-slide & its movement				
Compound slide & its travel				
Tailstock's parallel movement				
Electrical controls				
Safety guards				
Inspected by : Signature : Name :				