MANUFACTURING TECHNOLOGY

Volume II

Metal Cutting and Machine Tools

Fourth Edition

About The Author

P N Rao received his BE degree in Mechanical Engineering from the Government Engineering College, Anantapur (Sri Venkateswara University, Tirupati), in 1970, ME degree from Birla Institute of Technology and Science, Pilani, in 1973 and PhD from the Indian Institute of Technology, Delhi, in 1981.

Presently, he is working as a Professor in the University of Northern Iowa, Cedar Falls, USA, in the Department of Technology, since Fall of 2001. Earlier he worked in the Indian Institute of Technology, New Delhi, from 1975 till 1997, the final position being as a Professor in the Mechanical Engineering Department between 1990 and 1997. He has also taught at the MARA University of Technology, Shah Alam, Malaysia, between 1995 and 2001. Prior to joining IIT Delhi, he worked in the Birla Institute of Technology and Science, Pilani, for a period of two years. He has conducted post-doctoral research in UMIST, Manchester, UK (1980–1981), and Loughborough University of Technology, UK (1989).

His active areas of research are Manufacturing Engineering, Design Engineering, Sustainable Manufacturing, CAD/CAM, CIM, Tool design and MEMS, and Nano education. He is the author and co-author of more than 250 papers, published and presented in national and international journals and conferences, respectively.

He also has wide interaction with the industry through the process of consultancy work. He conducts continuing education programmes on various aspects related to modern manufacturing. He has received a number of grants from various bodies to support his research activities.

He has authored three books, all published by McGraw-Hill Education (India).

- *Manufacturing Technology, Volume 1: Foundry, Forming and Welding, 5e*
- *Computer Aided Manufacturing,* 1e (with *Kundra* and *Tewari* as co-authors*)*
- *CAD/CAM: Principles and Applications,* 3e

He has also worked as a co-editor of *Emerging Trends in Manufacturing* (Proceedings of the 12th All India Machine Tool Design and Research Conference, 1986) published by McGraw-Hill Education (India).

Dr P N Rao's biographical information has appeared in *Marquis Who's Who in The World, Marquis Who's Who in Finance and Industry, Marquis Who's Who in Science and Engineering, Marquis Who's Who in America, Marquis Who's Who in American Education*, and *Who's Who in Asia and the Pacific Nations*.

MANUFACTURING TECHNOLOGY

Volume II

Metal Cutting and Machine Tools

Fourth Edition

P N Rao

Professor
Department of Technology
University of Northern Iowa
Cedar Falls
USA

McGraw Hill Education (India) Private Limited
CHENNAI

McGraw Hill Education Offices
Chennai New York St Louis San Francisco Auckland Bogotá Caracas
Kuala Lumpur Lisbon London Madrid Mexico City Milan Montreal
San Juan Santiago Singapore Sydney Tokyo Toronto

McGraw Hill Education (India) Private Limited

Published by McGraw Hill Education (India) Private Limited
444/1, Sri Ekambara Naicker Industrial Estate, Alapakkam, Porur, Chennai - 600 116

Manufacturing Technology—Metal Cutting and Machine Tools, 4e (Volume II)

Print Edition:
ISBN-13: 978-93-5316-052-4
ISBN-10: 93-5316-052-9

E-Book Edition:
ISBN-13: 978-93-5316-053-1
ISBN-10: 93-5316-053-7

1 2 3 4 5 6 7 8 9 D101417 22 21 20 19 18

Printed and bound in India.

Director—Science & Engineering Portfolio: *Vibha Mahajan*

Senior Portfolio Manager—Science & Engineering: *Hemant K Jha*
Associate Portfolio Manager —Science & Engineering: *Vaishali Thapliyal*

Senior Manager—Content Development: *Shalini Jha*
Senior Content Developer: *Ruhhi Ralhan*

Production Head: *Satinder S Baveja*
Copy Editor: *Taranpreet Kaur*
Assistant Manager—Production: *Suhaib Ali*

General Manager—Production: *Rajender P Ghansela*
Manager—Production: *Reji Kumar*

Typeset at The Composers, 260, C.A. Apt., Paschim Vihar, New Delhi 110 063 and printed and bound in India at

Cover Printer:

Cover Designer: The Computer Minds

Cover Image Source: Panksvatouny/Shutterstock

Visit us at: www.mheducation.co.in

Write to us at: info.india@mheducation.com

CIN: U22200TN1970PTC111531

Toll Free Number: 1800 103 5875

DEDICATED TO

My Parents, Suramma and Kondala Rayudu

Contents

Preface to the Fourth Edition

Manufacturing technology is undergoing continuous changes along with the progress in other technological fields. Accordingly, this book, which has been used widely by the education community in India and abroad, needs to be updated to include all those changes. The curricula of mechanical engineering in various Indian universities keep changing to incorporate the changes in technology. As a result, it is necessary to incorporate those requirements in this new edition retaining the main theme of the book.

Based on the responses received from various academicians in the field, I have tried to incorporate the developments in the manufacturing technologies so that the Indian educators can have all the information they need in a single book.

Content Overview

Whenever a process or equipment is described, the practical information, such as specifications, operating parameters and designing for the process, have all been highlighted. Each process is supplemented with simple illustrations, numerical calculations for the design process and a discussion of the results so obtained. A large number of well-labelled illustrations are provided to give the necessary insight into the process and its design.

Chapter 1 introduces the different material-removal processes and machine tools. *Chapter 2* discusses the different aspects of metal cutting, laying emphasis on chips, BUE, cutting-tool materials, tool life and surface finish. *Chapter 3* is on machine tools, their classification and elements. *Chapter 4* describes centre lathe in detail, and *Chapter 5* is on special-purpose lathes. *Chapters 6 and 7* discuss reciprocating machine tools and milling, respectively. *Chapter 8* is on hole-making operations. *Chapter 9* explains the different types of abrasive processes, while *Chapter 10* describes some other machine tools like sawing and broaching machines. *Chapter 11* deals with some unconventional machining processes, *Chapter 12* provides micro machining processes and *Chapter 13* is on machine-tool testing. *Chapter 14* points out some general guidelines for designing machines. *Chapter 15* describes jigs and fixtures, while *Chapter 16* on metrology describes the different kinds of measurements used in machining processes. Finally, *Chapter 17* deals with the numerical control of machine tools.

New Features

All the chapters were thoroughly checked to see that written material is in line with the current practice such that some of the obsolete details are removed. Many of the illustrations are redrawn to simplify them for better understanding of the concepts in tune with the text. In addition to that, the following topics are added or updated in the book:

Chapter 1 – Automation
Chapter 2 – Updated topic on thermal analysis
Chapter 7 – Thread milling
Chapter 9 – Some improvements and revisions

Chapter 15 – Economics of tooling with additional problems
Chapter 16 – Clinometer
Chapter 17 – Expansion of CNC machine tools and CMM

One new chapter is also added, Chapter 12 Micro Manufacturing

A few case studies have been added to provide further discussion on the topics covered, from the practical viewpoint. The instructors can discuss these case studies in more detail in the class so that it will generate interest in the students towards the subject matter. For GATE aspirants, a dedicated section at book end is provided for thorough practice.

With these additions and changes, it is hoped that the current edition will be able to fully satisfy the curricula of most of the universities and thus serve the intended purpose with which I stated in the beginning.

Online Learning Centre

The website of this book can be accessed at *http://www.mhhe.com/rao/mtmcmt4,* which contains the Solution Manual and PowerPoint Lecture Slides for Instructors. Multiple Choice Questions, Chapter-wise GATE Previous Years' Questions, Chapter objectives and Summary for each chapter are given for students.

Acknowledgements

I wish to express my sincere thanks to the University of Northern Iowa, Cedar Falls, USA, where I am currently working, for providing the necessary facilities and environment for undertaking the revision work. Thanks are due to my colleague, Dr R. R. Srikanth, who helped in organizing the case studies of the book. I am thankful to Dr Matthias Pleil, Support Center for Microsystems Education, The University of New Mexico, who introduced me to the fascinating world of MEMS and helped in developing the case study on pressure sensor manufacturing. I also express heartfelt thanks to the editors and production department at McGraw Hill (India) for nudging and guiding me for the timely completion of this work.

I would like to thank the reviewers who have gone through the revised content of the book and have given their valuable suggestions. In particular, I would like to thank:

Pravin Kumar	Delhi Technical University (DTU), Delhi
Manoj Kumar Sinha	National Institute of Technology (NIT), Delhi
Dr. Avinash Kumar Dubey	Motilal Nehru National Institute of Technology (MNNIT), Allahabad
Vratraj K. Joshi	Gujarat Technical University (GTU), Ahmedabad
Chirag K. Balar	Gujarat Technical University (GTU), Ahmedabad
Mukund Dutt Sharma	National Institute of Technology (NIT), Hamirpur

I would welcome further suggestions regarding the coverage in the book and would be happy to incorporate the suggested improvements in future editions to make the book more suitable to the changing curriculum needs of the teaching of manufacturing technology.

Feedback

Despite the utmost care taken, it is not uncommon that some errors will be left in the book uncorrected, and I would request the readers to communicate any such errors and omissions so that I will be able to correct them at the earliest possible opportunity.

P. N. RAO

Preface to the First Edition

Manufacturing technology related to the machine tools is an important subject taught in most of the engineering curricula. The need is more for a balanced approach dealing with the engineering as well as technological aspects that direct towards the actual application of the concepts.

The first volume in the present series was first published in 1986. That book has deviated from the established tradition of the books on workshop technology that deal heavily with the theoretical descriptions by incorporating a variety of actual applications and numerical aspects to explain the concepts of manufacturing processes. The success of the book, which has now gone through the second edition recently, established the methodology adopted.

It was long felt a similar approach needed for teaching machine tools in engineering colleges. The author over the years of teaching the subject in Indian Institute of Technology Delhi felt the need for such a book and that is how the seeds for this new book have been sowed. There is a large variety of books available in the market to deal with the workshop machinery. A majority of these books deal with a very descriptive nature of the machine tools and though voluminous, often are not suitable to completely provide a thorough analytical basis required for manufacturing engineering. Many of these books are more suitable for operators and technicians rather than engineers. Other varieties of books that were written with engineer in mind give very little details and are thus difficult to use unless supplemented by other books.

This book is thus aimed at providing the material in a more balanced approach such that the engineer is in a position to appreciate the details of the process as well as the necessary analytical and design approaches that are essential in developing the manufacturing engineering aspects of these processes. Thus, it is sincerely hoped that the new approach will be appreciated and the various curricula in engineering colleges will take the new approach there by giving the students a better appreciation of the manufacturing processes by reducing the descriptive portions largely.

This book is the outgrowth of course material used by the author for teaching a number of undergraduate courses relating to metal cutting and machine tools. The author practised the concept of providing the practical information, such as specifications, operating parameters and designing for the process, whenever a manufacturing process is discussed. This helps in the understanding the nuances of the process better by the student. As far as possible, effort has been made to supplement the processes with simple illustrations, discuss the analytical aspects and carryout the design process and a discussion of the results so obtained. A large number of well-labelled illustrations are provided to give the necessary insight into the process and its design. The actual photographs of the machine tools are avoided deliberately to keep the size of the book in reasonable limits.

In this book, study of various categories of machine tools available along with their capabilities and applications would be carried. The study in this book would be restricted to some aspects of the machine tools from the standpoint of their application to either mass production or batch production depending upon the requirement. With each of the machine tool description, the method of selection for a given application and the setting process are also developed. These machine tools are so versatile that with a little ingenuity and use of accessories, it is possible to practically manufacture any type of job needed.

The book starts with a long introduction to the metal cutting aspects that form the core of all the subsequent chapters. Details are provided whereby analysis of the mechanics of orthogonal cutting can be understood along with the various practical aspects of its application in real life manufacturing situations.

A separate chapter is provided on the basics of machine tool construction wherein effort has been made to collate the common aspects of all the machine tool details in one place. These details are applicable to many of the machine tools discussed in the later chapters.

In the subsequent chapters, all the major categories of machine tools and processes namely lathes, shaping, milling, hole making, and grinding are covered in greater detail. The unconventional machining processes such as EDM, ECM are covered in a separate chapter. Though these are termed as unconventional, they are quite extensively used in the industries. From that viewpoint, reasonably large coverage of this is attempted.

Some of the other elements that form the integral portions for the study of machine tools namely, machine tool testing, metrology, and design for machining are also covered through separate chapters though the coverage is maintained in a low key to conserve the space. Process planning which is a very important component of the machine tools, which is normally neglected in most of the engineering curricula has been covered in this book with reasonable details. It is the author's conviction that through the understanding of the principles of process planning the student will be better appreciating the application of machine tools.

The book finally ends with a chapter on Numerical Control of machine tools in view of its importance in the industry. Though this subject merits a further coverage as a separate book, sufficient details are provided in this book to help the student getting the basics with which they will be able to use the NC machine tools.

I am grateful to the authorities of the MARA Institute of Technology, Shah Alam, Malaysia, which have provided excellent environment, opportunities and facilities to undertake this task. In particular, I would like to express my sincere thanks to the Faculty of Mechanical Engineering and CADEM Centre who allowed me to liberally use the available facilities and time for this venture. It is a pleasure to express heart-felt gratitude to my family members who have borne long hours of inconvenience during the preparation of the manuscript. I am also thankful to many of my students who have learned this subject from me, for the probing questions and comments through which the subject could be brought to this form.

Readers of the book are requested to provide comments and suggestions related to the coverage and examples used in the book. I would appreciate and welcome any helpful constructive criticism for improvement in future editions.

October 1998 **P. Nageswara Rao**

Visual Walkthrough

Historical Perspective

Provides a brief perspective of historical developments related to the processes discussed in the chapter

1.1 INTRODUCTION TO MATERIAL REMOVAL PROCESSES

The study of metal cutting and machine tools is one of the most fascinating experiences. Machining of materials is adopted basically to get higher surface finish, close tolerances and complex geometric shapes, which are otherwise difficult to obtain.

Of all the manufacturing processes available, metal removable is perhaps the most expensive one. The reason being that from the raw material, quite a substantial amount of material is removed in the form of chips in order to achieve the final shape required. Also, lot of energy is expended in the process of material removal. So the choice of material removal as an option for manufacturing should be considered when no other manufacturing process suits the purpose. However, invariably all the components will be undergoing a material removal operation at one point or other.

A machine tool is defined as one which while holding the cutting tools would be able to remove metal from a work piece in order to generate the requisite job of given size, configuration and finish. It is different from a machine, which essentially is a means of converting the source of power from one form to the other. The machine tools are the mother machines since without them no components could be produced in their finished form. They are very old and the success of the industrial revolution owes to a very great extent to them.

Existence of some form of crude machine tools is recorded as early as 700 B.C. However, the most prominent beginning of the machine tool is the John Wilkinson's horizontal boring machine towards 1775. This invention made the James Watt's steam engine a reality. Henry Maudslay followed this with an engine lathe in 1794. A later machine tool to be invented is the planer by Roberts in 1817. Maudslay combined a lead screw, a cross-slide and change gears in a form, which is almost similar to the current day centre lathe. At the same time another major machine tool to be invented is the milling machine by Eli Whitney in 1818.

The drill press is the next machine tool to be developed around 1840 by John Nasmyth. Stephen Fitch

Objectives

Material removal is the principal operation carried out by majority of the manufacturing industries. This chapter provides a summary of the major factors that need to be considered in metal cutting. After completing this chapter, the reader will be able to

- Understand the basic parameters in the metal cutting operation.
- Appreciate different types of chips formed in metal cutting and their relevance to manufacturing
- Calculate analytically the forces and other parameters associated with the orthogonal cutting
- Understand the importance of shear angle in metal cutting
- Select the cutting tool material for a given application
- Understand the temperatures developed in metal cutting and the variables that control it

Chapter Objectives

Each chapter begins with a clearly defined set of objectives that provide a quick reference to the chapter's key aspects. These help students better anticipate what they will be studying and help instructors measure student's understanding.

Well-labelled Illustrations

Neat illustrations within the chapters provide a complete description of the object in question, labelling the various parts describing the function.

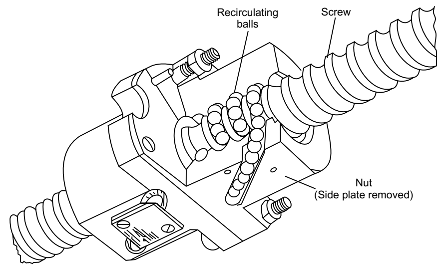


Fig. 3.22 *Recirculating ball screw*

Example 4.4

Estimate the actual machining time required for the component (C40 steel) shown in Fig. 4.42. The available spindle speeds are, 70, 110, 176, 280, 440, 700, 1100, 1760 and 2800. Use a roughing speed of 30 m/min and finish speed of 60 m/min. The feed for roughing is 0.24 mm/rev while that for finishing is 0.10 mm/rev. The maximum depth of cut for roughing is 2 mm. Finish allowance may be taken as 0.75 mm. Blank to be used for machining is 50 mm in diameter.

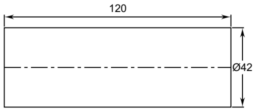


Fig. 4.42 *Machining time example 1*

Solution Stock to be removed $= \frac{50 - 42}{2} = 4$ mm

Finish allowance = 0.75 mm

Roughing:

Roughing stock available = 4 – 0.75 = 3.25 mm

Since maximum depth of cut to be taken is 2 mm, there are 2 roughing passes.

Given cutting speed, V = 30 m/min

Solved Examples

Throughout each chapter, various solved examples are given that help readers understand and apply the concept learnt in the chapter.

Summary

A detailed chapter-end summary is provided for a quick review of the important concepts discussed in the chapter.

SUMMARY

There are a large number of hole making operations depending upon the geometry of the hole to be made.

- Drilling is the most common hole making operation.
- Twist drill geometry with two cutting lips arranged helically around the central web acts as the main cutting tool for drilling.
- A large variety of drill types are used such as step drills, spade drills, and shell core drills.
- Drill press, radial drilling machine and multiple spindle drilling machines are some of the varieties of drilling machines used.
- Drilling time can be estimated based on the hole length and the drilling process parameters.
- Drilling force and power can be estimated using the empirical relations.
- Deep hole drilling requires special precautions to take care of the removal of large volume of chips.
- Reaming is a finishing operation done to provide close-toleranced holes.
- Boring is an operation used to enlarge a hole using a single point cutting tool.
- Tapping is used to make inside threads where a finished hole is already present.
- Tapping is an operation using slow cutting speeds.
- Counter boring, spot facing and counter sinking are the other operations that are used to finish the holes for specific requirements.

Questions

6.1 Explain with a neat sketch the operation and need for a clapper box in a mechanical shaper.
6.2 Give a schematic sketch of a shaper labelling important parts and their functions.
6.3 What are the applications of shaping machines in a typical machine shop?
6.4 Explain the following principal parts of a mechanical shaper.
  (a) Ram
  (b) Tool post
  (c) Quick return motion
6.5 Give the details of different types of shapers and their applications.
6.6 Give the various details that need to be specified for a shaping machine.
6.7 Describe the operation of the quick return motion in a mechanical shaper.
6.8 How are the tools held in a shaper?
6.9 Describe the methods of holding the work pieces in shapers. Give simple sketches of the same.
6.10 Give a neat sketch of the mechanical feed drive of a horizontal shaper and explain its function.

Revision Questions

A set of review questions has been carefully constructed to help students review their understanding of the concepts.

Practice Problems

Practice problems are given in each chapter to provide hands-on practice to students in solving problems related to real-life situations.

Problems

7.1 A 20 mm × 150 mm diameter HSS side and face milling cutter is to be used to cut a groove into a piece of brass with one cut. The groove is 20 mm wide, 4 mm deep and 250 mm long. Calculate the total machining time. Justify the assumptions made if any. [0.558 minutes]

7.2 In a slab milling operation, the milling cutter has 20 teeth and is 100 mm in diameter. The rotational speed of the cutter is 5 RPS. If the flat surface to be generated is 200 mm by 50 mm and feed per tooth is 0.013 mm/rev., calculate the machining time required for 100 pieces. The depth of cut may be taken as 6 mm. Specify any assumptions made. [1.15 minutes]

7.3 A surface 115 mm wide and 250 mm long is to be rough milled with a depth of cut of 6 mm by a 16-tooth cemented carbide face mill 150 mm in diameter. The work material is medium hard cast iron (220 – 260 BHN). Estimate the cutting time and justify the process parameters used. [4.6296 minutes]

7.4 A flat surface of 250 × 350 mm is to be produced on a horizontal axis milling machine. A slab mill of 100 mm diameter and 150 mm width is to be used for the purpose. Calculate the machining time assuming that entire stock can be removed in one depth of 1 mm. Make only the requisite assumptions. [6.25 minutes]

Multiple Choice Questions

8.1 Back taper is provided on a drill to
(a) Increase the strength of the drill
(b) Provide longitudinal clearance
(c) Decrease the cutting thrust
(d) Decrease the cost of the drill

8.2 Axial rake angle of a drill is
(a) The angle between the leading edge of the land and the axis of the drill
(b) The angle between the face and the line parallel to the drill axis
(c) The angle formed by the portion of the flank adjacent to the land and a plane at right angles to the drill axis
(d) None of the above

8.3 Helix angle of a drill is
(a) The angle between the leading edge of the land and the axis of the drill
(b) The angle between the face and the line parallel to the drill axis
(c) The angle formed by the portion of the flank adjacent to the land and a plane at right angles to the drill axis
(d) None of the above

8.4 The lip clearance angle of a drill is
(a) The angle between the leading edge of the land and the axis of the drill
(b) The angle between the face and the line parallel to the drill axis

Multiple Choice Questions

These help the students to have a quick recap of the important terms and concepts learnt in the chapter.

Case Studies

Provides a practical view point of the subject to increase interest and understanding.

Case Study

TURBINE BLADE MACHINING

Engine components for airplanes which need to maintain low weight, high temperature resistance and increased thermal efficiency require that these be manufactured with high nickel and titanium alloys. The commonly used materials are Ti-6Al-4V and Inconel 718. The geometry of turbine blade is complex and these blades are arranged on the rotor disk with specialized geometry that facilitates the flow of gases. The gap between the blades is relatively small and has complex geometry, the machining of which requires a very careful planning. Being hard materials these alloys can be machined using conventional milling, Electro Discharge Machining (EDM) and Electro Chemical Machining (ECM) processes.

In this case study, these alloys were machined using conventional milling, EDM and ECM processes in order to compare the material removal rates and economics. The conventional milling is done with trochoidal milling which is a method of machining used to create a slot wider than the milling cutter diameter. This is accomplished by moving the cutter through a series of circular cuts known as a trochoidal tool path. This provides a low radial depth of cut and a high axial depth of cut and achieves good material removal rate. In the EDM setup, initial roughing was done using the regular EDM process and then Wire EDM was used. The table below shows the variation of MRR among the processes.

Manufacturing Process Used		Material Removal Rate of Ti–6Al–4V (mm^3/min)	Material Removal Rate of Inconel 718 (mm^3/min)
Conventional Milling		6035	3401
EDM	EDM (Roughing)	220	500
	Wire EDM (Finishing)	3090	2388
ECM		4838	4163

GATE Previous Years' Questions

Chapter 2

2.1 In an orthogonal cutting process the tool used has rake angle of zero degree. The measured cutting force and thrust force are 500 N and 250 N, respectively. The coefficient of friction between the tool and the chip is ________.
(GATE-2016-ME-SET-1, 1-Mark)

2.2 The tool life equation for HSS tool is $VT^{0.14} f^{0.7} d^{0.4}$ = Constant. The tool life (T) of 30 min is obtained using the following cutting conditions:
V = 45 m/min, f = 0.35 mm, d = 2.0 mm

2.5 In a single point turning operation with cemented carbide tool and steel work piece, it is found that the Taylor's exponent is 0.25. If the cutting speed is reduced by 50%, then the tool life changes by ________ times.
(GATE-2016-ME-SET-3, 2-Marks)

2.6 Under certain cutting conditions, doubling the cutting speed reduces the tool life to $(1/16)^{th}$ of the original. Taylor's tool life index (n) for this tool-work piece combination will be ________.
(GATE-2015-ME-SET-1, 1-Mark)

GATE Section

Chapter-wise previous years GATE questions given for the students to have hands on practice.

CHAPTER 1

Introduction

1.1 INTRODUCTION TO MATERIAL REMOVAL PROCESSES

The study of metal cutting and machine tools is one of the most fascinating experiences. Machining of materials is adopted basically to get higher surface finish, close tolerances and complex geometric shapes, which are otherwise difficult to obtain.

Of all the manufacturing processes available, metal removable is perhaps the most expensive one. The reason being that from the raw material, quite a substantial amount of material is removed in the form of chips in order to achieve the final shape required. Also, lot of energy is expended in the process of material removal. So the choice of material removal as an option for manufacturing should be considered when no other manufacturing process suits the purpose. However, invariably all the components will be undergoing a material removal operation at one point or other.

A machine tool is defined as one which while holding the cutting tools would be able to remove metal from a work piece in order to generate the requisite job of given size, configuration and finish. It is different from a machine, which essentially is a means of converting the source of power from one form to the other. The machine tools are the mother machines since without them no components could be produced in their finished form. They are very old and the success of the industrial revolution owes to a very great extent to them.

Existence of some form of crude machine tools is recorded as early as 700 B.C. However, the most prominent beginning of the machine tool is the John Wilkinson's horizontal boring machine towards 1775. This invention made the James Watt's steam engine a reality. Henry Maudslay followed this with an engine lathe in 1794. A later machine tool to be invented is the planer by Roberts in 1817. Maudslay combined a lead screw, a cross-slide and change gears in a form, which is almost similar to the current day centre lathe. At the same time another major machine tool to be invented is the milling machine by Eli Whitney in 1818.

The drill press is the next machine tool to be developed around 1840 by John Nasmyth. Stephen Fitch designed the first turret lathes in 1845. It carried eight tools on a horizontally mounted turret for producing screws. A completely automatic turret lathe was invented by Christopher Spencer in 1869. This is the first form of the automatic lathe utilising cams for feeding the tool in and out of the work piece thereby automating most of the machining tasks. He is also credited with the development of a multiple-spindle lathe. Finally the surface grinder was developed around 1880. This probably completes the development of almost all basic machine tools.

Over the intervening period, the basic machine tools have been refined by adding various attachments as well as automating the movements. Also the invention of various precision measurement techniques helped in improving the accuracy and productivity of the machine tools.

Manufacturing technology is going through major technological changes because of the revolutionary changes being brought in by the developments in the microelectronics. The availability of computers and then microprocessors has completely changed the machine tool scenario by bringing in the flexibility which was not possible through the conventional mechanisms. The development of Numerical Control in 1952 has, for the first time, brought the kind of flexibility to the metal cutting operation, that at present, a majority of the manufacturing processes are making use of these principles in some form or the other. This allows for the Just In Time (JIT) manufacture leading to zero inventories, zero setup times and single component batches without losing any advantages of mass manufacture.

1.2 VARIETY OF MACHINE TOOLS

Casting processes and the metal working processes are the primary manufacturing processes where the metal is first shaped into an intermediate shape which is normally brought to its final form with metal cutting process. Assembling of parts into workable equipment often requires the mating surfaces to be complementary to each other in terms of form, dimensions and surface finish. The only way this can be achieved is through the use of material removal processes. The broad classification of the material removal processes is shown in Fig. 1.1. They can be classified into the traditional processes, which rely on the difference in the hardness of the cutting tool to that of the work piece to remove the material. These are the first processes that have been in use for a greater part of the time. These are further divided based on the type of work piece geometry and the method employed for generating that geometry.

The non-traditional processes have been mostly invented in the 20th century to take care of the space age materials that are too hard and difficult to be machined by the conventional processes. These processes rely on other methods for removal rather than the hardness difference between the work piece and the cutting tool material.

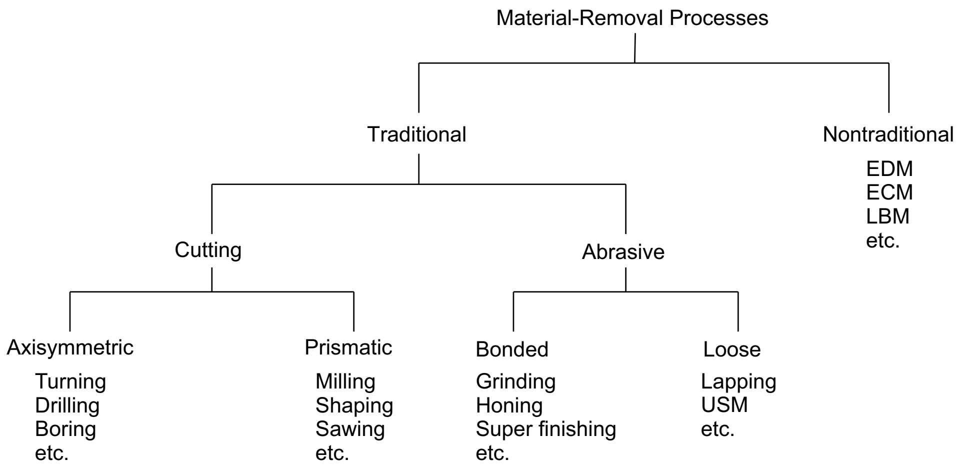


Fig. 1.1 *Classification of the material removal processes*

The large varieties of material removal processing machines that are available as such are:

- Turning machines (Lathes)
- Drilling machines
- Boring machines
- Milling machines
- Grinding machines
- Shaping and Planing machines

- Gear cutting machines
- Sawing machines
- Unconventional machining machines

Besides these main varieties of machine tools, a number of specialised variations of machine tools are available depending upon the requirement. They are:

- Automats
- Copy turning machines
- Form relieving lathes
- Reaming
- Copy milling machines
- Plano milling machines
- Centre less grinding machines
- Broaching machines

Table 1.1 lists the status of machine tool industry in India, which is considered as one yardstick towards the industrialisation.

Table 1.1 Status of machine tool industry in india*

	Production	Import	Export	Consumption	Share of Production (Less Exp.) to Total Consumption
	(in ₹ million)				(in %)
1987	2,454	1,118	592	2,980	62.5
1990	4,132	3,404	809	6,727	49.4
1995	7,198	5,976	445	12,729	53.1
1996	8,080	11,003	249	18,834	41.6
1997	7,963	7,221	321	14,863	51.4
1998	6,712	8,405	606	14,511	42.1
1999	5,970	4,727	382	10,315	54.2
2000	6,307	4,258	330	10,232	58.4
2001	5,282	3,103	373	8,012	61.3
2002	5,175	4,332	508	8,999	51.9
2003	6,782	6768	463	13,087	49.8
2004	10,122	16,001	491	25,632	37.6
2004–2005	10,950	18,208	526	28,632	36.4

(*Contd.*)

*Source: Indian Machine Tool Manufacturers Association, IMTMA annual reports – (https://www.imtma.in/index.php?page=1&subid=278 Retrieved on Sep 4, 2017)

2005–2006	13,498	28,986	502	41,905	31.0
2006–2007	17,307	46,557	731	63,044	26.3
2007–2008	19,020	59,919	1,467	77,472	22.7
2008–2009	14,244	62,706	894	76,055	17.6
2009–2010	16,562	48,422	810	64,173	24.5
2010–2011	36,240	67,032	1360	101,912	34.2
2011–2012	42,990	76,450	1800	117,640	35.0
2012–2013	38,850	75,980	2140	112,690	32.6
2013–2014	34,810	46,720	2460	79,070	40.9
2014–2015	42,300	53,180	2800	92,680	42.6
2015–2016	47,270	59,450	2960	103,760	42.7

As mentioned earlier, material removal processes are very expensive and hence, should be resorted to only when it is absolutely needed. Table 1.2 gives a relative comparison of the material removal processes with the other manufacturing processes as a qualitative comparison.

1.3 AUTOMATION

Beginning of the twentieth century, large volume manufacture has taken off in a big scale and in the process reduced the manufacturing costs. This made it possible that people have access to a large range of products and the economies have improved. Automation played a big part in this process. Merriam Webster dictionary defines automation as “the technique of making an apparatus, a process, or a system operate automatically”. The aftermath of automation is that quality and consistency of manufacturing have improved, while the need for human labour has decreased. The reduction in the manufactured goods is due to the productivity improvement achieved due to automation.

In the beginning, most of the manufacturing automation is hardware oriented with all mechanical devices such that the necessary activation is done through levers and actuators. Later, this has been improved by the addition of hydraulic and pneumatic devices to provide more flexibility. This type of industrial automation is specifically used to produce a single component that is normally termed as hard automation. When any small design is to be incorporated, the automation components need to be modified to suit the new design. This is time consuming, depending upon the type of modification involved. However, with the developments in the microelectronics, sensors and computers, the automation has changed drastically in what is now termed as soft automation, where the components remain fixed while their functionality is controlled through the program which can be modified to suit any design changes as required. The computer numerical control machines and robots are part of that industrial automation which leads to much improved productivity and quality.

TABLE 1.2 A typical comparison of different manufacturing processes

Manufacturing Process	Typical Application	Size Range, kg	Tolerance Surface Finish	Typical Production Volume	Relative Tooling Cost	Disadvantage to Use
Sand casting	All metals	Unlimited	±0.030 mm/mm 3.2 μm	Unlimited	Low	Casting must be machined
Die casting	Zinc and Aluminium alloys	Up to 7 kg	±0.0015 mm/mm 1.6 μm	Very high	High	Porosity
Drop forging	All materials	Unlimited		Very high	Medium	Slow cycle time
Hot Extrusion	All metals	Unlimited		Very high	Low	Low production speeds
Gas Metal Arc Welding	All metals	12 mm thick		High	High	Equipment cost and portability
Sheet metal blanking	All materials		±0.08 mm	Very high	Low	Leaves burr on the part
Turning	All materials	Unlimited	±0.050 mm 2.0 mm	Very high	Medium	Relatively slow Material wastage
Milling	All materials	Unlimited	±0.050 mm 2.0 μm	High	Medium	Relatively slow Material wastage
Grinding	All materials	Unlimited	±0.025 mm 0.4 μm	High	Medium	Expensive finishing operation
Electric discharge machining	Electrically conductive materials		±0.003 mm 0.1 μm	Low	Low	Dielectric fluid must be filtered

CHAPTER 2

Metal Cutting

Objectives

Material removal is the principal operation carried out by majority of the manufacturing industries. This chapter provides a summary of the major factors that need to be considered in metal cutting. After completing this chapter, the reader will be able to

- Understand the basic parameters in the metal cutting operation.
- Appreciate different types of chips formed in metal cutting and their relevance to manufacturing
- Calculate analytically the forces and other parameters associated with the orthogonal cutting
- Understand the importance of shear angle in metal cutting
- Select the cutting tool material for a given application
- Understand the temperatures developed in metal cutting and the variables that control it
- Understand tool wear and tool life and the variables that control them
- Determine how the surface finish varies with the process parameters
- Know the various cutting fluids and their application methods
- Empirically determine the cutting forces
- Optimise the machining process to satisfy the required conditions

2.1 INTRODUCTION

The importance of machining processes can be emphasized by the fact that every product we use in day-to-day life has used this process either directly or indirectly.

(a) In US, more than $ 100 billion were spent annually on the machining and related operations.
(b) Typically a large majority (above 80%) of all the machine tools used in the manufacturing industry are metal cutting in nature.
(c) An estimate in 1957 showed that about 10 to 15% of all the metal produced in US. is converted into chips.

These facts show the importance of metal cutting in general manufacturing. It is therefore important to understand the metal cutting process in order to make the best use of it. Before the end of 19th century, some amount of work was done by people like Tresca, Thime, Mallock, etc. But it was mostly scattered work. The monumental work done by F.W. Taylor in the last stages of 19th century and the beginning of 20th has been, in fact, the starting point for a rational thinking on the metal cutting process. He is mostly interested in empirical research and 30 years of the results of his experimental work was published in Transactions of ASME in 1907, running to about 300 pages.

Afterwards, a number of investigations have been carried out in understanding the metal cutting process and this knowledge has been used to help improve the manufacturing operations involving metal cutting. In this chapter we will study in brief the various understandings available from the metal cutting research work.

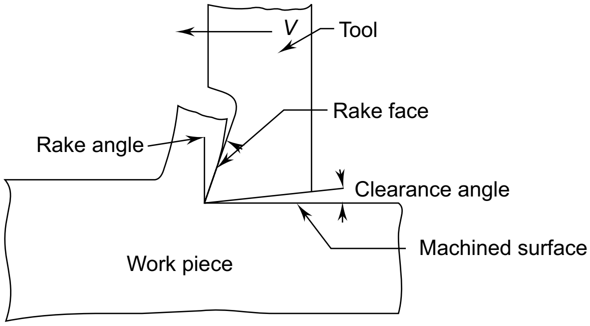


Fig. 2.1 *The general characteristics of a metal cutting tool*

A typical cutting tool in a simplified form is shown in Fig. 2.1 removing metal. The important features to be observed are:

Rake angle: It is the angle between the face of the tool called the rake face and the normal to the machining direction. This angle specifies the ease with which a metal is cut. Higher the rake angle better is the cutting and less is the cutting force. Increasing the rake angle reduces the metal backup available at the tool rake face. This reduces the strength of the tool tip as well as the heat dissipation through the tool. Thus, there is a maximum limit to the rake angle and is generally of the order of 15° for high speed steel tools cutting mild steel. It is possible to have rake angle as zero or negative as shown in Fig. 2.2. These are generally used in the case of highly brittle tool materials such as carbides or diamond for giving extra strength to the tool tip.

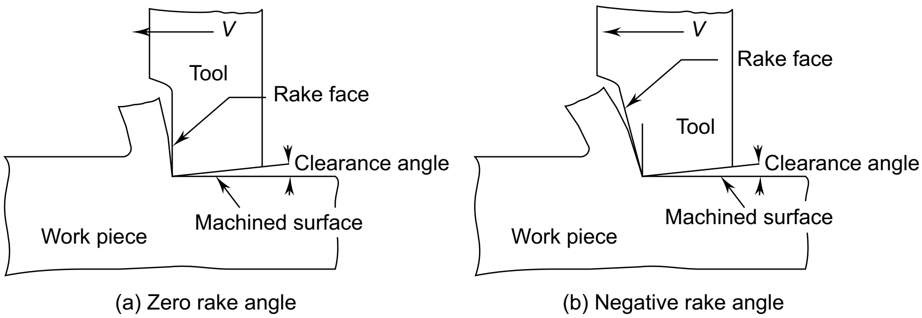


Fig. 2.2 *Tool cutting at different rake angles*

Clearance angle: This is the angle between the machined surface and the underside of the tool called the flank face. The clearance angle is provided such that the tool will not rub or spoil the machined surface, but at the same time will increase the cutting forces. A very large clearance angle reduces the strength of the tool tip, hence normally an angle of the order of 5 to 6° is generally used.

The conditions which have a predominant influence on the metal cutting are: work material, cutting tool material, cutting tool geometry, cutting speed, feed rate, depth of cut and cutting fluid used.

The cutting speed, V, is the speed with which the cutting tool moves through the work material. This is generally expressed in metres per second (ms^{-1}).

Feed rate, *f*, may be defined as the small relative movement per cycle (per revolution or per stroke) of the cutting tool in a direction usually normal to the cutting speed direction.

Depth of cut, *d*, is the normal distance between the unmachined surface and the machined surface.

2.2 CHIP FORMATION

Metal cutting process is one of the most complex processes. Fig 2.3 shows the basic material removal operation schematically. The metal in front of the tool rake face gets immediately compressed first elastically and then plastically. This zone is traditionally called the shear zone, in view of the fact the material in the final form would be removed by shear from the parent metal. The actual separation of the metal starts from the cutting tool tip as yielding or fracture, depending upon the cutting conditions. Then the deformed metal (called chip) flows over the tool (rake) face. If the friction between the tool rake face and the underside of the chip (deformed material) is considerable, then chip gets further deformed, which is termed as secondary deformation. The chip after sliding over the tool rake face would be lifted away from the tool, and the resultant curvature of the chip is termed as chip curl.

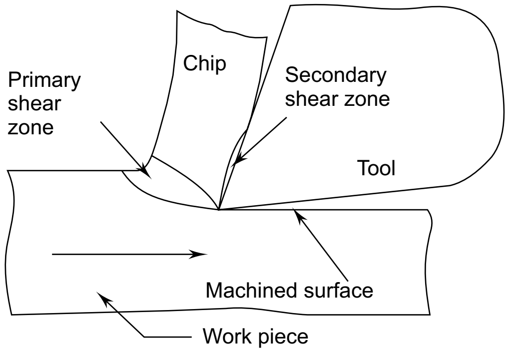


Fig. 2.3 *The possible deformations in metal cutting*

Plastic deformation can be caused by yielding, in which case strained layers of material would get displaced over other layers along the slip-planes which coincide with the direction of maximum shear stress.

Piispanen presented an interesting mechanism to account for the deformation process taking place at the cutting edge. He considered the undeformed metal as a stack of cards which would slide over one another as the wedge shaped tools moves under these cards as shown in Fig. 2.4. Though this idea is an over simplified one, it would account for a number of features that are found in practice. A practical example is when paraffin is cut; block wise slip is clearly evident.

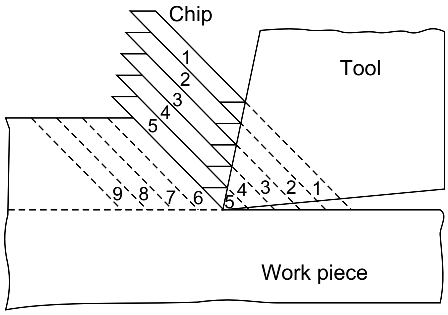


Fig. 2.4 *Piispanen's model of metal cutting*

The chip used in actual manufacturing practices is variable both in size and shape. Study of the chip is one of the most important things in metal cutting. As will be seen later, the mechanics of metal cutting are greatly dependent on the shape and size of the chips produced.

The chip formation in metal cutting could be broadly categorised into three types:

(i) Discontinuous chip
(ii) Continuous chip, and
(iii) Continuous chip with BUE

2.2.1 Discontinuous Chip

When brittle materials like cast iron are cut, the deformed material gets fractured very easily and thus the chip produced is in the form of discontinuous segments as shown in Fig. 2.5. In this type the deformed material instead of flowing continuously gets ruptured periodically. Discontinuous chips are easier from the chip

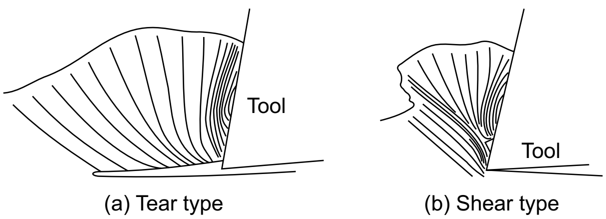


Fig. 2.5 *Possible discontinuous chip formations*

disposal view point. However, the cutting force becomes unstable with the variation coinciding with the fracturing cycle as shown in Fig. 2.6. Also they generally provide better surface finish. However, in case of ductile materials they cause poor surface finish and low tool life. Higher depths of cut (large chip thickness), low cutting speeds and small rake angles are likely to produce discontinuous chips.

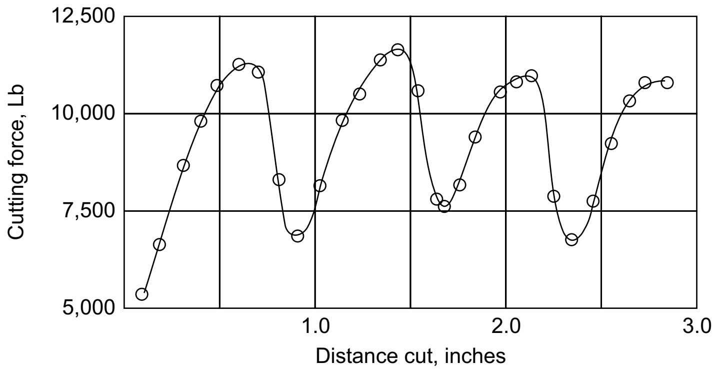


Fig. 2.6 *The variation of cutting force in discontinuous chip formation (Medium carbon steel, Rake angle = 30°, Cutting speed = 1.65 ft/min, Depth of cut = 0.41 in, Feed rate = 0.125 in/rev)*

2.2.2 Continuous Chip

Continuous chips are normally produced when machining steel or ductile metals at high cutting speeds. The continuous chip, which is like a ribbon flows (Fig. 2.7) along the rake face. Continuous chip is possible because of the ductility of metal (steel at high temperature generated due to cutting) flows along the shear plane instead of rupture. Thus on a continuous chip you do not see any notches. It can be assumed that each layer of metal flows along the slip plane till it is stopped by work hardening. Each of these layers gets welded to the previous ones because of the high temperature, thus forming a continuous chip. Some ideal conditions that promote continuous chips in metal cutting are: sharp cutting edge, small chip thickness (fine feed), large rake angle, high cutting speed, ductile work materials and less friction between chip tool interfaces through efficient lubrication.

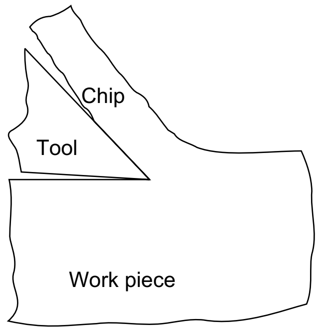


Fig. 2.7 *Continuous chip formation*

This is the most desirable form of chip since the surface finish obtained is good and cutting is smooth. It also helps in achieving higher tool life and lower power consumption. However, because of the large coils of chips, the chip disposal is a problem. To help in this direction various forms of chip breakers have been developed which are in the form of a step or groove in the tool rake face. The chip breakers allow the chips to be broken into small pieces so that they can be easily disposed of.

2.2.3 Continuous Chip with BUE

When the friction between tool and chip is high while machining ductile materials, some particles of chip adhere to the tool rake face near the tool tip. When such sizeable material piles up on the rake face, it acts as a cutting edge in place of the actual cutting edge as shown in Fig 2.8. This is termed as built up edge (BUE). By virtue of work hardening, BUE is harder than the parent work material. As the size of BUE grows , it becomes

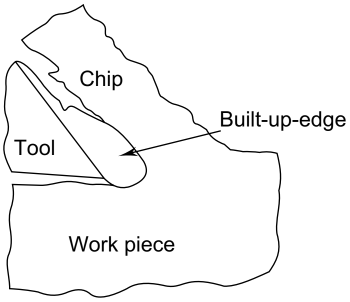


Fig. 2.8 *Close-up view of BUE*

unstable and parts of it get removed while cutting. The removed portions of BUE partly adhere to the chip underside and partly to the machined surface as shown in Fig. 2.9. This causes the finished surface to be rough. However, since the cutting is carried by the BUE and not the actual tool tip, the life of the cutting tool increases while cutting with BUE. In this way BUE is not harmful during rough machining.

The conditions that normally induce the formation of BUE are low cutting speed, high feed and low rake angle. One of the prerequisites for the formation of BUE is the work hardenability of the work piece material. Higher the work hardenability, rougher is the machined surface produced.

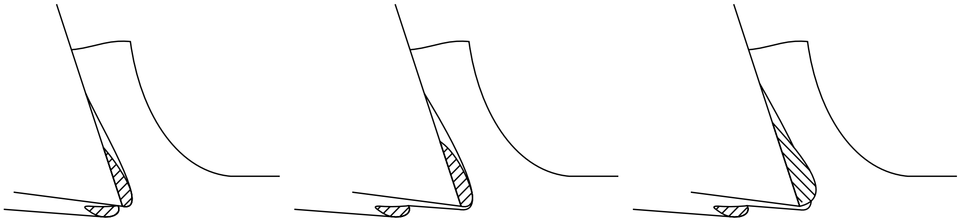


Fig. 2.9 *BUE cycle*

Though the above is a theoretical classification of chips, in actual practice many other types (which could appear in the border areas of these three types) would also be present.

2.2.4 BUE

The formation of a BUE on the tool is brought about by the high normal loads on the tool rake face leading to adhesion between the chip and the tool. This adhesion may be so severe that instead of the chip sliding over the tool face, considerable plastic flow and eventual rupture occurs within the chip. Further layers build up, leading to a large nose of the material projecting from the cutting edge.

The adhesion at the chip tool interface is very strong and different from the conventional adhesion characteristics of the material pair concerned. The conditions of machining are more extreme compared to most other deformation processes.

(i) It is a plastic flow process with exceptionally large strains. There is high compressive stress acting on the plastic zone and this prevents rupture until the strain is well above the rupture value, in say, a tensile test.
(ii) The deformation is localised to an extremely small plastic zone. The strain rate is unusually high.
(iii) The chip material rubbing over the tool face is freshly formed from the body of the work material and is in a chemically clean condition. This makes it more chemically active than the usual oxidised surfaces encountered in most sliding situations, a feature which increases the tendency for adhesion and so gives a higher friction force.

2.3 SHEAR ZONE

There are basically two schools of thought to analyse the metal removal process. One school of thought is that the deformation zone is very thin and planar as shown in Fig. 2.10(a). The other school thinks that the actual deformation zone is a thick one with a fan shape as shown in Fig. 2.10(b).

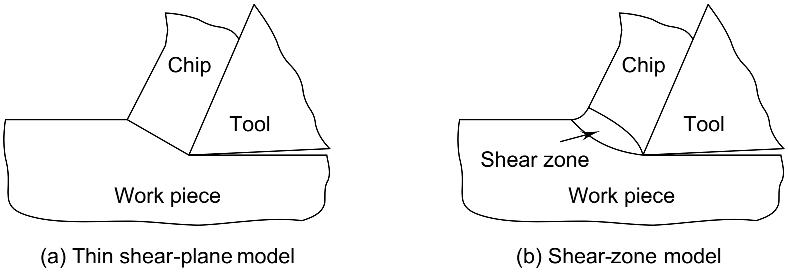


Fig. 2.10 *(a) Thin shear plane model, (b) Thick shear zone model*

Though the first model Fig. 2.10(a) is convenient from the stand point of analysis, physically it is impossible to create . The reason being that for transition from undeformed material to a deformed one along a thin plane, the acceleration across the plane has to be infinity, for the velocity to change instantaneously from cutting speed V_i to V_c. Similarly the stress gradient across the shear plane has to be very large to be practical.

In the second model Fig. 2.10(b) by marking the shear zone over a region, the transitions in velocities and the shear stresses could be realistically accounted for.

The angle made by the shear plane with the cutting speed vector, ϕ is a very important parameter in metal cutting. Higher the shear angle better is the cutting performance. From a view of the Fig. 2.10(a), it can be observed that higher rake angles give rise to higher shear angles.

2.4 ORTHOGONAL CUTTING

Investigators in the metal cutting field have attempted to develop an analysis of the cutting process which gives a clear understanding of the mechanisms involved and which enables the prediction of the important cutting parameters, without the need for testing. But none of the models developed so far could be fully substantiated and definitely stated to be the correct solution. It is worth examining them because they will be qualitatively explaining the phenomenon.

A general purpose metal cutting operation such as turning or milling is three-dimensional and is normally termed as oblique cutting. The obliquity comes from the angle between the cutting speed vector and the cutting edge of the tool as shown in Fig. 2.11(b). Though this is the most practical, it is far more difficult to analyse because of its 3 dimensions.

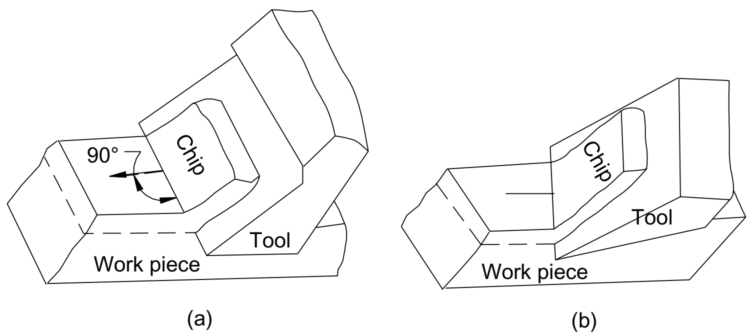


Fig. 2.11 *(a) Orthogonal cutting, (b) Oblique cutting*

To simplify the matters, researchers often resort to orthogonal cutting wherein the cutting edge is perpendicular to the cutting velocity as shown in Fig. 2.11(a). Though normal turning is oblique, a special case of turning a pipe from the side as shown in Fig. 2.12 is orthogonal. Similarly parting operation in turning is orthogonal. Since this type of cutting reduces the complexity, most of our discussion in this chapter would be based on orthogonal cutting only.

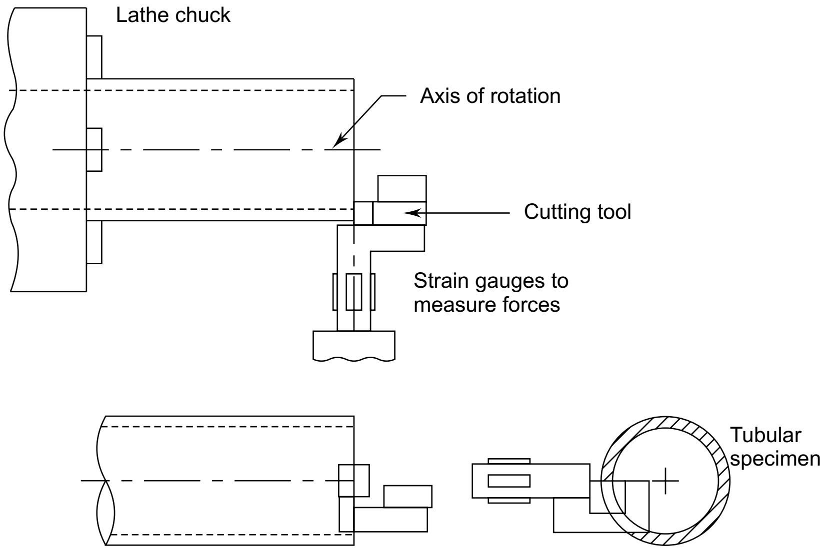


Fig. 2.12 *Realisation of orthogonal cutting in practice while turning a tube from the end*

2.4.1 Mechanics of Orthogonal Metal Cutting

As has been mentioned previously, there are two schools of thoughts regarding the plastic deformation that is taking place at the cutting zone. It seems that the thin zone model is likely to be the most useful since it is simple for analysis purpose.

The current analysis is based on Merchant's thin shear plane model considering the minimum energy principle. This model would be applicable at very high cutting speeds, which are generally practised in production.

Assumptions:

(i) The tool is perfectly sharp and has no contact along the clearance face.
(ii) The surface where shear is occurring is a plane.
(iii) The cutting edge is a straight line extending perpendicular to the direction of motion and generates a plane surface as the work moves past it.
(iv) The chip does not flow to either side or no side spread.
(v) Uncut chip thickness is constant.
(vi) Width of the tool is greater than the width of the work.
(vii) A continuous chip is produced without any BUE.
(viii) Work moves with a uniform velocity.
(ix) The stresses on the shear plane are uniformly distributed.

The resultant forces can be conveniently resolved in the direction of the shear plane, along the primary tool motion and along the rake face. In order to achieve the requisite deformation, the tool would be exerting a cutting force F_H along the primary cutting motion direction as shown in Fig. 2.13. Similarly, other force components are:

F_V – Force perpendicular to the primary tool motion (thrust force)

F_s – Force along the shear plane

N_s – Force normal to the shear plane

F – Frictional force along the rake face

N – Normal force perpendicular to the rake face

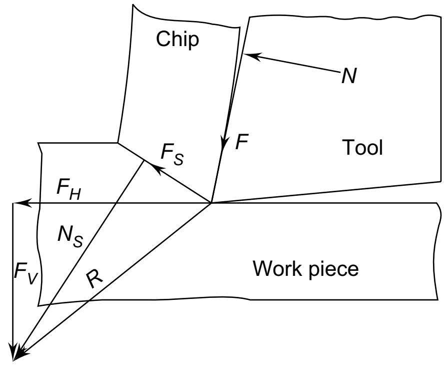


Fig. 2.13 *Various forces acting in orthogonal cutting*

When the chip is isolated as a free body as shown in Fig. 2.14, we need consider only two forces, the force between the tool face and the chip (R) and the force between the work piece and the chip along the shear plane (R'). For equilibrium

$$R = R'$$

It is possible to represent all these forces be acting at the tool point in place of their actual point of action. By doing so it is possible to construct a cutting force circle as shown in Fig. 2.15 which is often called Merchant's circle who demonstrated it for the first time. It is then a simple exercise to derive the various relationships among the forces.

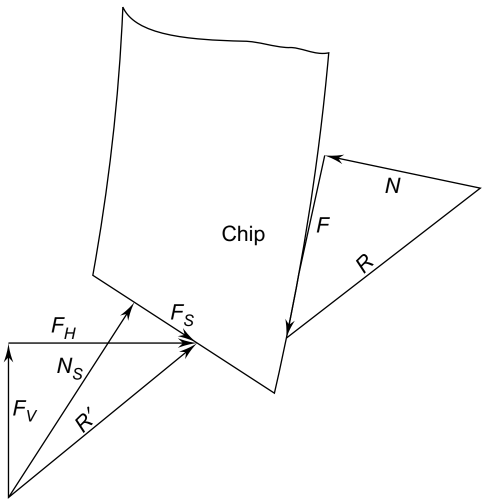


Fig. 2.14 *Forces acting on an isolated chip in metal cutting*

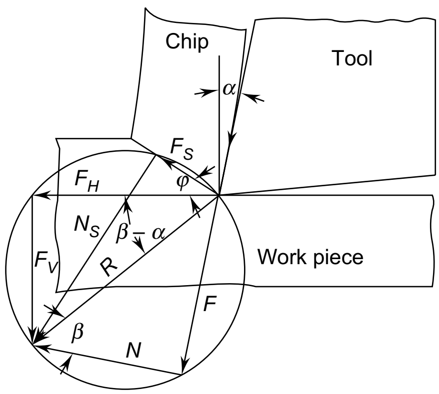


Fig. 2.15 *Merchant's cutting force circle in orthogonal cutting*

We will make some construction into the Fig. 2.15 to get the relationships between the various forces as in Fig. 2.16 and 2.17.

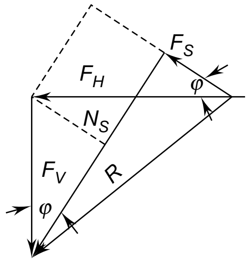


Fig. 2.16 *Part of Merchant's cutting force diagram*

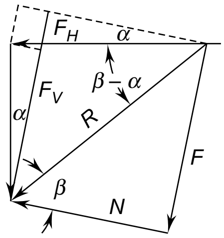


Fig. 2.17 *Part of Merchant's cutting force diagram*

From Fig. 2.15 and 2.16, we can write

$$F_S = F_H \cos\varphi - F_V \sin\varphi \quad (1)$$

$$N_S = F_V \cos\varphi + F_H \sin\varphi \quad (2)$$

$$= F_S \tan(\varphi + \beta - \alpha) \quad (3)$$

From Fig. 2.15 and 2.17, we can write

$$F = F_H \sin\alpha + F_V \cos\alpha \quad (4)$$

$$N = F_H \cos\alpha - F_V \sin\alpha \quad (5)$$

If μ is the coefficient of friction along the rake face, then

$$\mu = \tan\beta = \frac{F}{N} = \frac{F_V + F_H \tan\alpha}{F_H - F_V \tan\alpha} \quad (6)$$

where β is the friction angle, and

ϕ is the shear angle

This friction is not similar to the usual sliding case since F and N are not uniformly distributed over the sliding area. This aspect is discussed later.

Now, the area of shear plane, A_s is given by

$$A_s = \frac{bt}{\sin\phi} \quad (7)$$

The shear force is given by

$$F_s = \tau A_S = \frac{\tau bt}{\sin\varphi} \quad (8)$$

where τ is the mean shear stress in the shear plane.

b is the width of cut

and t is the uncut chip thickness

$$\sigma = \frac{N_S}{A_S} \quad \text{or} \quad N_S = \frac{\sigma bt}{\sin\varphi} \quad (9)$$

Where σ is the mean normal stress in the shear plane.

We can show that by resolving

$$F_H = F_s \cos\varphi + N_s \sin\varphi \tag{10}$$

$$F_V = N_s \cos\varphi - F_s \sin\varphi \tag{11}$$

Substituting Eq. (3) in (10), we get

$$F_H = F_s [\cos\varphi + \sin\varphi \tan(\varphi + \beta - \alpha)] \tag{12}$$

Similarly,

$$F_V = F_s [\cos\varphi \tan(\varphi + \beta - \alpha) - \sin\varphi] \tag{13}$$

Rearranging, we get

$$F_H = F_s \left[\frac{\cos(\alpha - \beta)}{\cos(\varphi + \beta - \alpha)}\right] \tag{14}$$

$$F_H = \frac{\tau b t \cos(\beta - \alpha)}{\sin(\varphi)\cos(\varphi + \beta - \alpha)} \tag{15}$$

$$F_V = \frac{\tau b t \sin(\beta - \alpha)}{\sin(\varphi)\cos(\varphi + \beta - \alpha)} \tag{16}$$

Merchant considered that τ would have the value of the yield shear stress for the work material and that μ would have the usual value for any dry sliding friction. To determine ϕ he assumed that the minimum energy principle applied in metal cutting, so that the deformation process adjusted itself to a minimum energy condition, or

$$\frac{dF_H}{d\varphi} = \frac{\tau b t \cos(\beta - \alpha)\cos(2\varphi + \beta - \alpha)}{\sin^2\varphi \cos^2(\varphi + \beta - \alpha)} = 0 \tag{17}$$

or $$\cos(2\varphi + \beta - \alpha) = 0 \tag{18}$$

or $$2\varphi + \beta - \alpha = \frac{\pi}{2} \tag{19}$$

$$\varphi = \frac{\pi}{4} - \frac{1}{2}(\beta - \alpha) \tag{20}$$

Substituting back, we can show that

$$F_H = 2\,\tau\, b\, t \cot\varphi \tag{21}$$

$$F_v = \tau\, b\, t\,(\cot^2\varphi - 1) \tag{22}$$

The above deductions were obtained assuming two things that are not supported by experimental evidence. Firstly the minimum energy principle, though appealing is not supported by evidence. Next, it assumes that β and α are constant w.r.t. ϕ. But later studies on metal cutting have shown that at least β is dependent on ϕ. Of course, this is concerning only the steady state operation. But under dynamic conditions α also varies considerably. Experiments were run and it was found that this equation is not valid. We will see more about this shear angle relationship later.

To determine experimentally the shear angle we have to stop the cutting process and study the zone with the help of a microscope or a photograph. Alternatively we can also derive a relationship from the geometry of chip formation as shown in Fig. 2.18.

From Fig. 2.18

$$t = AB \sin \varphi \quad (23)$$

$$t_c = AB \cos (\varphi - \alpha) \quad (24)$$

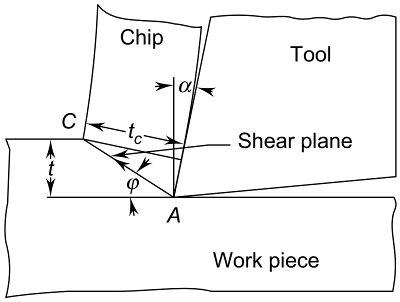


Fig. 2.18 *Orthogonal cutting with thin shear plane*

The chip thickness ratio, r, which is also termed as cutting ratio, would be

$$r = \frac{t}{t_c} = \frac{\sin \varphi}{\cos (\varphi - \alpha)} = \frac{1}{\cot \varphi \cos \alpha + \sin \alpha} \quad (25)$$

$$\cot \varphi \cos \alpha = \frac{1 - r \sin \alpha}{r} \quad (26)$$

$$\tan \varphi = \frac{r \cos \alpha}{1 - r \sin \alpha} \quad (27)$$

Experimentally the chip thickness ratio, r could be determined by measuring the average thickness of the chips produced under given conditions of feed and speed. From this it is possible to evaluate the shear angle using the above equation. However, direct measurement of chip thickness is difficult, because of the roughness on the outside of the chip. For this purpose an indirect measurement is followed wherein the length of a chip, l_c equivalent to a known length of uncut chip is measured. Then considering the fact that the depth being same, the average chip thickness, t_c would be given by

$$t_c = \frac{t\,l}{l_c} \quad (28)$$

where l = length of uncut chip

To get an exact size of uncut chip length, l, we may introduce a small saw cut parallel to the axis on the work piece so that uncut chip size is

$$l = \pi D \quad (29)$$

where D is the diameter of the work piece.

The chip velocity, V_c is the velocity of the chip relative to the tool and directed along the tool face. The shear velocity, V_s is the velocity of the chip relative to the work piece and directed along the shear plane. These two velocities along with the cutting velocity, V would form a closed triangle as shown in Fig. 2.19. From this we can get

$$\frac{V}{\sin \{90° - (\varphi - \alpha)\}} = \frac{V_S}{\sin (90° - \alpha)} = \frac{V_C}{\sin \varphi} \quad (30)$$

$$V_C = \frac{V \sin \varphi}{\cos (\varphi - \alpha)} \quad (31)$$

$$V_S = \frac{V \cos \alpha}{\cos (\varphi - \alpha)} \quad (32)$$

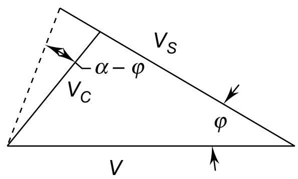


Fig. 2.19 *Velocity relationships in orthogonal cutting*

To evaluate the shear strains, we take the help of Piispanen's model as shown in Fig. 2.20.

Shear strain γ is given by

$$\gamma = \frac{\Delta S}{\Delta Y} = \frac{AB}{CD} = \frac{AD}{CD} + \frac{DB}{CD} = \tan \varphi + \cot (\varphi - \alpha) \quad (33)$$

or

$$\gamma = \frac{\cos \alpha}{\sin \varphi \cos (\varphi - \alpha)} = \frac{V_S}{V \sin \varphi} \quad (34)$$

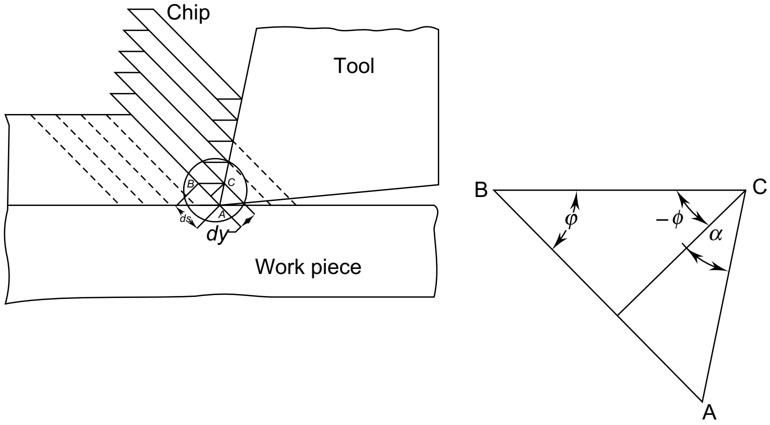


Fig. 2.20 *Strain and strain rate in orthogonal cutting*

The strain rate is given by

$$\dot{\gamma} = \frac{\Delta S}{\Delta Y \; \Delta t} = \frac{V_S}{\Delta Y} = \frac{\cos \alpha}{\cos(\varphi - \alpha)} \frac{V}{\Delta Y} \tag{35}$$

where ΔY is the thickness of the deformation zone and t is the time to achieve the final value of strain. ΔY can be considered as the mean value of the spacing of successive slip planes, which is of the order of 2.5 microns.

Most of the energy consumed in metal cutting is utilised in the plastic deformation. The total work done, W is given by

$$W = F_H V \tag{36}$$

The work done in shear W_s is

$$W_s = F_s V_s \tag{37}$$

Similarly the work done in friction W_f is

$$W_f = F V_c \tag{38}$$

Thus

$$W = F_H V = F_s V_s + F V_c \tag{39}$$

To get a better picture of the efficiency of the metal cutting operation it is necessary to have a new parameter, which does not depend upon the cutting process parameters. The specific cutting energy, u_s is such a parameter which can be obtained by dividing the total work done with the material removal rate. The material removal rate is:

$$MRR = V b t \tag{40}$$

$$u_S = \frac{F_H V}{MRR} = \frac{\tau \cos(\beta - \alpha)}{\sin(\varphi) \cos(\varphi + \beta - \alpha)} \tag{41}$$

To understand the significance of the various equations derived a number of numerical examples were solved below.

Example 2.1

A bar of 75 mm diameter is reduced to 73 mm by a cutting tool while cutting orthogonally. If the mean length of the cut chip is 73.5 mm, find the cutting ratio. If the rake angle is 15°, what is the shear angle?

Solution Length of uncut chip, $l = \dfrac{\pi(75 + 73)}{2} = 232.4779$ mm

Cutting ratio, $r = \dfrac{t_c}{t} = \dfrac{73.9}{232.4779} = 0.3179$

Shear angle, $\varphi = \tan^{-1}\left[\dfrac{r\cos\alpha}{1 - r\sin\alpha}\right] = \tan^{-1}\left[\dfrac{0.3179\cos 15}{1 - 0.3179\sin 15}\right]$

Shear angle, $\varphi = \tan^{-1}(0.3346) = 19°$

Example 2.2

In an orthogonal cutting test with a tool of rake angle 10°, the following observations were made:

Chip thickness ratio = 0.3
Horizontal component of the cutting force = 1290 N
Vertical component of the cutting force = 1650 N

From the Merchant's theory, calculate the various components of the cutting forces and the coefficient of friction at the chip tool interface.

Solution Given $r = 0.3$ $\alpha = 10°$

The shear plane angle, ϕ is

$$\tan\phi = \frac{r\cos\alpha}{1 - r\sin\alpha} = \frac{0.3\cos 10}{1 - 0.3\sin 10} = 0.311679$$

$$\text{shear angle, } \phi = \tan^{-1}(0.311679) = 17.31°$$

Given $F_V = 1650$, $F_H = 1290$

The friction force along rake face is

$$F = F_H\sin\alpha + F_V\cos\alpha = 1290\sin 10 + 1650\cos 10 = 1848.94 \text{ N}$$

the normal force on the rake face is

$$N = F_H\cos\alpha - F_V\sin\alpha = 1290\cos 10 - 1650\sin 10 = 983.88 \text{ N}$$

The coefficient of friction, μ, at the chip tool interface is given by

$$\mu = \frac{F}{N} = \frac{1848.94}{983.88} = 1.8792$$

The friction angle, β is given by

$$\beta = \tan^{-1}\mu = \tan^{-1}(1.8792) = 62°$$

The resultant cutting force, R is given by

$$R = \sqrt{1650^2 + 1290^2} = 2094.42 \text{ N}$$

The shear force along the shear plane is

$$F_s = F_H \cos\varphi - F_V \sin\varphi = 1290 \cos 17.31 - 1650 \sin 17.31 = 740.63 \text{ N}$$

The normal force on the shear plane is

$$N_s = F_V \cos\varphi + F_H \sin\varphi = 1650 \cos 17.31 + 1290 \sin 17.31 = 1959.10 \text{ N}$$

The area of the shear plane is given by

$$A_s = \frac{bt}{\sin\varphi} = \frac{6 \times 0.10}{\sin 17.31} = 2.0165 \text{ mm}^2$$

To verify the shear angle from the relation suggested by Merchant:

$$\varphi = \frac{\pi}{4} - \frac{1}{2}(\beta - \alpha) = \frac{\pi}{4} - \frac{(62-10)}{2} = 19°$$

It can be seen that the actual value of the shear angle is 17.31, whereas the value calculated from the shear angle relation of Merchant's is 19. The error is 9.76%.

Example 2.3

The orthogonal cutting of steel with 10° rake tool with a depth of cut of 2 mm and feed rate of 0.20 mm/rev. The cutting speed is 200 m/min. The chip thickness ratio is 0.31. The vertical cutting force is 1200 N and the horizontal cutting force is 650 N. Calculate from the merchant's theory, the various work done in metal cutting and shear stress.

Solution Given $r = 0.31$, $\alpha = 10°$

Shear plane angle, ϕ is

$$\tan\phi = \frac{0.31 \cos 10}{1 - 0.31 \sin 10} = 0.32266$$

$$\text{Shear angle, } \phi = \tan^{-1}(0.32266) = 17.88°$$

Given $F_V = 1200$, $F_H = 650$

The shear force along the shear plane is

$$F_s = 650 \cos 17.88 - 1200 \sin 17.88 = 250.18 \text{ N}$$

The normal force on the shear plane is

$$N_s = 1200 \cos 17.88 + 650 \sin 17.88 = 1341.61 \text{ N}$$

The area of the shear plane is given by

$$A_s = \frac{bt}{\sin\phi} = \frac{2 \times 0.20}{\sin 17.88} = 1.3028 \text{ mm}^2$$

Friction force along rake face is

$$F = 650 \sin 10 + 1200 \cos 10 = 1294.64 \text{ N}$$

Normal force on the rake face is

$$N = 650 \cos 10 - 1200 \sin 10 = 431.75 \text{ N}$$

The coefficient of friction, μ, at the chip tool interface is given by

$$\mu = \frac{F}{N} = \frac{1294.64}{431.75} = 2.9986$$

The friction angle, β is given by

$$\beta = \tan^{-1} \mu = \tan^{-1} (2.9986) = 71.56°$$

To verify the validity of the shear angle relationship suggested by Merchant:

$$\varphi = \frac{\pi}{4} - \frac{1}{2}(\beta - \alpha) = \frac{\pi}{4} - \frac{(71.56 - 10)}{2} = 14.22°$$

It can be seen that the actual value of the shear angle obtained from measured values is 17.88, whereas the value calculated from the shear angle relation of Merchant's is 14.22, the resulting error being 20.5%.

The shear velocity V_s is given by

$$V_s = \frac{V \cos \alpha}{\cos (\varphi - \alpha)} = \frac{200 \cos 10}{\cos (17.88 - 10)} = 198.84 \text{ m/min}$$

The chip velocity V_c is given by

$$V_C = \frac{V \sin \varphi}{\cos (\varphi - \alpha)} = \frac{200 \sin 17.88}{\cos (17.88 - 10)} = 61.99 \text{ m/min}$$

Shear strain γ is given by

$$\gamma = \frac{V_S}{V \sin \varphi} = \frac{198.84}{200 \sin 17.88} = 3.2382$$

The strain rate is given by

$$\dot{\gamma} = \frac{V_S}{\Delta Y} = \frac{\cos \alpha}{\cos (\varphi - \alpha)} \frac{V}{\Delta Y} = \frac{\cos 10 \times 200000}{\cos (17.88 - 10) \times 0.0025} = 79.5357$$

taking $\Delta Y = 2.5$ microns.

The shear work done W_s is

$$W_s = F_s V_s = 250.1767 \times 198.84 = 49745.14 \text{ Nm/min}$$

The work done in friction W_f is

$$W_f = F \times V_C = 1294.64 \times 61.99 = 80254.77 \text{ Nm/min}$$

The total work done is

$$W = F_H \times V = 200 \times 650 = 130000 \text{ Nm/min}$$

The shear work proportion out of the total work done is $\frac{49745.14}{130000} = 38.27\%$

Friction work proportion in total work done is $\frac{80254.77}{130000} = 61.73\%$

Friction

It is found that μ determined in this way is exceptionally high in value and it varies with tool geometry and other cutting conditions. But this is contrary to the sliding friction studies.

Under a microscope the actual contact of two sliding surfaces through the high spots (asperities). In the case of normal contacting surfaces as shown Fig. 2.21, the real area of contact is different from the apparent area of contact. Real area changes first by the elastic deformation, and when load increases by plastic deformation. Thus

$$A_r = \frac{N}{p} \tag{42}$$

where p = mean yield stress of the asperities, and

N = applied load.

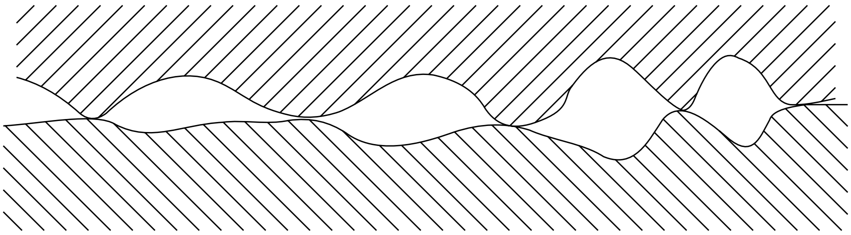


Fig. 2.21 *Micro view of asperities in contact*

Under the influence of normal and tangential load, it has been shown that very high temperatures are developed at the contacting asperities and that metallic bonding of the contacting high spots can occur. Thus sliding of one surface relative to the other must be accompanied by shearing of the welded asperities.

When plastic deformation takes place at the contacting surfaces, then the friction mechanism is different because of the fact that real area of contact approaches that of apparent area of contact. Under these conditions the friction force is independent of normal force.

Another anomaly noted with the experimental data of metal cutting is that the friction coefficient increases with an increase in the rake angle as shown in Table 2.1 below.

It is normally expected that with an increase in the rake angle, the metal cutting forces decrease and should normally be associated with a decrease in the friction. However, in actual practice the friction coefficient increases as shown in Table 2.1.

Table 2.1 **Variation of coefficient of friction with rake angle in orthogonal cutting**

Rake Angle, $\alpha°$	Coefficient of Friction, μ
–20	0.58
–5	0.725
0	0.78
5	0.90
20	1.19

This is happening because the influence of the rake angle is not the same on the different components of the cutting force. The normal force on the rake face decreases to a great extent compared to the friction face as shown in Table 2.2. Thus, though there is an overall decrease in the forces, the coefficient of friction increases. That is why Kronenberg calls this friction coefficient as 'apparent coefficient of friction'.

Table 2.2

Rake Angle, $\alpha°$	Friction Force F	Normal Force N	Coefficient of Friction, μ	% Decrease	
	N	N		F	N
16	3025	4518	0.67		
30	2524	2938	0.86	16.87	35
45	2470	2034	1.21	18.62	55

In metal cutting, we have a sliding situation under conditions of high normal load and with a metal surface, which is chemically clean, having been recently exposed from the body of the parent metal. The cleanness of the metal surfaces can explain the high value of μ and high normal load can explain the departure from the usual laws of friction.

Thus the friction along the rake face of a cutting tool can be considered as partially sticking and partially sliding as shown in Fig. 2.22. In the sticking zone, the shear stress would be constant, approaching the work material's yield stress, while in the sliding zone, the normal Coulomb's laws of friction would hold good.

Another aspect to be noted is the inclusion of the rubbing force component at the clearance face in the measured forces. This component can be obtained by plotting the measured cutting force against the depth of cut and extrapolating back to zero depth. Even after this deduction, which is not contributing to the shear, we get a higher value of μ which can only be explained by the distribution of stresses on the rake face as shown in Fig. 2.22.

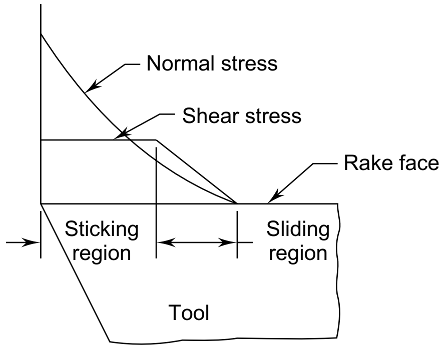


Fig. 2.22 *Stress distribution expected along the rake face*

2.5 SHEAR ANGLE AND ITS RELEVANCE

The importance of shear angle has already been discussed previously. There were a number of attempts to derive a simple enough relationship for shear angle which could be predicted from the cutting process parameters itself, without going in for extensive experimentation. Merchant, in 1941, derived the following formula:

$$\varphi = \frac{\pi}{4} + \frac{1}{2}(\alpha - \beta) \tag{43}$$

However, the experimental observations made (Fig. 2.23) were in variance with this formula over its entire range of operation. He assumed further that shear stress, τ_s, on the normal plane would be affected by the normal compressive stress τ_n in a linear fashion as follows:

$$\tau_s = \tau_o + K \times \tau_n \tag{44}$$

The modified equation therefore is

$$\varphi = \frac{C}{2} + \frac{1}{2}(\alpha - \beta) \tag{45}$$

Where C represents the factor dependent on the plastic properties of the material and is given by

$$C = \cot^{-1} K \tag{46}$$

Merchant called this factor 'machinability constant', but it is not a constant. An average value of C for steel is 75°.

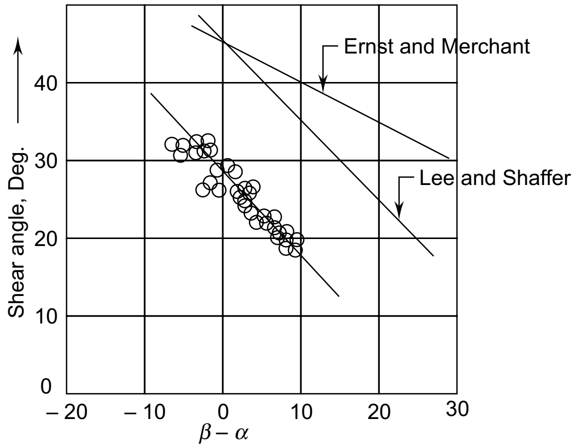


Fig. 2.23 *Comparison of experimental results with the shear angle relationships*

Lee and Shaffer have used the mathematical theory of plasticity based on the behaviour of a rigid plastic material to produce a solution of the orthogonal machining problem. The solution involves the construction of a slip line field pattern using a shear plane model (Fig. 2.24). They considered that there must be a stress field within the chip to transmit the cutting forces from the shear plane to the tool face. They represented this by a slip line field in which no deformation occurs, even when it was stressed up to the yield point. This shows the Mohr's circle for the stresses at the boundaries of the stressed zone, which results in the equation:

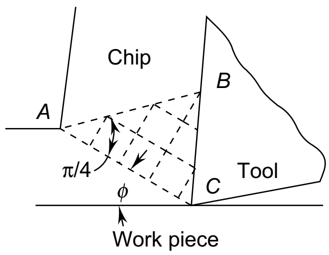


Fig. 2.24 *Shear plane model of Lee and Shaffer*

$$\varphi = \frac{\pi}{4} + (\alpha - \beta) \tag{47}$$

So far an impressive list of equations have been suggested, some of which were based purely on theory while others on theory with some empirical constants with differing assumptions, but none of them agreed with the experimental results. The main reason could be that the various assumptions made are not strictly valid which may be the cause for the deviation. The assumption that the cutting tool is perfectly sharp, so that the tool nose force is negligible could be a rough approximation to the actual condition, particularly at small values of undeformed chip thickness. Further, the representation of the primary deformation zone by a shear plane may be unrealistic.

2.6 CUTTING TOOL GEOMETRY

In the case of practical cutting tools, specification of the cutting tool geometry is a very important aspect that has to be carefully considered. The two considerations that need to be considered (Armarego and Brown) are:

- the ease with which the tool geometry can be maintained through the grinding and inspection process, and
- the mechanics of the process and its relationship with the tool geometry.

To this extent a number of tool geometries have been specified by the different agencies, and some of them are discussed below. Before looking at the geometries a look at the designation of some of the features of the tools are shown in Fig 2.25.

2.6.1 ASA System

The American Standard Association (ASA) designation of the tool nomenclature is purely geometrical in nature and is not related to the mechanics of the process. ASA system specifies the tool geometry with two intersecting orthogonal planes; one parallel to and the other perpendicular to the axis of the cutting tool. Both of these orthogonal planes are perpendicular to the base of the tool Fig. 2.26(a).

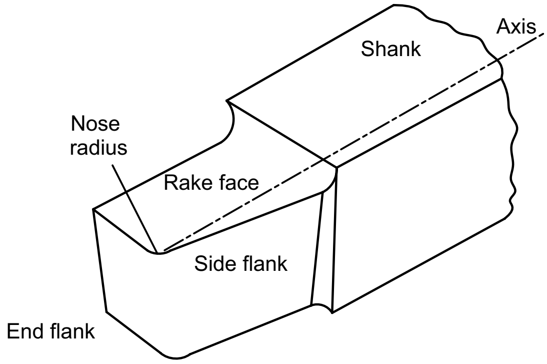


Fig. 2.25 *A typical single point cutting tool with various features and their nomenclature.*

- P_B – Base plane; plane perpendicular to the velocity vector
- P_L – Machine longitudinal plane; plane perpendicular to P_B and taken in the direction of assumed longitudinal feed
- P_T – Machine Transverse plane; plane perpendicular to both P_B and P_L (This plane is taken in the direction of assumed cross feed)

The main advantage of the system is the ability to set the angles on a tool and cutter grinder for grinding the tool angles. However a tool cannot be ground accurately to the back rake and side rake without using equations or curves.

The various tool angles as specified in ASA system are:

- Back rake angle (α_b) – angle of inclination of the rake surface from the base plane (P_B) and measured on Machine Transverse plane, P_T.
- Side rake angle (α_s) – angle of inclination of the rake surface from the base plane (P_B) and measured on Machine Longitudinal Plane, P_L.
- End relief angle (φ_e) – angle of inclination of the principal flank from the machined surface and measured on P_T plane.
- Side relief angle (φ_s) – angle of inclination of the principal flank from the machined surface and measured on P_L plane.
- End cutting edge angle (C_e) – angle between the end cutting edge (its projection on P_B) from P_L and measured on P_T.
- Side cutting edge angle (C_s) – angle between the principal cutting edge (its projection on P_B) and P_T and measured on P_B

Tool signature: $\alpha_b - \alpha_s - \varphi_e - \varphi_s - C_e - C_s - r$
where r is the nose radius.

2.6.2 Orthogonal Rake System (ORS)

As explained earlier the problem with the ASA system is that it utilizes the rectangular coordinate system but not the actual cutting planes of the cutting tool. In the ORS system (also called as the old ISO system) the actual cutting plane is utilized and all the angles are therefore measured in the planes corresponding to the cutting tools as shown in Fig. 2.27. A base plane is defined as the plane where the base of the cutting tool is present. The cutting plane is defined as the plane normal to the base plane and passing through the principal

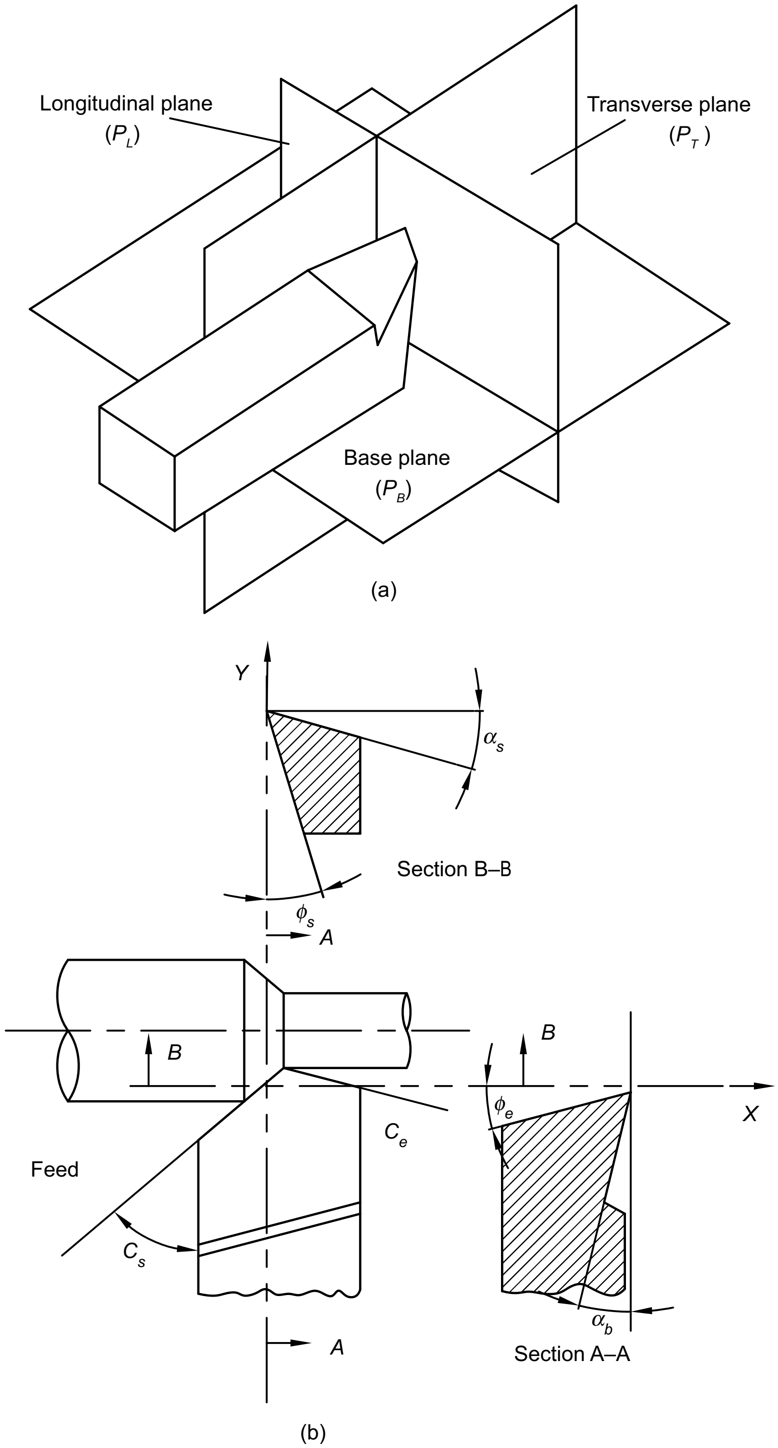


Fig. 2.26 *The tool specification system as per the ASA.*

cutting edge (side cutting edge angle C_s in ASA). A third plane called the orthogonal plane as shown in Fig. 2.27 is perpendicular to these two planes.

- Base plane - P_B – perpendicular to the cutting velocity vector
- Cutting plane - P_C – plane perpendicular to P_B and taken along the principal cutting edge
- Orthogonal plane - P_O – plane perpendicular to both P_B and P_C and the axes

The drawback of this system is that it needs some calculations to obtain setting angles on tool grinding fixture.

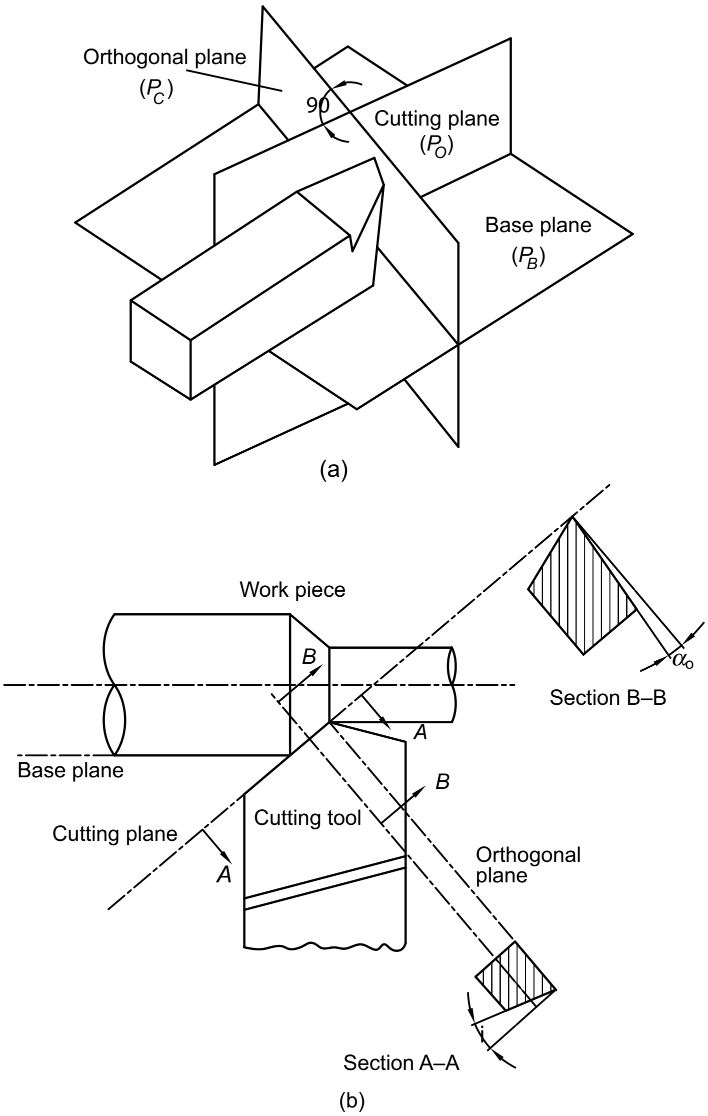


Fig. 2.27 *The tool specification system as per the ORS.*

The various angles identified and defined are as follows:

- Inclination angle (i) – angle between P_C from the direction of assumed longitudinal feed (P_L) and measured on P_C
- Orthogonal rake angle (α_o) – angle of inclination of the rake surface from base plane, P_B and measured on the orthogonal plane, P_O
- Orthogonal clearance of the principal flank angle (φ_o) – angle of inclination of the principal flank from P_C and measured on P_O
- Auxiliary orthogonal clearance angle (φ_o') – angle of inclination of the auxiliary flank from auxiliary cutting plane, P_C' and measured on auxiliary orthogonal plane, P_O'
- Principal cutting edge angle (C) – angle between P_C and the direction of assumed longitudinal feed or P_L and measured on P_B
- Auxiliary cutting angle (C') – angle between P_C' and P_L and measured on P_B
- Nose radius (r) – radius of curvature of tool tip
- Tool signature i, α_o, φ_o, φ_o', C, C_o', r (mm)

Equations for conversion between ASA and ORS system are as follows:

$$\tan \alpha_b = \sin C_s \tan \alpha_o + \cos C_s \tan i \tag{48}$$

$$\tan \alpha_s = \cos C_s \tan \alpha_o - \sin C_s \tan i \tag{49}$$

$$\tan i = -\tan \alpha_s \sin C_s + \tan \alpha_b \cos C_s \tag{50}$$

$$\tan \alpha_o = \tan \alpha_s \cos C_s + \tan \alpha_b \sin C_s \tag{51}$$

2.6.3 ISO System or Normal Rake System (NRS)

Two major problems with the ORS are:

- The true geometry of the cutting tool is not revealed when the cutting edges are inclined from the base plane.
- Also tool grinding in ORS requires additional calculations for setting of angles in the cutter grinder.

To overcome these limitations the NRS system, also called the new ISO system, is useful. Compared to the ORS system the rake angle is visualized in the normal plane in place of the orthogonal plane in ORS. In this system the base and the cutting planes are same as ORS, but the third plane is perpendicular to the cutting edge of the tool as shown in Fig. 2.28. It is called normal plane. In this system, the side rake angle is defined as the angle between the base plane of the tool and the rake face of the tool measured in a plane normal to the side cutting edge.

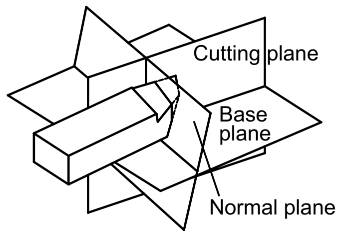


Fig. 2.28 *The tool specification system as per the NRS.*

2.6.4 Significance of Various Tool Angles

Rake Angles (back and side)

These can be positive, zero, or negative. As seen earlier in orthogonal cutting mechanics, shear angle is directly affected by the rake angle. Larger rake angles are beneficial for machining efficiency giving rise to lower cutting force and power. However, increasing to a very high value decreases the strength of the tool tip. Small rake angles are used for cutting hard materials while large angles are used for cutting soft and ductile materials. An exception is brass where, to prevent digging of tool in work, it is machined with small rake angles.

Side Cutting-Edge Angle

Side Cutting-Edge Angle (SCEA) prevents the sudden engagement of the entire depth of cut when the tool enters the work material. As a result it affects the resulting tool life, and surface finish. It can vary from 0° to 90°. When it is zero, the entire cutting edge will engage at the same time with the work piece. It is somewhat similar to orthogonal cutting since the radial component of the cutting force becomes almost negligible. It is used to produce square shoulders. It is particularly desirable while machining castings and forgings that normally have hard and scaly skins.

When the SCEA is increased the entry of the tool is smooth to start the cut since the depth of cut will gradually increase until the entire cutting edge is in contact with the work piece. The chip produced is thinner and wider, with increased SCEA. This helps in distributing the produced heat over a larger cutting edge. However with larger SCEA, the radial component of the cutting force increases thereby promoting the possibility of chatter. The complimentary angle (90° – C_S) of SCEA is called plan approach angle.

End Cutting-Edge Angle

The purpose of the End Cutting-Edge Angle (ECEA) is to relieve the trailing end of the cutting edge to prevent rubbing the machined surface. To that extent only a small angle is sufficient for this purpose. It is not desirable to have a large ECEA as it takes away material that supports the cutting edge and hinders the conduction of heat away from the point. In most cases, values of 8° to 15° have been found satisfactory for boring and turning tools.

Relief Angles (Side and End)

The function of the relief angles is to prevent the rubbing of the flank of the tool with the machined surface. In general, turning relief angles ranging from 5° to 15° are used. Similar to ECEA small relief angles give strength to the cutting edge when machining hard and strong materials. Increased values of relief angles allow the tool to penetrate and cut the work piece material more efficiently, thereby reducing the cutting forces. However too large relief angles weaken the cutting edge and are not desirable.

2.7 DYNAMOMETERS

As noted earlier, the cutting force is an important indication for the performance of the machining operation. The equipment that is used to measure the cutting force is called dynamometer. The measurement of cutting force can be accomplished by a number of approaches. They are:

(a) By measuring the deflection of a body that is directly influenced by the acting cutting forces,
(b) By measuring the strain induced in the body that is directly influenced by the acting cutting forces, and
(c) By measuring the pressure exerted on a medium that is directly influenced by the acting cutting forces.

Any typical setup for measuring the cutting force consists of the following two components:

- A medium that experiences the cutting force, and
- A sensor that measures and converts it to a measurable quantity such as a strain gage

The sensor output is then normally amplified and measured using any of the traditional measuring equipment that is used in the laboratory.

While designing a dynamometer, it is important to consider a number of design requirements. They are:

- Sensitivity—it should provide sufficient sensitivity for different ranges of measurements.
- Rigidity—It should have high stiffness and rigidity.
- Cross sensitivity—In the case of multi-channel dynamometer it should separate the individual force components without their having any cross-sensitivity
- Stability—The dynamometer measuring component should not be affected by cutting fluids, humidity or temperature.
- Solid construction—It is preferable that the dynamometer be manufactured from a single block of material, rather than an assembly of a number of parts in the force transmission, as they may cause loss of accuracy.
- Ease of calibration—The dynamometer should be calibrated with simple procedures, and the calibration curve should remain linear in the operating range.

2.7.1 Sensing Technologies Used in Dynamometers

As described earlier there are a number of methods by which the cutting force can be measured. For each type of quantity measured, a number of measurement tools are available. They are:

(a) Measuring deflection (relatively simple)
- Potentiometers
- Capacitance pickup
- Inductance pickup
- LVDT (Linear Variable Differential Transformer)

(b) Measuring the strain (more common for low cost equipment)
- Strain gauges in full bridge, half bridge and quarter bridge configurations.

(c) Measuring the pressure (more rugged and expensive)
- Piezoelectric crystals

2.7.2 Two-component Dynamometer Design

Though a number of dynamometers have been designed for the range of metal cutting activities specifically oriented towards most of the practical machining operations, in this book the details of a two-component dynamometer used for orthogonal cutting is described. This is the design developed at MIT in the early 1950's. It utilizes strain gages to measure the force. A strain gage system actually measures strain and not force. Therefore a strain gage transducer is designed in such a way that the force to be measured develops a suitable stress level in the sensing element. The strain gage then measures the strain in one or more locations of the sensing element. Strain is measured only locally on several points on the surface of the sensing element to compensate for the temperature variations.

The dynamometer structure consists of a rod of suitable diameter to provide the necessary rigidity. A part of the rod in the middle portion is reduced in size and machined flat which acts as the sensing structure. The strain gages are then fixed at the flat portions in appropriate locations as shown in Fig. 2.29 to measure both the horizontal and vertical components of the cutting forces. The strain gages are then formed into a full Wheatstone bridge structure, as shown, to measure the change in the resistance which corresponds to the acting cutting forces.

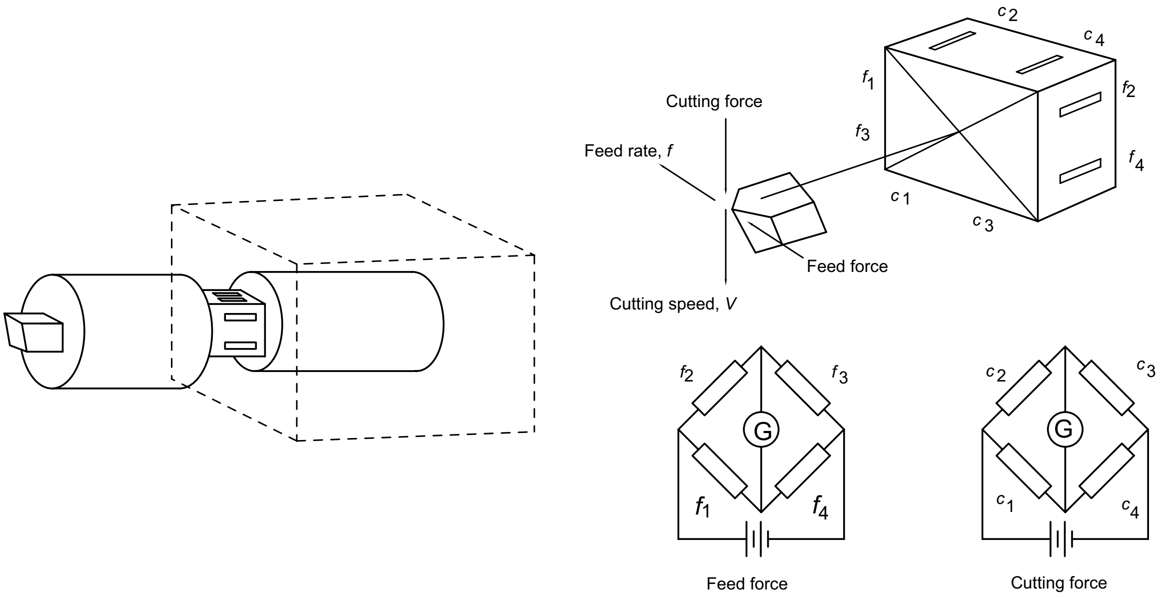


Fig. 2.29 *Two component dynamometer used in orthogonal cutting.*

2.8 CUTTING TOOL MATERIALS

Various cutting tool materials have been used in the industry for different applications. A number of developments have occurred in the 20th century, thanks to the aerospace and nuclear programmes. A large variety of cutting tool materials have been developed to cater to the variety of materials used in these programmes. Before we proceed to know these materials, let us look at the important characteristics expected of a cutting tool material:

(i) Higher hardness than that of the work piece material being machined, so that it can penetrate the work material.

(ii) Hot hardness, which is the ability of the material to retain its hardness at elevated temperatures, in view of the high temperatures existing in the cutting zone. This requirement becomes more and more stringent with the increasing emphasis on higher cutting speeds to bolster productivity.

(iii) Wear resistance – The chip-tool and chip-work interfaces are exposed to such severe conditions that adhesive and abrasion wear is very common. The cutting tool material should therefore have high abrasion resistance to improve the effective life of the tool.

(iv) Toughness – Even though the tool is hard, it should have enough toughness to withstand the impact loads at the beginning of the cut or to force fluctuations due to imperfections in the work material. This requirement is more useful for interrupted cutting, e.g. milling.

(v) Low friction – The coefficient of friction between chip and tool should be low which would allow for lower wear rates and better chip flow.

(vi) Better thermal characteristics – Since a lot of heat is generated at the cutting zone, it is necessary that the tool material should have higher thermal conductivity to dissipate this heat in the shortest time, otherwise the tool temperature will become too high thus reducing its life.

All these properties may not be found in a single tool material. A comparison of the various properties of the cutting tool materials are presented in Table 2.3. Improvements in tool materials having been taking place over the past century to give us better cutting performance. Some of these tool materials have been discussed next.

Table 2.3 Comparative properties of cutting tool materials

Cutting Tool Material	Hardness, R_A			Transverse Rupture Strength × 10^3 MPa
	Room Temperature	540°C	760°C	
High-speed steel	85 to 87	77 to 82	Very low	3.8 to 4.5
Cast cobalt	82 to 85	75 to 82	70 to 75	1.4 to 2.8
Carbides	89 to 94	80 to 87	70 to 82	to 2.4
Ceramics	94	90	87	0.5 to 0.4
Diamond	7000 Knoop	7000 Knoop	7000 Knoop	—

2.8.1 Carbon Tool Steels

These are the earliest tool materials used. These are essentially plain carbon steels with carbon percentages between 0.6 to 1.5% and some very small alloy additions such as manganese, silicon, tungsten, molybdenum, chromium and vanadium. The major disadvantage with this range of cutting tool materials is their inability to withstand high temperatures. Beyond 200°C they lose their hardness and cease to cut. Thus these are useful only for very low cutting speeds (about 0.15 m/s) and can be used with low temperature generating operations such as machining wood, magnesium, brass and aluminium. They are easy to prepare and ground, as a result they are used for form tool making to be used for low quantity production.

2.8.2 High Speed Steel

Taylor and White developed this new generation tool material at the turn of the 20th century. They were able to significantly improve the cutting speeds by 3 to 5 times (about 0.5 m/s) than the speed prevalent at that time, using carbon tool steels. Because of this high cutting speed capability they were termed as high speed steels or more popularly called HSS.

This class of tool materials have significant quantities of tungsten, molybdenum, chromium and vanadium. The complex carbides of tungsten, molybdenum and chromium distributed throughout the metal matrix provide very good hot hardness and abrasion resistance. The major alloying elements, which contribute to the hardness is tungsten and molybdenum. Tungsten is expensive, while molybdenum is cheap but has higher toughness. For the same hardness, less amount of molybdenum needs to be added, however more care need to be exercised in hardening as decarburizing takes place in molybdenum steels. Also they have narrow temperature range for heat treatment. Molybdenum tool steels are more popular.

The main advantages of high speed steels are their high hardness, hot hardness, good wear resistance, high toughness and reasonable cost. Toughness of high speed steels is highest among all the cutting tool materials. Thus they are quite extensively used in interrupted cutting such as milling. The hardness of HSS falls rapidly beyond 650°C as shown in Fig. 2.30, and thus they are limited to lower cutting speeds of the order of 0.5 to 0.75 m/s.

Tool steels have been classified by AISI as T-type and M-type depending on whether tungsten or molybdenum is the major alloying element present in the steel. Some typical compositions have been given in Table 2.4.

Recently the HSS tool steels are also being produced through the powder metallurgy route. In this method fine powder of alloy tool steel is compressed under hot isostatic pressure. With suitable hardening and tempering, this method provides for uniform dispersion of carbides in the matrix. These have been found to grind more easily, exhibit uniform properties and perform more consistently in cutting.

A recent development is the physical coating process (PVD - Physical Vapour Deposition) at lower temperatures, which allows the HSS tools to be coated with hard nitrides of titanium and aluminium. With much favourable cutting geometries and the hard coatings, the cutting performance and tool life of HSS tools has improved substantially. The PVD coatings are generally done at low temperatures, as a result the adherence of coating is a problem, which is solved by improved cleaning and etching techniques. There are efforts to further improve the cutting performance by improving the coating characteristics by combining various nitrides.

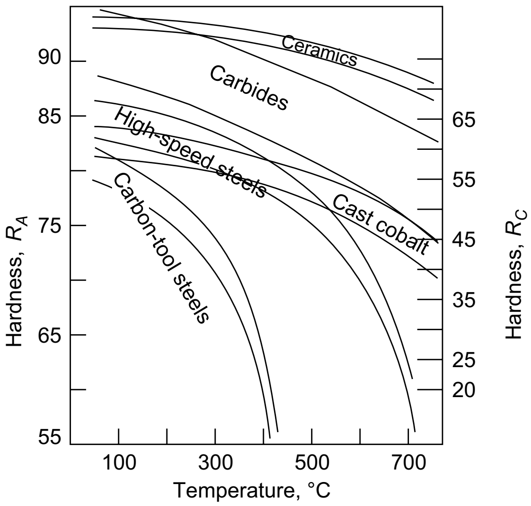


Fig. 2.30 *Variation of hardness with temperature for various cutting tool materials*

Table 2.4 **Typical compositions of high speed steel materials**

AISI Steel Type	% Chemical Composition						
	C	*Cr*	*V*	*W*	*Mo*	*Co*	W_{eq}
T1	0.70	4.0	1.0	18.0			18.0
T6	0.80	4.25	1.5	2.0	0.90	12.0	21.8
M1	0.80	4.0	1.0	1.5	8.0		17.5
M6	0.80	4.0	1.50	4.0	5.0	12.0	14.0
M30	0.85	4.0	1.25	2.0	8.0	5.0	18.0
M42	1.10	3.75	1.15	1.50	9.50	8.25	20.5

2.8.3 Cast Cobalt Alloys

These, termed as stellites, are normally produced by the powder metallurgy method, though casting is also used by some manufacturers. Fine powders of a number of non-ferrous metals (having compositions as shown in Table 2.5) are thoroughly mixed and compacted to the final shape under hot isostatic pressure. They are then ground to their final geometry. They retain their hardness even at elevated temperatures better than HSS and consequently are used at cutting speeds higher (25% higher) than HSS. Because of their formability these are used for making form tools. They have higher toughness and higher stiffness. Currently these are being phased out since carbides are now available which have a larger range of properties.

Table 2.5 Typical compositions and uses of cast non-ferrous alloys

Nominal % Composition								Grade
Cr	W	Mo	C	Mn	Si	Ni	Co	
30	4.5	1.5	1.1	1.0	1.5	3.0	rest	Roughing
31	10.5	—	1.7	1.0	1.0	3.0	rest	General purpose
32	17.0	—	2.5	1.0	1.0	2.5	rest	Finishing

2.8.4 Cemented Carbides

The best thing for the metal cutting industry is the invention of cemented carbides, which happened around 1926 in Germany. By far this is the largest percentage of cutting tools used in metal cutting production. Cemented carbides are produced by the cold compaction of the tungsten carbide powder in a binder such as cobalt, followed by liquid-phase sintering. These have a very large number of advantages compared to the other cutting tool materials.

(i) High hot hardness. These can retain their hardness to much higher temperatures and as a result the cutting speeds used are 3 to 6 times (about 5 to 6 m/s) than that of HSS.

(ii) Higher Young's modulus. This results in stiffer cutting tools with less tendency towards chatter.

However, carbides are more brittle and expensive.

It is possible to vary the composition of carbides to get a range of properties. The variations achieved are based on the amount of Co binder, different types of carbides and the grain size of carbide. Increasing the cobalt binder decreases the hot hardness and wear resistance while increasing the strength. The usual composition of the straight grade carbides is 6 wt. % Co and 94 wt. % WC with the cobalt composition ranging from 5 to 12 wt. %. For heavy interrupted and roughing operations high cobalt (Co) content is required while medium coarse grain tungsten carbide is used to withstand the shock. For finishing applications lower cobalt content is required as hardness becomes the important requirement. Addition of titanium carbide (TiC) increases the hot hardness, wear resistance, and resistance to thermal deformation, but decreases the strength. The usual composition is about 5–25 wt. %. Similarly the presence of tantalum carbide (TaC) increases the hot hardness and resistance to thermal deformation while decreasing the wear resistance and strength.

The ISO classification of carbide grades and their possible application is given in Table 2.6. The lower designation numbers such as P10, M10, and K10 are for higher speed finishing cut application, while the higher numbers such as P40 are for lower speed roughing applications.

The following guidelines would be useful for selecting a carbide grade.

(a) Choose a grade with the lowest cobalt content and the finest grain size consistent with adequate strength to eliminate chipping.

(b) Use straight WC grades if cratering, seizure or galling is not experienced in case of work materials other than steels.

(c) To reduce cratering and abrasive wear when machining steel, use grades containing TiC.

(d) For heavy cuts in steel where high temperature and high pressure deform the cutting edge plastically, use a multi-carbide grade containing W-Ti-Ta and/or lower binder content.

As the cobalt content increases, toughness and impact strength of cemented carbide increase while hardness, Young's modulus and thermal conductivity decrease. Fine grain carbides are harder compared to coarse grain carbides. Multi-carbide grades increase chemical stability, hardness and hot hardness.

Since tungsten and cobalt are expensive, some special cemented carbides having predominantly tantalum carbides with Ni and Mo as binder have been developed, for auto industry applications for finish machining of steels and malleable cast irons. These are sometimes called 'cermets'. These are relatively brittle and easy to chip. These are relatively cheap and should find widespread use in future.

TABLE 2.6 ISO Classification of cemented carbide tools

Main Groups of Chip Removal			Group of Application			Direction of Increase	
Symbol	Broad Categories of Materials to be Machined	Colour	Designation	Material to be Machined	Use and Working Condition	of Cutting Speed	of Feed Rate
P	Ferrous metals with long chips	BLUE	P01	Steel, steel castings	Finish turning and boring, high cutting speeds, small chip section, accuracy of dimensions and finish, vibration free operation	↑	↓
			P10	Steel, steel castings	Turning, copying, threading and milling, high cutting speeds, small or medium chip sections		
			P20	Steel, steel castings, malleable cast iron with long chips	Turning, copying, milling, medium cutting speeds and chip sections, planing with small chip sections		
			P30	Steel, steel castings, malleable cast iron with long chips	Turning, milling, planing, medium or low cutting speeds, medium or large chip sections, and machining in unfavourable conditions		
			P40	Steel, steel castings with sand inclusion and cavities	Turning, planing, slotting, low cutting speeds, large chip sections, with possibilities of large cutting angles for machining in unfavourable conditions and work on automatic machines		
			P50	Steel, steel castings of medium or low strength with sand inclusion and cavities	For operations demanding very tough carbides, turning, planing, slotting, low cutting speeds, large chip sections, with possibilities of large cutting angles for machining in unfavourable conditions and work on automatic machines		

(*Contd.*)

Main Groups of Chip Removal			Group of Application			Direction of Increase	
Symbol	**Broad Categories of Materials to be Machined**	**Colour**	**Designation**	**Material to be Machined**	**Use and Working Condition**	**of Cutting Speed**	**of Feed Rate**
M	Ferrous metals with long or short chips and nonferrous metals	YELLOW	M10	Steel, steel castings, manganese steel, grey cast iron, alloy cast iron	Turning medium or high cutting speeds, small or medium chip sections	↑	↓
			M20	Steel, steel castings, austenitic or manganese steel, grey cast iron	Turning, milling, medium or cutting speeds and chip sections		
			M30	Steel, steel castings, austenitic steel, grey cast iron, high-temperature resistant steels	Turning, milling, planing, medium or cutting speeds and medium or large chip sections		
			M40	Mild free cutting steel, low tensile steel, non-ferrous metals and light alloys	Turning, parting off particularly on automatic machines		
K	Ferrous metals with short chips, non-ferrous metals and non-metallic materials	RED	K01	Very hard grey cast iron, chilled castings of over 85 shore, high silicon aluminium alloys, hardened steel, highly abrasive plastics, hard cardboard, ceramics	Turning, finish turning, boring, milling, scraping	↑	↓
			K10	Grey cast iron over 220 BHN, malleable cast iron with short chips, silicon aluminium alloys, hardened steel, copper alloys, plastics, glass, hard rubber, hard cardboard, porcelain, stone	Turning, drilling, boring, milling, broaching, scraping		
			K20	Grey cast iron up to 220 BHN, non-ferrous metals, copper, brass, aluminium	Turning, planing, boring, milling, broaching, demanding very tough carbides		
			K30	Low hardness grey cast iron, low tensile steel, compressed wood	Turning, planing, milling, slotting, for machining in unfavourable conditions and with the possibility of large cutting angles		
			K40	Soft wood or hard wood, nonferrous metals	Turning, planing, milling, slotting, for machining in unfavourable conditions and with the possibility of large cutting angles		

Cemented carbides being expensive, are available in insert forms in different shapes such as triangle, square, diamond, and round. Each of the edge would act as a cutting edge. The typical construction of a cemented carbide tool is shown in Fig. 2.31. As seen from Fig. 2.31, the tool bit is made of tungsten carbide, while the tool holder shank is made from alloy steel to provide the necessary strength and reduce the total cost. After the use of a single edge, the tip would be indexed in the cutting tool holder, and thus these are called indexable bits. After all the edges are utilized, the tools are thrown out and a new bit is used in the tool holder. Thus these are also called throwaway bits. Because of their brittleness, generally small negative rake angles are used with the bits. However, in view of the developments in the processing methods and compositions, a number of grades are being offered by the various manufacturers that can have a positive rake angle also.

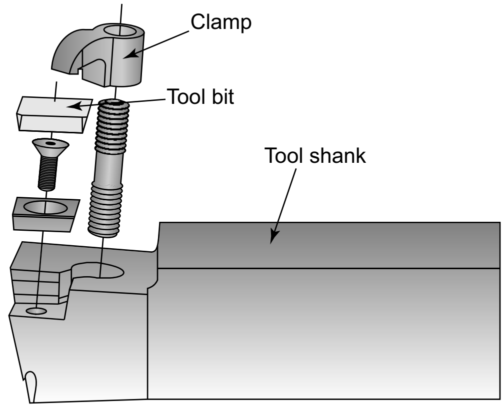


Fig. 2.31 *Construction of a Tungsten carbide turning tool*

Cemented carbides are not suitable for lower cutting speeds since the chips tend to weld and consequently chipping takes place. It is also not economical to use at lower speeds, since they can withstand higher temperatures.

2.8.5 Coated Carbides

With the increase in material characteristics to cater to the increasing service requirements, the need for developing better cutting materials has been felt since the World War II. Since the range of work materials is large, there is a need for hard and refractive coatings on conventional tool materials, so that the same could be used in diverse situations. Thus several coatings and coating methods have been developed for cutting tools. Since late 60's thin (about 5 μm) coating of TiN has been used on cemented carbide tools.

Ceramic coatings used are hard materials and therefore provide a good abrasion resistance. They also have excellent high temperature properties such as high resistance to diffusion wear, superior oxidation wear resistance, and high hot hardness. Further the good lubricating properties of the coatings minimise friction at the tool–chip and tool–work piece interfaces, thereby lowering the cutting temperature. All these translate into lower forces generated during machining compared to uncoated tools.

The substrate is a normal cemented carbide tool that has the necessary strength and toughness. The coating on the top, as shown in Fig. 2.32, provides the required hardness and refractoriness that prolongs the life of the tool. The life of the coated tools is often two to three times of the uncoated, while these can be used at higher cutting speeds, thus increasing productivity.

The coatings need to be metallurgically bonded to the substrate. These coatings such as titanium carbide, titanium nitride, aluminium oxide, hafnium nitride and hafnium carbide or multiple coatings of the above, are deposited on the carbide tool bits by the chemical vapour deposition (CVD) process. The chemical reaction necessary to deposit the required coating takes place close to the substrate. The coating is deposited literally atom by atom onto the surface thereby providing a very strong adhesion between the coating and the substrate.

Typical coating materials used include TiC, TiN, Al_2O_3, TiCN, TiAlN, TiZrN, TiB_2 and diamond. Typical physical properties of the coating materials are given in Table 2.7. It has been noted (Jindal, Santhanam, Schleinkofer, and Shuster) that the Vickers micro hardness varies as a function of temperature for some of the coatings. The TiCN coating has the highest room temperature hardness, but above 750°C the TiAlN coating

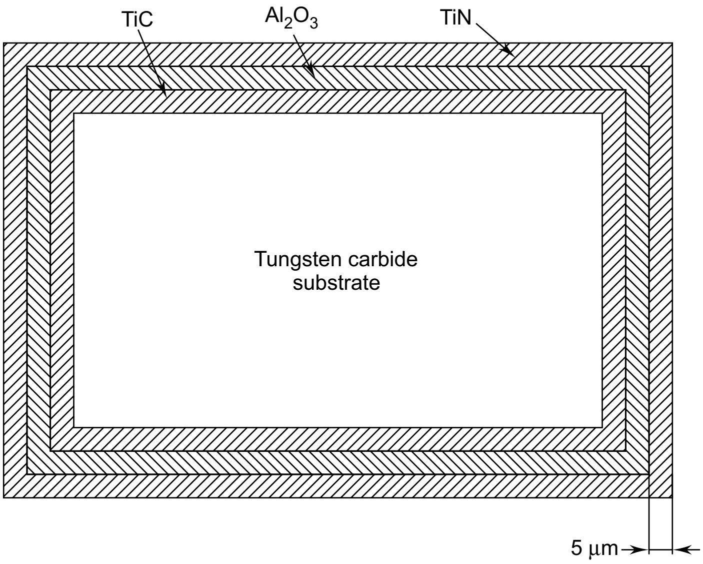


Fig. 2.32 *Schematic representation of a multicoated carbide tool bit*

is harder than TiCN or TiN coatings. At 1000°C, TiAlN is considerably harder than TiCN and TiN. Titanium Nitride is one of the first coatings and the most widely used one. It provides low friction, high hardness, higher refractoriness and good adhesion to the substrate. It also has greater resistance to flank wear. Titanium carbide has higher resistance to flank wear. Ceramic (such as Al_2O_3) coatings have higher refractoriness and resist crater wear as well as flank wear. However these do not bond well with the substrate. It has been noticed that the tool life will improve with different types of coatings. During the actual experiments when machining AISI 1045 steel, it was noticed (Jindal, Santhanam, Schleinkofer, and Shuster) that PVD coating of TiN has less improvement compared to the PVD coating of TiAlN. The improvement in the coating of TiCN remained in between that of TiN and TiAlN.

Table 2.7 **Properties of some coating materials**

Coating	Room Temperature Hardness (HV)	Oxidation Resistance, °C	Coefficient of Friction
TiN	1930–2200	600	0.4–0.5
TiCN	2730–3000	400	0.3
TiAlN	3000–3500	800	0.7
TiN/AlN	4000	950	—
TiAlCN	3200	600	—

Multiple coatings generally provide higher tool life and offer broader use for machining differing work materials. The combinations that have found wide use are TiCN + Al_2O_3 + TiN; and TiN + TiC + Al_2O_3. By virtue of the general applicability of a single grade for a spectrum of machining situations, the shop needs to maintain an inventory of small number of varieties. Coated carbides are being increasingly used in industry in comparison to the uncoated varieties. It is estimated that 40% of all cutting tools used in the industry are coated.

2.8.6 Ceramics

Ceramics are essentially alumina (Al_2O_3) based high refractory materials introduced specifically for high speed machining of difficult to machine materials and cast iron. These can withstand very high temperatures, are chemically more stable and have higher wear resistance than the other cutting tool materials. In view of their ability to withstand high temperatures, they can be used for machining at very high speeds of the order of 10 m/s. It is possible to get mirror finish on cast iron using ceramic turning. The main problems of ceramic tools are their low strength, poor thermal characteristics and the tendency to chipping. About 2 to 5 weight% of zirconium oxide (ZrO_2) is added to alumina that increases the fracture toughness of the tool without affecting its wear resistance. The machine tools used for ceramic machining have to be extremely rigid to provide smooth machining conditions for machining with ceramics and should be able to provide high cutting speeds. They are not suitable for intermittent cutting or for low cutting speeds.

Apart from the pure alumina based ceramics, sometimes other materials such as Titanium carbide (TiC), Titanium Nitride (TiN), and Titanium diboride (TiB_2) are added to enhance the transverse rupture strength, hardness and thermal shock resistance. Some yittria may also be added as a sintering agent. Other ceramics of relatively recent origin are alumina-titanium diboride, alumina-zirconia-tungsten compound, and silicon-aluminium-oxygen-nitrogen (Si-Al-O-N) complex compound. These are less hard but tougher than alumina ceramics.

Whisker reinforced alumina ($Al_2O_3 + SiC_w$) are alumina based ceramic tools that have strengthening silicon carbide fibres (about 25 wt. %) added into the alumina matrix. This material has increased fracture toughness, high thermal conductivity and lower thermal expansion coefficient. Silicon nitride (Si_3N_4) based ceramics are also used as cutting tool materials. These have good oxidation resistance, good mechanical strength and high hardness. Typical properties of ceramic materials that are used for cutting tool materials are given in Table 2.8.

Table 2.8 Properties of ceramic materials

Base System	Density g/cm^3	Hardness		Transverse Rupture Strength, MPa
		25°C (HRA)	1000°C (HV)	
Al_2O_3	3.98	93.9	710	50
Al_2O_3 + TiC	4.24	94.3	770	80
Si_3N_4	3.27	92.6	1100	100

Ceramic tools should be used with very high cutting speeds on steels. They are not suitable for low cutting speeds or for intermittent cutting. Cutting fluid if applied should be in flooding with copious quantity of fluid to thoroughly wet the entire machining zone, since ceramics have very poor thermal shock resistance. Else it can be machined with no coolant. Ceramic tools are used for machining work pieces, which have high hardness such as hard castings, case hardened and hardened steels. Typical products that can be machined are brake discs, brake drums, cylinder liners, and flywheels. Correct cutting speed produces good surface finish, optimum productivity and better tool life.

Ceramic tools cannot machine some materials such as aluminium, titanium, since they have strong affinity towards them, as a result of which chemical reactions are likely to take place.

Among other things, some of the vital requirements when machining with ceramics are:

- Use the highest cutting speed recommended and preferably select square or round inserts with large nose radius.
- Use rigid machine with high spindle speeds and safe clamping angle.

- Machine rigid work pieces.
- Ensure adequate and uninterrupted power supply.
- Use negative rake angles so that less force is applied directly to the ceramic tip.
- The overhang of the tool holder should be kept to a minimum; not more than 1.5 times the shank thickness.
- Large nose radius and side cutting edge angle on the ceramic insert to reduce the tendency of chipping.
- Always take a deeper cut with a light feed rather than a light cut with heavy feed; ceramic tips are capable of cuts as deep as one-half the width of the cutting surface on the insert.
- Avoid coolants with aluminium oxide based ceramics.
- Review machining sequence while converting to ceramics and if possible introduce chamfer or reduce feed rate at entry.

2.8.7 Diamond

Diamond is the hardest known (Knoop hardness ~ 8000 kg/mm^2) material that can be used as a cutting tool material. It has most of the desirable properties of a cutting tool material such as high hardness, good thermal conductivity, low friction, non-adherence to most materials, and good wear resistance. However, the factors that weigh against its use are the high cost, possibility of oxidation in air, allotropic transformation to graphite above temperatures of 700°C, very high brittleness and difficulties associated in shaping it to suitable cutting tool form.

Natural diamond tools could be used for relatively light cuts where these provide extremely high tool life, which can easily justify the high cost of diamond. However, natural diamond is unreliable in performance because of the impurities present and easy cleavage. Artificial diamonds are basically polycrystalline (PCD) in nature. These are extensively used in industrial application because they can be formed for any given shape with a substrate of cemented carbide.

Polycrystalline diamond tools are metallurgically bonded to a tungsten carbide substrate and cut into small bits. The tungsten carbide provides the necessary elastic support for the hard diamond tool. This is then placed in the carbide inserts that have precision pockets to receive the diamond bit and then brazed as shown in Fig. 2.33.

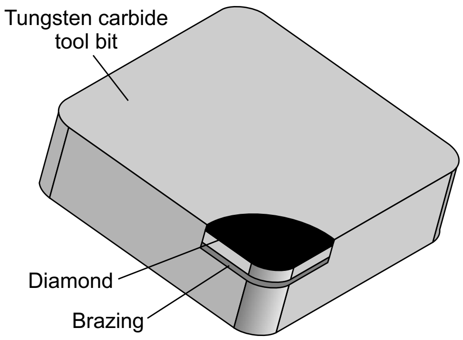


Fig. 2.33 *Polycrystalline diamond brazed to the carbide tool bit*

They are used with a negative rake angle (–5°) for machining hard materials while positive rake angles (15°) can be used for soft materials such as copper. They cannot be used for machining low carbon steels, titanium, nickel, cobalt or zirconium because of the possible reaction with the work material. Typical materials that are machined with diamond tools and the suggested process parameters are given in Table 2.9.

2.8.8 Cubic Boron Nitride (CBN)

Cubic Boron Nitride (CBN) is next in hardness only to diamond (Knoop hardness ~ 4700 kg/mm^2). It is not a natural material but produced in the laboratory using a high temperature/ high pressure process similar to the making of artificial diamond. CBN is less reactive with materials like hardened steels, hard chill cast iron, and nickel base and cobalt based super alloys, and hence is used effectively for machining these alloys. These are more expensive than cemented carbides but in view of the higher accuracy and productivity for difficult to machine materials, they are used in special applications as listed above.

Table 2.9 Cutting data for Polycrystalline Diamond (PCD) tool bits (From Seco catalogue)

Material	Cutting Speed, m/min	Depth of Cut, mm	Feed Rate, mm/rev
Aluminium alloys > 11% Si	300–3000	Up to 4	0.10–0.50
MMC SiC-particles (15–30%)	200–800	Up to 3	0.10–0.50
Copper, Brass and Bronze	600–1200	Up to 3	0.10–0.50
Carbon and graphite	100–400	Up to 3	0.10–0.50
Sintered carbide	10–40	Up to 3	0.10–0.50
Green carbide	80–200	Up to 0.5	0.10–0.50
Green ceramic	100–600	Up to 2	0.05–0.20
Plastic composites	100–1000	Up to 3	0.10–0.50

Thus, there is a large variety of cutting tool materials available which should be selected carefully for a given application, taking all the intervening factors into account. The recommendations and characteristics have been summarised in Table 2.10. These can act as guidelines, however many of the cutting tool manufacturers such as Sandvik, Widia provide detailed literature to help in the choice of cutting tools. These along with the Metal Cutting Handbook should be used for finalising the tool material selection.

Table 2.10 Summary of applications for various cutting tool materials [Komanduri]

Tool Material	Work Materials	Remarks
Carbon steels	Low strength, softer materials, nonferrous alloys, plastics	Low cutting speeds, low strength materials
Low/medium alloy steels	Low strength, softer materials, nonferrous alloys, plastics	Low cutting speeds, low strength materials
HSS	All materials of low and medium strength and hardness	Low to medium cutting speeds, low to medium strength materials
Cemented carbides	All materials up to medium strength and hardness	Not suitable for low speed application
Coated carbides	Cast iron, alloy steels, stainless steels, super alloys	Not for Titanium alloys, not for non-ferrous alloys as the coated grades do not offer additional benefits over uncoated.
Ceramics	Cast iron, Ni-base super alloys, nonferrous alloys, plastics	Not for low speed operation or interrupted cutting. Not for machining Al, Ti alloys.
CBN	Hardened alloy steels, HSS, Ni-base super alloys, hardened chill cast iron, commercially pure nickel	High strength, hard materials
Diamond	Pure copper, pure aluminium, Al-Si alloys, cold pressed cemented carbides, rock, cement, plastics, glass-epoxy composites, non-ferrous alloys, hardened high carbon alloy steels (for burnishing only), fibrous composites	Not for machining low carbon steels, Co, Ni, Ti, Zr.

2.9 THERMAL ASPECTS

Benjamin Thomson (1798) conducted the first experiments on machine tools when he measured the thermal energy involved during the boring of brass cannon. He observed that all the mechanical energy is converted into thermal energy. He used the calorimetric method.

Temperature of cutting is a very important parameter, which is of great consequence with reference to the life of a tool. The surface of the tool, if proper precautions are not taken, may be overheated at isolated points and localized phase transformations can occur. This may result in the softening of the surface of the tool and frequently very small cracks will be formed as a result of the intense residual stress system that accompanies surface transformation.

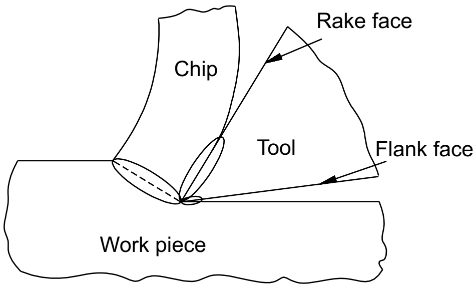


Fig. 2.34 *The main regions of heat generation in metal cutting*

Because of the very large amount of plastic strain involved in metal cutting, it is unlikely that more than 1% of the work done is stored as elastic energy (which can be neglected), the remaining 99% goes to heat the chip, the tool and the work. The typical zones in metal cutting where the heat is generated (Fig. 2.34) are

(i) in the shear plane where the heat is generated because of internal friction, and this accounts for 65 to 75% of the total heat generated,
(ii) the friction at the chip tool interface which causes heat of the order of 15 to 25%, and
(iii) the friction at the tool work interface which causes heat of the order of 10%.

The metal in the area ahead of the cutting edge of the tool is severely compressed, resulting in temperatures high enough to allow plastic flow. As the atoms in the metal ahead of the tool are disturbed, the friction involved in their sliding over one another is probably responsible for the shear plane heat. As the tool continues to push through the work piece, a chip eventually slides up the rake face of the tool. This sliding is responsible for frictional heat.

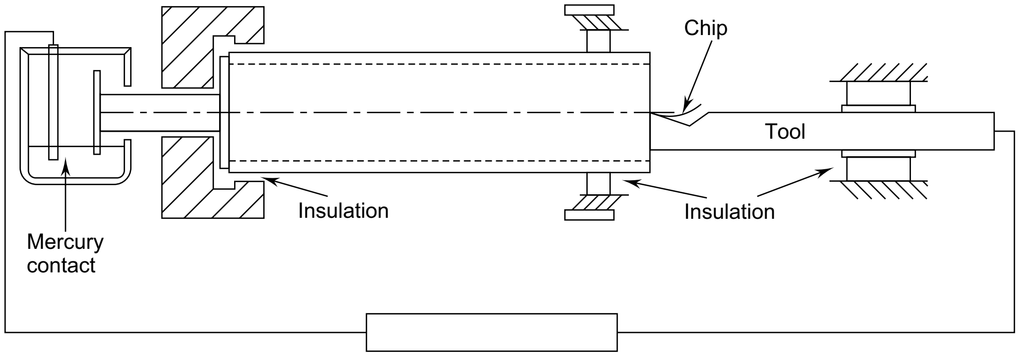
There are a number of methods for measuring the chip tool interface temperature.

(a) Radiation pyrometers
(b) Embedded thermocouples
(c) Temperature sensitive paints
(d) Temper colours
(e) Indirect calorimetric technique
(f) Tool work thermocouple.

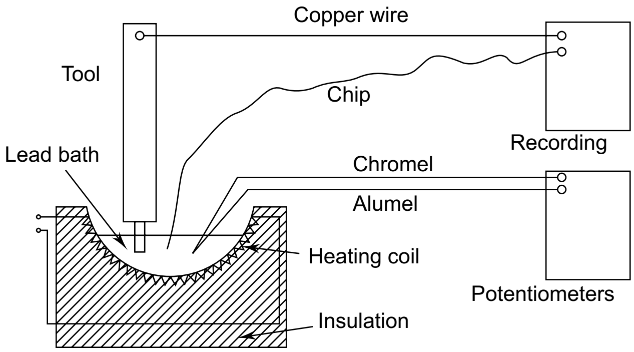
Of all these methods, the tool work thermocouple technique is the most widely used technique for the measurement of the average chip tool interface temperature. The other methods suffer from various disadvantages such as slow response, indirectness, and complications in measurement.

2.9.1 Tool Work Thermocouple

Tool and work materials are dissimilar and the temperature in the cutting zone is higher than the rest of the tool or work. Hence the tool work contact area serves as the hot junction in a thermoelectric circuit and the emf generated is proportional to temperature. A typical setup is shown in Fig. 2.35. The end of the tool is ground to a very small diameter such as 3 mm and a long chip generated by machining with very light feed



(a) Setup for measuring interface temperature



(b) Setup for calibrating the tool-chip thermocouple

Fig. 2.35 *Setup for measuring average chip-tool interface temperature using chip-tool thermo couple technique*

is made as the tool work thermocouple. A mercury slip ring connection at the end of the work piece through the spindle bore is a convenient way for completing the emf circuit.

Sources of errors in tool work thermocouple:

(i) Not an ideal thermocouple, which means emf, is low and the calibration need not always be a straight line.

(ii) Doubt the calibration procedure because it is done in the stationary situation.

As shown, it is possible only to find the average chip-tool interface temperature from the above experiment. However, it is possible to use analytical techniques for predicting the average temperatures as well as temperature distributions.

2.9.2 Analytical Determination of Temperature in Metal Cutting

Majority of the energy involved in shearing the metal will be converted to heat that will transfer to the tool and work piece, thereby raising their temperatures. This heat will move partly to the work piece, chip and the tool. While the component that gets into the tool tip raises the temperature in that small area and remains in the cutting zone, the rest of heat will be removed by the chips and the work piece moving away from the cutting zone. As a result, the temperature of the tool tip would be highest. The temperature then would conduct through the tool along the rake face as well as the flank face. Most of the tool wear mechanisms are thermally activated, which will increase the wear along the rake and flank faces of the tool. To get a generic idea, Cook developed a formula to estimate the tool tip temperature using dimensional analysis as follows:

$$T = 0.4\, T_a \left(\frac{V\, d}{K}\right)^{\frac{1}{3}} \tag{52}$$

Where K = thermal diffusivity of the work material, mm^2/s

Specific heat = c; Density = ρ

Specific cutting energy, u_s given by eq. 41.

$$\text{Adiabatic temperature, } T_a = \frac{u_s}{\rho c} \tag{53}$$

This equation provides a reasonable estimate for the temperature for rake angles between –5 and +15°. More complex analytical determination methods have been developed which are beyond the scope of this book.

Highest temperature on the tool can be seen in a typical temperature contour plot in metal cutting in Fig. 2.36.

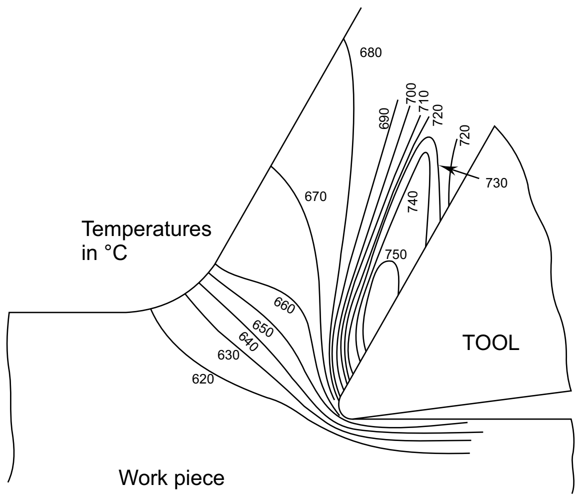


Fig. 2.36 *Temperature contours in a cutting tool [Boothroyd]*

The cutting process parameters - speed, feed and depth of cut have considerable effect on the cutting temperatures generated. Of these, cutting speed has the highest effect as shown in Fig. 2.37, since this controls the total energy input to the metal cutting operation. The next influence is that of feed as shown in Fig. 2.38. The effect of depth of cut is the least of all.

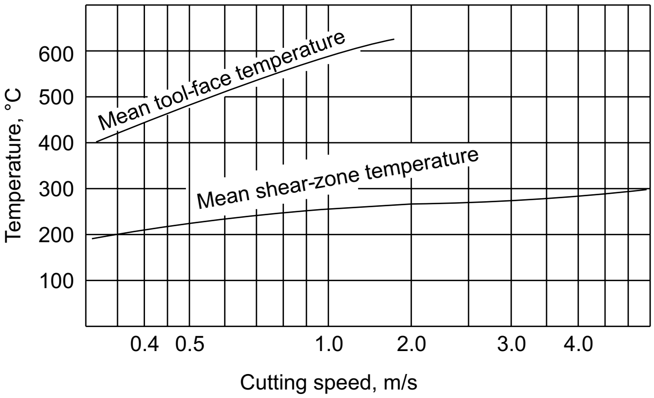


Fig. 2.37 *The effect of cutting speed on the temperature of the tool*

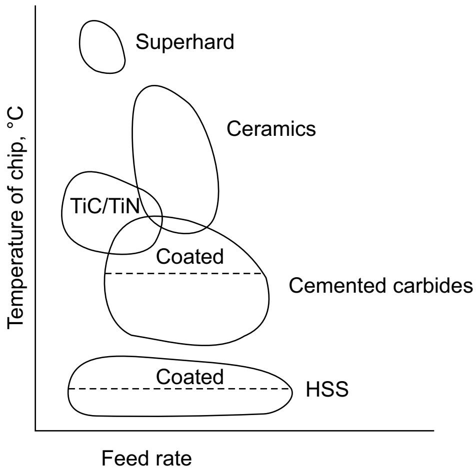


Fig. 2.38 *The effect of feed on the temperature of the tool*

2.10 TOOL WEAR AND TOOL LIFE

With the usage of tools over a long time, they are subjected to wear. The type of wear found in cutting tools is shown in Fig. 2.39. There are two major types of wear found in tools. They are:

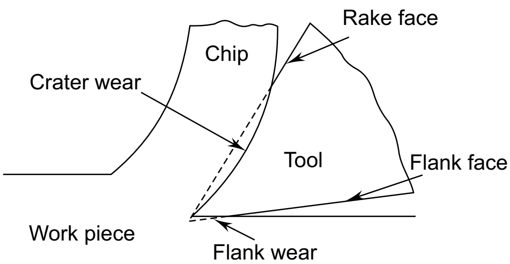


Fig. 2.39 *Typical wear patterns present in cutting tools*

Crater wear: The crater is on the rake face and is more or less circular. The crater does not always extend to the tool tip, but may end at a distance from the tool tip. It increases the cutting forces, modifies the tool geometry, and softens the tool tip.

Flank wear: Flank wear or wear land is on the clearance surface of the tool. The wear land can be characterised by the length of wear land, *w*. It modifies the tool geometry and changes the cutting parameters (depth of cut).

The typical wear patterns used as tool life criteria, as standardised by ISO are shown in Fig. 2.40. These are to be used as tool life criteria as discussed later.

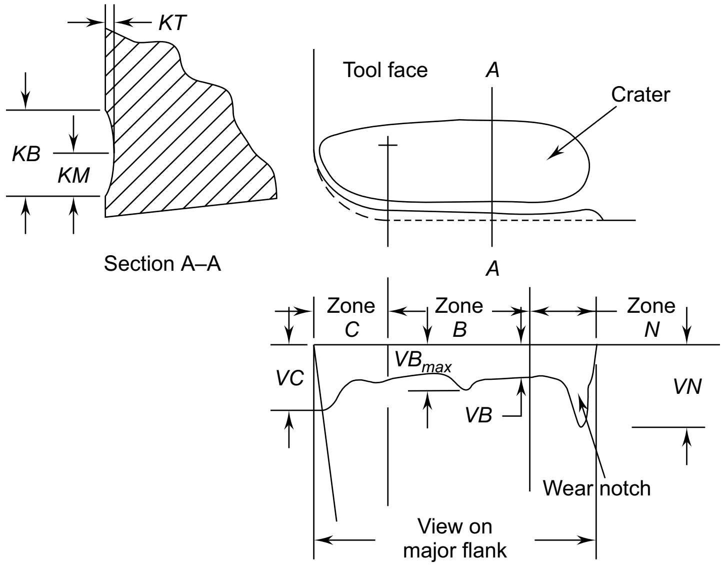


Fig. 2.40 *The wear parameters and their characterisation as suggested by ISO*

Cutting tools are subjected to extremely severe cutting conditions such as the following:

- metal to metal contact with work and chip
- very high stress
- very high temperature
- virgin metal
- very high temperature gradients
- very high stress gradients

Because of all the above mentioned factors, the tool-chip and tool-work interfaces exhibit the type of wears found. As tool wear progresses, cutting forces and vibrations increase. Tool tip softens, flows plastically, and gets a blunt edge, which results in further progression of plastic deformation from tool tip to the interior. After that the tip of the tool almost gets separated.

The presence of crater wear in very small sizes is not of much consequence as far as the machining performance is concerned. Initially, they may increase the rake angle and thus decrease the cutting forces. However, as the depth increases the friction increases, the chip contact length increases and consequently the decrease in the machining performance. Ultimately with very large crater depth (KT), the tool tip weakens and fails catastrophically. However, normally the tool is removed from service much before this, because of flank wear.

Flank wear directly affects the component dimensions. Thus there is always a close limit on the value of the wear land. In addition, as the wear land progresses, the tool tip becomes weak because of the progress of crater wear. Thus in many situations, the crater wear is more harmful if is deep from the tool re-sharpening point of view, since sizeable tool material volume needs to be re-ground away.

A number of wear mechanisms as follows have been proposed to explain the observed tool wear phenomenon.

- Adhesion
- Abrasion
- Diffusion
- Fatigue

The rate of change in wear patterns with time is shown in Fig. 2.41 for crater wear and Fig. 2.42 for flank wear. Note that at low speeds, the land wear predominates, whereas at higher speeds, the two wear rates are essentially equal. At even greater speeds crater wear predominates. At high speeds, probably the temperature is a predominant factor which governs the wear growth. The land wear curves generally exhibit a portion of initial rapid wear, whereas the crater wear curves are generally linear (Fig. 2.42).

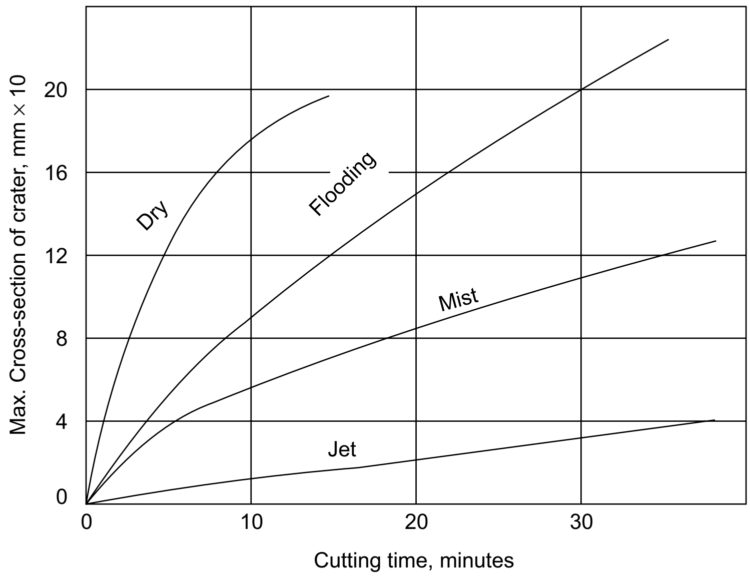


Fig. 2.41 *Progress of crater wear with time*

Initially when the sharp tool comes into contact with the work piece for metal removal, all the heat generated quickly raises the tool temperature to a very high level. This is because the tool material present at the tip being small will not be able to conduct all the heat. This causes a very rapid wear rate as seen in Fig. 2.42 in the first region. With a small amount of flank wear taking place, the tool material becomes sufficient to conduct the heat conveniently, and thus the temperature gets distributed uniformly in the tool. This results in a lower temperature and consequently a uniform and lower rate of flank wear growth. This is characterised by the second region in Fig. 2.42. As time progresses, with increased friction, the cutting forces increase and consequently the temperature rises catastrophically, bringing the end of the tool. This is characterised by the third region in Fig. 2.42. The end of second region can be conveniently used as a tool failure criterion as discussed later. Also the wear rate is substantially affected by the cutting speed as shown in Fig. 2.43.

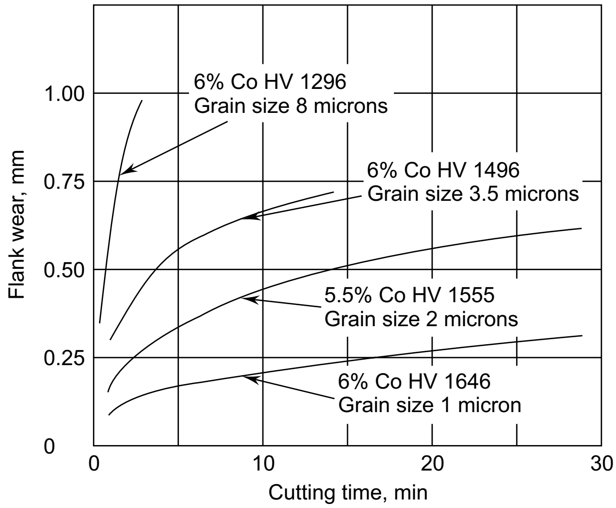


Fig. 2.42 *Progress of flank wear with time*

The following crater wear mechanism for HSS was suggested by Cook. One of the possible mechanisms is that iron atoms in the tool diffuse into the chip material as it moves by. The chip material is in the process of deformation and contains many potential sites for the diffusing iron atoms, which is more than the tool will have for reverse diffusion. Cook said that the chip-tool interface on an atomic scale will somewhat resemble a grain boundary. Hence a very crude model suggested that every time an atom at the surface attained thermal energy equal to that required for grain boundary diffusion, it would transfer to the chip structure and be swept away.

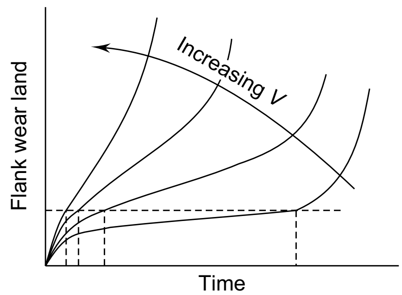


Fig. 2.43 *Progress of flank wear with time as affected by cutting speed*

Carbide tools fail mainly by plastic deformation and gradual wear. One of the possible mechanisms is the diffusion of cobalt binder thus weakening the carbide matrix. Another possibility is the decomposition of the free tungsten carbide phase along with the decomposition of the complex carbide particles.

2.10.1 Tool Life

Tool life represents the useful life of the tool, generally expressed in time units from the start of a cut to an end point defined by a failure criterion.

A tool that no longer performs the desired function is said to have failed and hence reached the end of its useful life. At such an end point the tool is not necessarily unable to cut the work piece but is merely unsatisfactory for the purpose . The tool may be re-sharpened and used again.

The tool life values as suggested by ISO are:

$VB = 0.3$ mm if the flank is regularly worn in zone *B*, or

$VB_{max} = 0.6$ mm if the flank is irregularly worn, scratched, chipped or badly grooved in zone *B*.

The tool life can be specified by any of the following measurable quantities:

- Actual cutting time to failure
- Length of work cut to failure
- Volume of metal removed to failure
- Number of components produced
- Cutting speed for a given time to failure.

All these various parameters are related and are used depending upon the final function of a given operation. However, the problem normally faced is in the definition of the failure criterion.

For example, in a roughing operation, the surface finish and dimensional accuracy are of little consequence. Therefore, excessive rise in cutting forces and the subsequent rise in power consumed will be of major importance, and hence the failure criterion. Whereas, for finishing operations, the reverse is true.

Sometimes, in roughing operations, the tool will be allowed to complete failure but this is to be avoided because of possible damage to work and tool. Therefore just before this happens, the tool should be withdrawn and reground.

The following are some of the possible tool failure criteria that could be used for limiting tool life.

Based on tool wear:

(i) Chipping or fine cracks developing at the cutting edge.
(ii) Wear land size
(iii) Crater depth, width or other parameters
(iv) A combination of the above two
(v) Volume or weight of material worn off the tool
(vi) Total destruction of the tool

Based on consequences of worn tool

(i) Limiting value of surface finish
(ii) Limiting value of change in component size
(iii) Fixed increase in cutting force or power required to perform a cut

The actual tool life values obtained will depend on the failure criterion adopted. Typical wear land sizes used as tool life limits are shown in Table 2.11.

Table 2.11 The various possible tool life criteria

Wear Land, mm	Tool Material	Remarks
0.75	Carbides	Roughing
0.25 to 0.38	Carbides	Finishing
1.50	HSS	Roughing
0.25 to 0.38	HSS	Finishing
0.25 to 0.38	Oxide	Roughing and finishing

The main advantage of wear land as a failure criterion is that it is fairly easy to measure and also is directly proportional to the surface finish and cutting forces. But practically, it is not uniform. At high speeds and feeds, diffusion wear will be more and then the crater wear also may provide a failure criterion.

But the problem is with the measurement of the crater to be scanned for maximum depth, which is laborious and time consuming. Total destruction criterion is easier to apply but then there are associated problems, e.g. in carbides it may spoil the nearby edges and also damage the work. Therefore a large wear land size can be equivalent to total destruction.

Alternatively the indirect methods mentioned above such as the fixed increase in the value of cutting force, or increase in the power consumed or measured vibrations, etc. can be used as an indication of the end of tool life. These are generally used in CNC machine tools where automatic tool life monitoring facilities are present.

2.10.2 Tool Life Equation

Taylor thought that there is an optimum cutting speed for maximum productivity. He reasoned this from the fact that at low cutting speeds, the tools have higher life but productivity is low, and at higher speeds the reverse is true. This inspired him to check the relationship between tool life and cutting speed. Based on his experimental work he proposed the formula for tool life

$$V\,T^n = C \tag{54}$$

where T is the tool life in minutes

V is the cutting speed, m/min

C and n are constants

Though this is a fairly good formula, but it does not take all the effecting parameters into account. As a result the applicability of the above formula is restricted to very small regions of cutting process parameters. This formula was extended by a number of researchers to reduce this deficiency, as given below.

$$T\,\theta^B = C \tag{55}$$

$$VT^{\frac{0.5-2x}{1-2x}} = \left[\frac{T_C\,H^{0.5}}{C'u_s\,A^x}\right]^{\frac{1}{1-2x}} \tag{56}$$

where H = specific heat × thermal conductivity

θ = tool temperature

A = area of cut

u_s = specific cutting energy/unit cutting force

C and x are constants

$$\theta = \frac{c_o\,u_s\,V^{0.44}\,A^{0.22}}{k^{0.44}\,\tau^{0.56}} \tag{57}$$

where k = thermal conductivity of work

τ = specific heat of work

$$V\,T^n\,f^{n1}\,d^{n2} = C \tag{58}$$

This is the most commonly employed tool life equation by a number of researchers. The constants for the above equation for some common work materials are given in Table 2.12 [Widia]. Apart from the cutting process parameters the tool life depends on the work material as well as tool material. The constants therefore are given for each combination of work and tool material.

Table 2.12 Constants for extended tool life equation

Tool Material	Work Material	Exponent for			Constant C
ISO Grade	AISI	Tool Life, n	Feed, n_1	Depth of Cut, n_2	
P01, P10	1020	–0.38	–0.06	–0.10	1150
P20, P30	1020	–0.38	–0.17	–0.11	780
P01, P10	1045	–0.22	–0.21	–0.11	350
P20, P30	1045	–0.22	–0.34	–0.12	226

2.10.3 Machinability

Machinability is the characteristic of the work material expressing its ease of machining. However convenient it looks, it is a characteristic which is difficult to quantify. Unfortunately like other characteristics of the material it is not a simple property. For example, hard work materials are difficult to machine. However, hardness alone would not be able to specify the machinability, since it also depends on the other characteristics such as tool materials used, process parameters, etc.

2.10.4 Variables Affecting Tool Life

The variables that affect tool life are:

(i) cutting conditions,
(ii) tool geometry,
(iii) tool material,
(iv) work material, and
(v) cutting fluid

The effect of cutting speed, feed and depth of cut is represented in the above tool life equation. If any of them increases, the tool life decreases as exemplified by the exponents in the above tool life equation.

Increasing the rake angle decreases the cutting forces and the heat produced at the tool tip, therefore increases tool life. However, increasing the rake angle to a large value reduces the tool material available at the tool tip for conducting heat generated, thus increasing the tool tip temperature. This would decrease the tool life, thus explaining the existence of an optimum value for the rake angle as shown in Fig. 2.44.

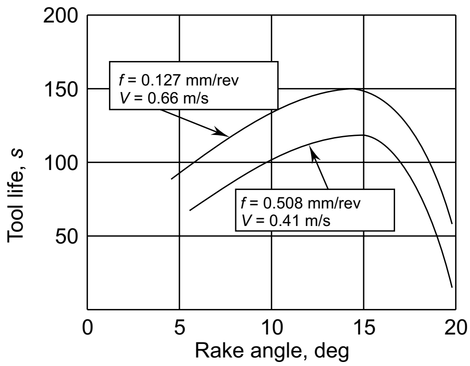


Fig. 2.44 *Tool life as affected by rake angle, and feed rate.*

Under the same conditions, tool material affects tool life. For example carbides have higher tool life than HSS as can be seen from the following Table 2.13 of exponents:

Table 2.13

Tool Material	Speed Exponent, n
HSS	0.08 to 0.20
Carbides	0.20 to 0.49
Oxides	0.50 to 0.70

Work material also plays a very great influence on the tool life. It is not the hardness alone, but the physical microstructure and the constituent phases that make a large difference in the actual tool life values. Steels containing free ferrite are soft and their machining involves no tool wear, however they are likely to form BUE and consequently poor surface finish. Presence of pearlite improves machinability giving good surface finish with no BUE formation, but decreases tool life. Presence of cementite, because of its hardness reduces tool life. Similarly in cast irons, the free graphite improves machinability and improves tool life.

2.11 SURFACE FINISH

Machining operations are performed in order to achieve better surface finish as compared to other manufacturing operations. Thus it is important to know the effective surface finish that can be achieved in a machining operation. The surface finish in a given machining operation is a result of two factors:

(i) the ideal surface finish, which is a result of the geometry of the manufacturing process, can be determined by considering the geometry of the machining operation.

(ii) the natural component, which is a result of a number of the uncontrollable factors in machining, is difficult to predict.

2.11.1 Ideal Surface Finish in Turning

In Fig. 2.45, the geometry of the surface produced in turning with a sharp cornered tool is shown. The situation shown is possible only with no sharp tool, no BUE, no machine tool chatter and elimination of all possible machine tool inaccuracies in movements.

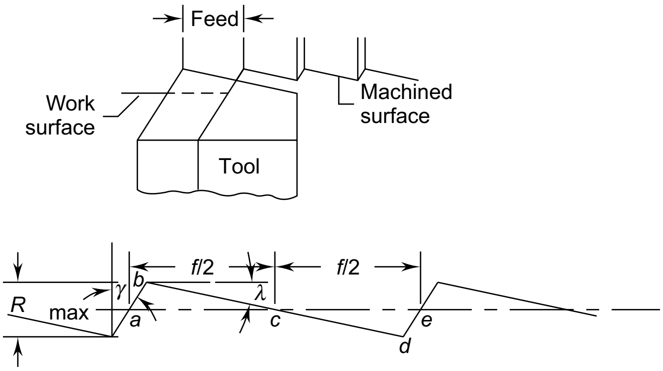


Fig. 2.45 *Surface profile as produced by turning with a sharp cutting tool*

Normally the surface finish is represented by any suitable index, such as arithmetic average, R_a, or centre line average, R_{cla} or any other suitable parameter. For our convenience, let us derive the finish expression for centre line average, R_{cla}. It is defined as "the arithmetic average value (AA) of the departure of the whole of the profile both above and below its centre line throughout the prescribed meter cut-off in a plane substantially normal to the surface".

Referring to Fig. 2.45, the surface roughness value is given by

$$R_{\text{cla}} = \frac{|\text{Area } abc| + |\text{Area } cde|}{f} \tag{59}$$

where f is the feed rate.

Since triangles *abc* and *cde* are equal,

$$R_{cla} = \frac{2\,(\text{area } abc)}{f} = \frac{R_{max}}{4} \tag{60}$$

From the geometry,

$$R_{max} = \frac{f}{\cot\lambda + \cot\gamma} \tag{61}$$

Substituting this in the above equation, we get

$$R_{cla} = \frac{f}{4\,(\cot\lambda + \cot\gamma)} \tag{62}$$

However, the actual turning tool used would have a nose radius in place of the sharp tool point, which modifies the surface geometry as shown in Fig. 2.46(a). If the feed rate is very small, as is normally happens in finish turning, the surface is produced by the nose radius alone as shown in Fig. 2.46(b).

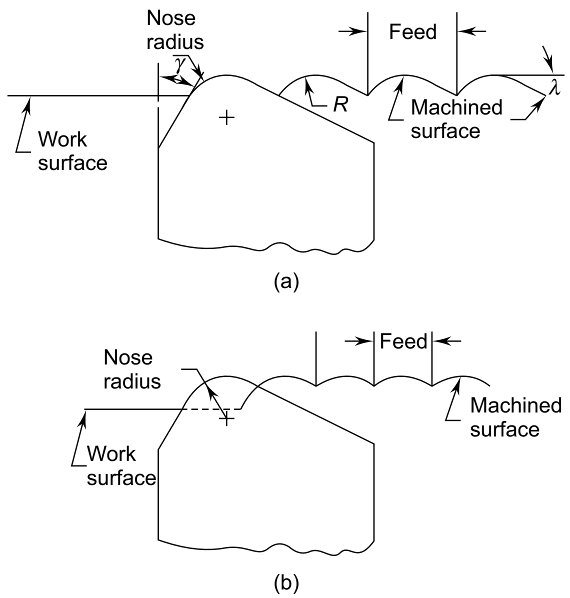


Fig. 2.46 *Surface profile as produced by turning with a cutting tool having a nose radius*

It can be shown that for case (a), the surface roughness value to be

$$R_{cla} = R\,(1 - \cos\gamma) + f\sin\gamma\cos\gamma - \sqrt{2\,f\,R\sin^3\gamma - f^2\sin^4\gamma} \tag{63}$$

It can be shown that for case (b), the surface roughness value to be

$$R_{cla} = \frac{8f^2}{18\sqrt{3}\,R} \tag{64}$$

The above are essentially geometric factors, and the values represent an ideal situation. However, the actual surface finish depends to a great extent upon a number of factors such as:

(i) the cutting process parameters, speed, feed and depth of cut
(ii) the geometry of the cutting tool
(iii) application of cutting fluid
(iv) work and tool material characteristics
(v) rigidity of the machine tool and the consequent vibrations

The major influence on surface finish is exerted by the feed rate and cutting speed. As the feed decreases, from the above equations, we can see that the roughness index decreases. Similarly as the cutting speed increases, we have better surface finish. Thus while making a choice of cutting process parameters for finish, it is desirable to have high cutting speed and small feed rate.

2.12 CUTTING FLUIDS

The functions of cutting fluids, which often are erroneously called coolants are:
(i) to cool the tool and work piece
(ii) to reduce the friction
(iii) to protect the work against rusting
(iv) to improve the surface finish
(v) to prevent the formation of built up edge
(vi) to wash away the chips from the cutting zone

However, the prime function of a cutting fluid in a metal cutting operation is to control the total heat. This can be done by dissipating and reducing the heat generated. The mechanisms by which a cutting fluid performs these functions may be listed as follows:

- cooling action
- lubricating action

Cooling Action

Originally it was assumed that the cutting fluid improves the cutting performance by its cooling properties alone. That is why the name coolant was given to it. Since most of the tool wear mechanisms are thermally activated, cooling the chip tool interface helps in retaining the original properties of the tool and hence prolongs its life. However, a reduction in temperature of the work piece may under certain conditions increase the shear flow stress of the work piece thereby decreasing tool life. It has been shown through a number of investigations that cooling, in fact, is one of the major factors in improving the cutting performance.

Lubricating Action

The best improvement in cutting performance can be achieved by the lubricating action since this reduces the heat generated, thus reducing the energy input to the metal cutting operation. However, if the cutting fluid is to be effective, it must reach the chip tool interface. However, it is not easy to visualize how it is accomplished in the case of a continuous turning with a single point turning tool, especially when the chip-tool contact pressure is as high as 70 MPa. Merchant suggested that minute asperities exist at the chip-tool interface and the fluid is drawn into the interface by the capillary action of the interlocking network of these surface asperities.

There are three possible directions through which cutting fluid could be applied as shown in Fig. 2.47.
(i) On the back of the chip
(ii) Along the rake crevice between the chip and rake face of the tool
(iii) Along the clearance crevice between the finished work surface and clearance face of the tool

Applying the cutting fluid on the back is the most convenient. But applying through the clearance crevice offers the best possible opportunity for the cutting fluid to enter into the chip-tool interface.

In each of these three locations, the cutting fluid can be applied by any of the following methods.

(i) Flooding
(ii) Jet application
(iii) Mist application

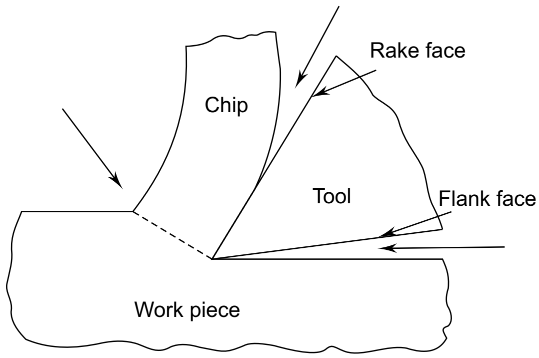


Fig. 2.47 *The directions along which cutting fluid could be applied in metal cutting*

In flooding a high volume flow of the cutting fluid is generally applied on the back of the chip. This cutting fluid floods the entire machining area and thus takes away a large amount of heat generated in the process. The used fluid is then collected in the chip pan and returned by gravity to the sump of the pump. Effectiveness of flooding depends to a great extent on the direction in which the cutting fluid is directed to the machining zone. Enough fluid should be able to enter the machining area without splashing. For this purpose a number of nozzles may be used in place of one to effect a proper distribution of the cutting fluid.

In jet application, the cutting fluid, which may be either a liquid or a gas, is applied in the form of a fine jet under pressure (up to 4 MPa). This process has been found to allow the penetration of cutting fluid into the chip-tool interface, better than flooding. Consequently better cutting performance and surface finish was observed with jet application.

In mist application, very small droplets of the cutting fluid are dispersed in a gas medium, generally air, and the mixture is applied at the cutting zone, through the clearance crevice. This method combines the attractive properties of gases and liquids. The advantages claimed for the mist application are:

(i) Large surface to volume ratio for each drop provides the possibility of rapid vaporisation, which is an important step that must precede penetration of the chip-tool interface.
(ii) The small size of the particles improves the penetrating ability of the cutting fluids.
(iii) The consumption of the cutting fluid is much less of the order of 300 ml/hour.
(iv) There is neither reclamation of the cutting fluid from the chips nor frequent cleaning of the sumps.
(v) The compressed air in the mist helps keep the chips away from the cutting zone, thus allowing the operator to follow the machining layouts easily.
(vi) No spillage of the cutting fluid, thus keeping the machine, the operator and the surroundings clean.

It has been reported that mist application gives generally low flank wear (Fig. 2.48) and better surface finish. Its effectiveness increases at higher feed rates.

Types of Cutting Fluids

There are three basic types of cutting fluids used in metal cutting. They are:

Water based emulsions: Pure water is by far the best cutting fluid available because of its highest heat carrying (high specific heat) capacity. Besides this it is cheap and easily available. Its low viscosity makes it flow at high rates through the cutting fluid system and also penetrates the cutting zone. However, water corrodes the work material very quickly, particularly at high temperatures prevalent in the cutting zone as well as the machine tool parts on which it is likely to spill.

Hence other materials would be added to water to improve its wetting characteristics, rust inhibitors and any other additives to improve lubrication characteristics. These are also called as water soluble oils. The concentrated oil is normally diluted in water to any desired concentration, such as 30:1 to 80:1.

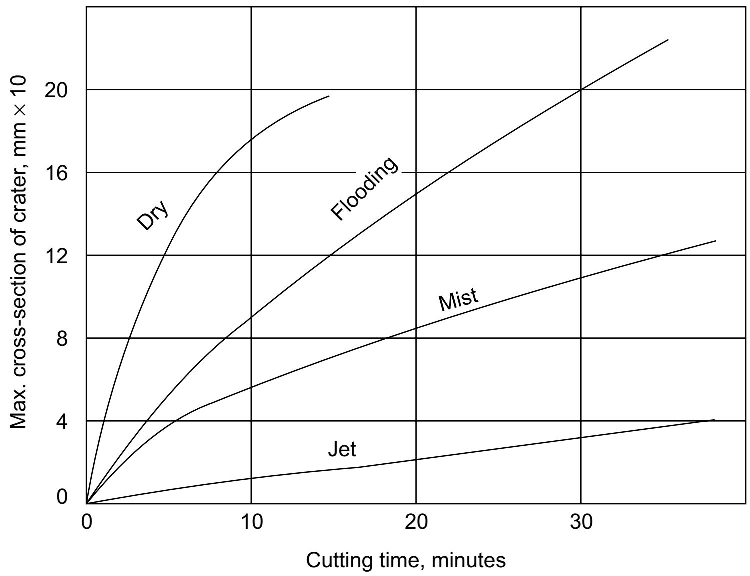


Fig. 2.48 *Comparative performances of cutting fluid application methods*

Straight Mineral oils: These are the pure mineral oils without any additives. Their main function is lubrication and rust prevention. These are chemically stable and lower in cost. However, their effectiveness as cutting fluids is limited and therefore would be used for light duty application only.

Mineral oils with additives (Neat oils): This is by far the largest variety of cutting fluids available commercially. A number of additives have been developed which when added to the mineral oils would produce the desirable characteristics for the different machining situations. Many difficult to machine situations would be helped by the use of these cutting fluids. These are generally termed as neat oils.

The additives generally improve the load carrying capacity as well as chemical activity. Fatty oils are generally used for adding the load carrying properties. Other class of additives termed as EP (Extreme Pressure) additives are used for more difficult to machine situations. These EP agents come into effect whenever minute high-spots on the mating surfaces break through the oil film and rub together to setup localised high temperature spots. This high temperature causes the EP additives to react with the adjacent metals and create an anti-welding layer of solid lubricant precisely where it is required. The layer is continuously broken by the severe rubbing action between the chip and the tool.

EP additives are basically chlorine, sulphur or a combination of both of them. As a result, the anti-welding compounds formed in the cutting zone are iron chloride and iron sulphide, both of which have very low shear strengths.

Cutting Fluid Selection

The selection of cutting fluid for a given application requires the examination of a number of parameters such as

(i) work piece material
(ii) machining operation
(iii) cutting tool material, and
(iv) other ancillary factors

Table 2.14 gives some selection guidelines based on the work material characteristics.

Table 2.14 Selection of cutting fluid based on work material

Material	Characteristics	Cutting Fluid to be Used
Grey Cast iron	Grey cast iron could be machined dry since the graphite flakes act as solid lubricants in cutting.	Soluble oils and thinner neat oils are satisfactory for flushing swarf and metal dust.
Copper alloys	Better machinability. Some could be machined dry.	Water based fluids could be conveniently used. Tougher alloys a neat oil blended with fatty or inactive EP additive is used.
Aluminium alloys	They are generally ductile and can be machined dry. However in combination with steel they have a high friction coefficient. generally BUE forms on the tool and prevents the chip flowing smoothly away from the work.	It is necessary to have the tool surface highly polished. Generally a soluble oil to which an oiliness agent has been added. For more difficult application straight neat oil or fatty oil or kerosene could be used.
Mild steel and low to medium carbon steels	Easiest to machine. Low carbon steels have lower tensile strength and hence may create problems because of their easy tearing.	Milky type soluble oil or mild EP neat cutting oil could be used.
High carbon steels	They are tough and pressures and temperatures in the cutting zone are higher.	EP cutting oil is used. In some applications milky soluble can be used.
Alloy steels	Particularly with chromium and nickel, the steels are tough. They are similar to high carbon steels.	EP cutting oil is used. In some applications milky soluble can be used.
Stainless steels and heat resistant alloys	They have great toughness, corrosion resistance and pronounced work hardening. Very difficult to machine. Because of their toughness they create very high pressure in the work area.	Use high performance neat oils with high concentration of chlorinated additives. Sulphur additives are to be avoided. Some high performance EP soluble oils may sometimes be useful.

Table 2.15 gives some selection guidelines based on the tool material characteristics.

Table 2.15 Selection of cutting fluid based on tool material

Tool Material	Characteristics	Cutting Fluid Requirements
High carbon steel	Not widely used. They should be well cooled.	Water based coolants are generally used
High speed steels	Have better hot hardness characteristics	For general machining water based cutting fluids can be used. For heavy duty work EP neat oils are preferable.
Nonferrous materials	During a cut they should never be overcooled or subjected to intermittent cooling because they are brittle and are likely to suffer thermal shock.	Neat cutting oils are the most suitable choice for most applications.
Carbides, Ceramics and diamond	These are used for very high speeds. Hence the requirement is to reduce the large amount of heat produced to reduce the thermal distortion of work piece.	Water based coolant is recommended. In low speed applications EP based oils could reduce the problem of adhesion of chip with tool.

2.13 EMPIRICAL AND ANALYTICAL DETERMINATION OF CUTTING FORCES

It has been shown previously through Merchant's analysis the relationships between the various parameters in metal cutting mechanics. However, the experience is that though these are able to give qualitative variability, quantitative predictions cannot be made from such analysis. Hence a large amount of literature is devoted to the empirical research, which allows the various parameters to be determined from empirical relations.

Cutting force is one of the major concerns in metal cutting. Thus considerable interest and a large number of relationships have been derived from empirical and semi-empirical approaches in the past. As per the general oblique cutting case there are three components of the cutting force. They are main thrust force, F_z, radial cutting force, F_r, and axial cutting force, F_a.

A major presentation of empirical aspects has been mentioned by Kronenberg. He suggests that the cutting forces be varied in direct proportion to the chip cross sectional area, and thus arrives at equations and gives the constants for some of the more generally used materials.

Another approach is the semi analytical approach of Nakayama and Arai which is based on the assumption that "the conventional cutting can be approximated to the orthogonal cutting when the ratio (feed/depth of cut) and (corner radius/depth of cut) are small". They give the following equations:

$$F_t = \tau f d (\cot \varphi + \tan \beta) \tag{65}$$

$$F_r = \tau f d (\cot \varphi \tan \beta - 1) \sin (C_s + \nu) \tag{66}$$

$$F_a = \tau f d (\cot \varphi \tan \beta - 1) \cos (C_s + \nu) \tag{67}$$

where ν is the chip flow angle. The various parameters of the cutting process are related with the help of empirical equations based on wide ranging experimental data as follows:

$$\beta = \xi_o - k_1 \alpha \tag{68}$$

$$\varphi = \varphi_o + k_2 \alpha - \frac{A}{\sqrt{V f \cos C_s}} \tag{69}$$

$$\nu = \tan^{-1} \frac{r + \frac{f}{2}}{d} \tag{70}$$

The constants used in the above equations are given in Table 2.16.

Table 2.16

	C45 Steel	C25 Steel	Low Alloy Steel	Cast Iron
k_1	0.25	0.25	0.25	0.25
k_2	0.20	0.30	0.33	0.30
ϕ_o	34.0	28.5	35.0	28.5
ξ_o	52.0	52.0	52.0	52.0
τ, N/mm^2	706.0	588.0	715.0	392.0
A*	0.20	0.20	0.20	0.10

* for using f in m, and V in m/s

2.14 ECONOMICS

The ultimate objective of the manufacturing engineer is to produce the objects at the most economical cost. To do this he should be able to analyse the machining process for all the possible costs, so that he is able to optimise the process to get the minimum possible costs, after satisfying all the requirements.

The various costs associated with machining process are:

(i) the manpower cost, C_l which is measured in ₹ per unit time, generally hours that operator is employed

(ii) the machine tool operating (overhead) cost, C_m which includes machine depreciation, and other costs associated with the running of the machine tool such as power consumed, maintenance overheads, consumables such as oils, etc. This may also include the other overhead costs, which takes care of all the fixed overheads such as buildings, land and administrative overheads.

Combining the above two costs we can have an overall overhead cost as, C_o

$$C_o = C_m + C_l \tag{71}$$

(iii) the job handling cost, which arises because of the time spent in loading and unloading of the job, during which time the machine tool is kept idle, and also requires the operator to attend to the job. It is also possible that some special equipment such as crane, etc. may be used for heavy jobs.

(iv) the tool cost, C_t which is the cost of the cutting tool for the given operation.

The three optimisation criterion that are generally considered are:

(i) Minimisation of the machining cost,

(ii) Maximizing the production rate, and

(iii) Maximizing the profit rate.

Of the three criteria, the profit rate criterion requires more information in terms of various costs, which may not always be available to the process planning department. Hence the other two are more practical in terms of actual application. Later, practical examples show the effects of each of the criterion on the results obtained.

These costs could be demonstrated conveniently with the help of a simple turning operation. Let us assume turning of a bar of length, L mm and diameter D mm, with a cutting speed of V m/min, feed rate of f mm/rev, and depth of cut of d mm.

Time for machining is given by

$$T_m = \frac{L}{f\,N} \tag{72}$$

where N is the spindle RPM which is related to cutting speed by the following relation

$$V = \frac{\pi D N}{1000} \tag{73}$$

Substituting this into the previous equation, we get

$$T_m = \frac{\pi D L}{1000\, f V} \tag{74}$$

For a given job, the cost of all overheads is given by

$$C_l = C_o\,[\,p\,(t_l + t_{ul} + t_a) + t_o\,] \tag{75}$$

where t_a = tool advance and withdrawal time

t_1 = job loading time

t_{ul} = job unloading time

t_o = initial setup time of the machine for a batch of components

and p = number of parts produced per batch

For evaluating the tool cost, we have to consider initial cost, the cost of re-grinding, and the number of re-grindings possible in case of HSS tools whereas the time taken for indexing and the number of indexable edges available in case of carbide tools.

For carbide and ceramic tools with throw away tips:

$$C_e = \frac{\text{Cost of the bit}}{\text{Number of cutting edges per bit}} \tag{76}$$

For HSS and brazed carbide tools:

$$C_e = \frac{\text{Cost of tool} + \text{Regrind cost} \times \text{Number of regrinds}}{\text{Number of regrinds} + 1} \tag{77}$$

$$C_e = \frac{C_t + rC_g}{r + 1} \tag{78}$$

where Ct = initial cost of the tool, ₹

r = number of regrinds possible for one tool

C_g = cost of regrinding the tool, ₹

p_g = number of components produced between regrinds = $\frac{T}{T_m}$

where T is the useful life of the tool

$$\text{Cost of tool replacement} = \frac{p\, C_o\, t_c}{p_g} \tag{79}$$

where tc = tool change time

Thus the total cost of the tool is

$$C_2 = \frac{p}{p_g}[C_e + C_o t_c] \tag{80}$$

The machining cost, C_3 is given by

$$C_3 = C_o\, p\, T_m \tag{81}$$

Now combining all the components of costs, we get the cost per batch, C_B as

$$C_B = C_o\,[\,p\,(t_1 + t_{u1} + t_a) + t_o] + \frac{p}{p_g}[C_e + t_c\, C_o] + C_o\, p\, T_m \tag{82}$$

Similarly the cost per piece of production is given by

$$C_P = C_o\left[t_1 + t_{u1} + t_a + \frac{t_o}{p}\right] + \frac{1}{p_g}[C_e + t_c\, C_o] + C_o\, T_m \tag{83}$$

Considering the Taylor's tool life equation,

$$V\,T^n = C \tag{84}$$

The number of pieces produced per regrind is

$$p_g = \frac{T}{T_m} = \frac{1000\,V\,f\,T}{\pi\,D\,l} \tag{85}$$

From the tool life equation

$$T = \left(\frac{C}{V}\right)^{\frac{1}{n}} \tag{86}$$

Thus, for the straight turning application,

$$p_g = \frac{1000\,V\,f}{\pi\,D\,l}\left(\frac{C}{V}\right)^{\frac{1}{n}} = \frac{1000\,f\,C^{\frac{1}{n}}}{\pi\,D\,l}V^{\left(l-\frac{1}{n}\right)} \tag{87}$$

The cost per piece, C_P is given by

$$C_P = C_o\left[t_1 + t_{u1} + t_a + \frac{t_o}{p}\right] + \frac{\pi Dl\,[\mathrm{C}_e + t_c\,C_o]}{1000\,f\,C^{\frac{1}{n}}\;V^{\frac{n-1}{n}}} + \frac{C_o\,\pi\,D\,l}{1000\,fV} \tag{88}$$

For minimum cost, the above equation could be differentiated with respect to cutting speed V, considering the fact that all other conditions such as feed and depth of cut could be maintained constant, we get

$$V = C\left[\frac{C_o}{C_e + t_c\,C_o}\left(\frac{n}{1-n}\right)\right]^n \tag{89}$$

$$V^{\frac{1}{n}} = C^{\frac{1}{n}}\left[\frac{C_o}{C_e + t_c\,C_o}\right]\left[\frac{n}{1-n}\right] \tag{90}$$

$$\frac{\pi\,D\,l}{1000\,f}\left[\frac{C_e + t_c\,C_o}{C^{\frac{1}{n}}}\right] - [\,C_o V^{-2}\,] = 0 \tag{91}$$

$$\frac{\partial C_P}{\partial V} = \frac{\pi\,D\,l\,[C_e + t_c\,C_o\,]}{1000\,f\,C^{\frac{1}{n}}}\left(\frac{1-n}{n}\right)V^{\frac{1-2n}{n}} - \frac{C_o\,\pi Dl}{1000\,f}V^{-2} = 0 \tag{92}$$

From the tool life equation we can also obtain the minimum cost tool life as

$$T^n = \left[\left(t_c + \frac{C_e}{C_o}\right)\left(\frac{1-n}{n}\right)\right]^n \tag{93}$$

or

$$T = \left[t_c + \frac{C_e}{C_o}\right]\left(\frac{1-n}{n}\right) \tag{94}$$

In order to arrive at a maximum production rate we take the total time for the manufacture as

$$\text{Time} = t_l + t_{ul} + t_a + \frac{t_o}{p} + T_m + \left(\frac{T_m t_c}{T}\right) \tag{95}$$

For a single pass turning job, it can be shown that

$$\text{Time} = t_l + t_{ul} + t_a + \frac{t_o}{p} + \frac{\pi D l}{1000 Vf} + \frac{t_c \pi D l}{1000 f C^{\frac{1}{n}} V^{1-\frac{1}{n}}} \tag{96}$$

For getting the maximum production rate, the above expression for manufacturing time is differentiated w.r.t. cutting speed, V and equated to zero.

$$\frac{\partial \text{Time}}{\partial V} = -\frac{\pi D l V^{-2}}{1000 f} + \frac{t_c \pi D l}{1000 f C^{\frac{1}{n}}}\left[\frac{1-n}{n}\right] V^{\frac{1-2n}{n}} = 0 \tag{97}$$

Simplifying, we get

$$\frac{t_c}{C^{\frac{1}{n}}}\left(\frac{1-n}{n}\right) V^{\frac{1}{n}} = 1 \tag{98}$$

Simplifying, we get

$$V = C\left[\frac{n}{t_c\,(1-n)}\right]^n \tag{99}$$

$$T = \frac{t_c\,(1-n)}{n} \tag{100}$$

Example 2.4

A 600 mm long job of diameter 150 mm of AISI 4140 steel is to be turned with a depth of cut of 1.5 mm and feed rate 0.25 mm/rev. The following data is applicable for the problem:

Labour cost per hour = ₹12.00

Machine overhead per hour = ₹40.00

Grinding cost per hour = ₹15.00

Grinding machine overhead per hour = ₹50.00

Idle time = 5 minutes

The Taylor’s tool life equation is given by

$$V\,T^{0.22} = 475$$

The operation can be carried out using tungsten carbide tools either as brazed tools or throw away tools.

For brazed tools:

Initial cost = ₹60

Grinding time = 5 minutes/edge

Tool change time = 2 minutes

9 grinds per tool before salvage

For throw away tips:

Initial cost = ₹40

Tool change time = 1.5 minutes

Total cutting edges = 8

Find the optimum cutting speed, tool life and the cost of the operation for both the brazed tip and the throw away type using the following criteria:

(a) Minimum production cost, and

(b) Maximum production rate.

Comment on the results obtained.

Solution Data given:

$n = 0.22$; $C = 475$: $r = 9$;

Idle time = $t_1 + t_{ul} + t_a = 5$ minutes

Overhead cost $C_o = 12 + 40 = 52$ ₹/hour = 0.8667 ₹/minute

Brazed Tools:

Tool-grinding cost = $\dfrac{5(15 + 50)}{60} = 5.417$ ₹

Tool cost, $C_e = \dfrac{60 + 5.417 \times 9}{10} = 10.875$ ₹

Tool change time, $t_c = 2$ minutes

Minimum Cost criterion:

$$\text{Optimum cutting speed, } V = C\left[\frac{C_o}{C_e + t_c C_o}\left(\frac{n}{1-n}\right)\right]^n \tag{101}$$

$$= 475\left[\frac{0.8667}{10.875 + 20.8667}\left(\frac{0.22}{1-0.22}\right)\right]^{0.22} \tag{102}$$

$$= 199.5 \text{ m/min}$$

$$\text{Optimum tool life, } T = \left[t_c + \frac{C_e}{C_o}\right]\left[\frac{1-n}{n}\right] \tag{103}$$

$$= \left[2 + \frac{10.875}{0.8667}\right]\left[\frac{1-0.22}{0.22}\right] \tag{104}$$

$$= 51.58 \text{ minutes}$$

$$\text{Machining time, } T_m = \frac{\pi D l}{1000 f V} \tag{105}$$

$$= \frac{\pi \times 150 \times 600}{1000 \times 0.25 \times 199.5} \tag{106}$$

$$= 5.669 \text{ minutes}$$

Total time = $t_l + t_{ul} + t_a + \frac{t_o}{p} + T_m \left[1 + \frac{t_c}{T}\right]$ (107)

$$= 5 + 5.669\left[1 + \frac{2}{51.58}\right] = 10.89 \text{ minutes}$$

Cost of operation = $C_o \left(t_l + t_{ul} + t_a + \frac{t_o}{p}\right) + T_m\,[C_e + t_c C_o]$ (108)

$= 0.8667 \times 5 + 5.669\,[(10.875 + 2 \times 0.8667)]$ (109)

$= 10.63$ ₹

Maximum Production Rate criterion:

Optimum cutting speed, $V = C\left[\frac{n}{t_c\,(1-n)}\right]^n$ (110)

$$= 475\left[\frac{0.22}{2 \times (1-0.22)}\right]^{0.22} = 308.7 \text{ m/min}$$

Optimum tool life, $T = \frac{t_c\,(1-n)}{n}$ (111)

$$= \frac{2\,(1-0.22)}{0.22} = 7.091 \text{ minutes}$$

Machining time, $T_m = \frac{\pi \times 150 \times 600}{1000 \times 0.25 \times 308.7} = 3.664$ minutes

Total time = $5 + 3.664\left[1 + \frac{2}{7.091}\right] = 9.697$ minutes

Cost of operation = $C_o \left(t_l + t_{ul} + t_a + \frac{t_o}{p}\right) + T_m\,[C_e + t_c C_o]$

$= 0.8667 \times 5 + 3.664\,[(10.875 + 2 \times 0.8667)]$

$= 14.02$ ₹

Throw away Tools:

Tool cost, $C_e = \frac{40}{8} = 5$ ₹ (112)

Tool change time, $t_c = 1.5$ minutes

Minimum Cost criterion:

Optimum cutting speed, $V = 475\left[\frac{0.8667}{5 + 1.5 \times 0.8667}\left(\frac{0.22}{1-0.22}\right)\right]^{0.22}$ (113)

$= 232.4$ m/min

Optimum tool life, $T = \left[1.5 + \frac{5}{0.8667}\right]\left[\frac{1-0.22}{0.22}\right] = 25.77$ minutes

Machining time, $T_m = \dfrac{\pi \times 150 \times 600}{1000 \times 0.25 \times 232.4} = 4.866$ minutes

Total time $= 5 + 4.866\left[1 + \dfrac{1.5}{25.77}\right] = 10.149$ minutes

Cost of operation = 0.8667 × 5 + 4.866 [(5 + 1.5 × 0.8667)]
= 9.74 ₹

Maximum Production Rate criterion:

Optimum cutting speed, $V = 475\left[\dfrac{0.22}{1.5 \times (1 - 0.22)}\right]^{0.22} = 328.87$ m/min

Optimum tool life, $T = \dfrac{1.5(1 - 0.22)}{0.22} = 5.318$ minutes

Machining time, $T_m = \dfrac{\pi \times 150 \times 600}{1000 \times 0.25 \times 328.87} = 3.439$ minutes

Total time $= 5 + 3.439\left[1 + \dfrac{1.5}{5.318}\right] = 9.409$ minutes

Cost of operation = 0.8667 × 5 + 3.439 [(5 + 1.5 × 0.8667)]
= 11.388 ₹

The above results are summarised in the following table:

Parameters	Brazed Tools		Throw Away Tools	
	Minimum Cost	Maximum Production Rate	Minimum Cost	Maximum Production Rate
Cutting speed, m/min	199.5	308.7	232.4	328.87
Tool life, Minutes	51.58	7.09	25.77	5.318
Machining time, minutes	5.669	3.664	4.866	3.439
Operation time, Minutes	10.89	9.679	10.149	9.409
Cost, ₹	10.63	14.02	9.74	11.388

The results present interesting reading. For example, when the production rate is to be maximised, the cutting speed has increased to nearly the maximum value, giving rise to a very low value for the tool life. The reverse is also true. This is happening because the earlier optimisation does not have any constraints on the variables. Hence the variables go to the extreme. However, it is necessary to get more useful values so that the optimisation can be carried out using various constraints on the variables. For example, the following are some of the possible constraints. More details can be found in the literature cited at the end of this chapter.

(i) The maximum cutting power available at the machine tool spindle,
(ii) the maximum cutting force permissible,
(iii) the surface finish and the diametral tolerance to be achieved on the machined surface
(iv) the limits on speed, feed and depth of cut imposed by the machine tool and the cutting tool.
(v) maximum permissible cutting temperature,
(vi) maximum permissible chatter,

(vii) maximum permissible work piece static and dynamic instability,
(viii) tool life and tool fracture.

Some of the possible constraints as mentioned above have been explained further in terms of the possible formulation in case of turning:

Power constraint: The power constraint assumes significance only in case of rough machining and hence can be ignored in the case of finish machining. The power consumed, P during a turning operation can be expressed as,

$$P = \frac{V^{a_p} f^{b_p} d^{c_p} k_c}{60 \times 1000 \times \eta}$$

This value of cutting power should not exceed the maximum power, P_{max} available on the machine tool. In the above expression, the value of specific cutting force is constant for a particular work material. Hence, the expression for this constraint can be written as

$$C_P V^{a_p} f^{b_p} d^{c_p} \le P_{max}, \qquad C_p = \frac{k_c}{60 \times 1000 \times \eta}$$

Surface finish constraint: This constraint limits the maximum feed that can be used to attain the required surface finish on the machined feature. This constraint becomes active during finish turning. The expression for CLA value of the geometric surface finish obtained during turning operation with a tool of nose radius '*r*' is given as

$$SF = 1000 \times \frac{f^2}{18\sqrt{3}r}$$

Based on the surface finish, SF_{max} specified on the turned surface, the constraint on feed can be expressed as

$$C_s f^2 \le SF_{max}, \qquad C_s = \frac{1000}{18\sqrt{3}r}$$

Maximum and minimum speed constraints: Usually, these constraints are imposed by the machine tool. However, in the case of carbide and ceramic tools, certain minimum cutting speeds need to be maintained to avoid the failure of these cutting tools due to BUE formation or micro-chipping. Hence, the minimum speed constraint is determined as the larger of the values of minimum cutting speed from machine tool and minimum cutting speed due to cutting tool (V_{min}). Thus, the speed constraint can be expressed as

$$\max\left\{\frac{\pi D N_{min}}{1000}, V_{min}\right\} \le V \le \frac{\pi D N_{max}}{1000}$$

Maximum and minimum feed constraints: The maximum feed permissible for the cutting tool (f_{tmax}) is set as per the recommendations given by the cutting tool manufacturer. As per these recommendations, f_{tmax} should not exceed 0.4 – 0.5 times the insert corner radius for triangular inserts and 0.6 – 0.7 times the corner nose radius for square inserts. The minimum feed value is set to the smallest feed rate available on the machine tool. This constraint prevents the feed rate during finishing operation from becoming too small. Thus the feed constraints can be expressed as,

$$f_{min} \le f \le \min\{f_{max}, f_{tmax}\}$$

Maximum and minimum depths of cut constraints: The maximum depth of cut constraint has relevance in roughing operations, while the minimum depth of cut constraint should be considered in finishing operations. The maximum limit on depth of cut due to cutting tool ($d_{t\,max}$) is set to half the cutting edge length for inserts with included angle of 55° or 60° and for inserts with included angle of 80°–100°, d_{tmax} is set to two-thirds of the cutting edge length. Minimum limit on depth of cut (d_{min}) is set to 0.5mm for finishing operations and 1mm or depth of pocket, whichever is lower, for roughing operations. The depth of cut constraints can thus be expressed as,

$$d_{min} \leq d \leq d_{tmax}$$

It is possible to develop mathematical models using the above constraints and the extended Taylor's tool life equation to get more realistic and practical results for optimised machining process parameters.

SUMMARY

Metal cutting forms the basis for all the machine tools. In this chapter an overview of metal cutting is presented, which explains the underlying principles used in all the metal cutting machine tools.

- A cutting tool performance is significantly affected by its geometric parameters such as rake angle and clearance angle.
- Cutting takes place due to the shear ahead of the cutting edge, and different types of chips such as discontinuous, continuous and continuous with BUE are formed during this process.
- Merchant's thin shear plane model predicts the performance of orthogonal cutting and based on this theory it is possible to predict the performance of the process.
- Shear angle plays a very important role in metal cutting, and a lot of effort was put to develop a simple equation for it.
- Cutting tool geometry specification as standardised by ASA is widely practiced because of its simplicity. Improved specifications by ISO helps in clarifying the concepts better.
- Dynamometers are used to measure the cutting force utilizing some form of transducing the cutting force into some easily measurable quantity.
- High speed steel and cemented carbides are the most widely used cutting tool materials. Ceramics, CBN and diamond tools are selectively used for specific applications.
- Cutting temperature is affected to a great extent by cutting speed and to a lesser extent by the other parameters.
- Tool wear is predominantly of two types: crater wear and flank wear.
- There are a number of tool life criteria that are used in metal cutting. Tool life is controlled in a majority of situations by the flank wear due to processing requirements.
- Surface finish is an important requirement that is controlled by the feed rate used and the tool nose radius.
- Different types of cutting fluids such as water soluble, mineral oils and mineral oils with additives are used in metal cutting operations.
- There are a number of ways the cutting fluids can be applied.
- It is possible to optimise a metal cutting operation based on a number of constraints that need to be satisfied along with the necessary optimal criterion.

Questions

2.1 Explain with a neat sketch what you understand by ‘Orthogonal Cutting’ and its relevance in metal cutting study. How can orthogonal cutting be realised in practice?

2.2 What are the conditions that would allow a continuous chip to be formed in metal cutting?

2.3 From the machining performance view point which type of chip is preferred? Explain your answer with suitable justification. Also show the conditions which favour such a chip formation.

2.4 Explain with a neat sketch what you understand by ‘Orthogonal Cutting’ and its importance? How can orthogonal cutting be realised in practice.

2.5 In an orthogonal cutting tool what are the important angles that are to be maintained? For each of the angle explain its influence on the machining performance.

2.6 How is metal removed in metal cutting? Explain the process by giving any simple model to explain the metal removal process.

2.7 What is meant by built-up-edge (BUE)? With a neat sketch explain the formation of a BUE. Explain the conditions, which promote the growth of BUE along with its consequences.

2.8 Derive the expression for shear angle in orthogonal cutting in terms of rake angle and chip thickness ratio.

2.9 Show schematically the Merchant's force circle in orthogonal cutting. Derive the equations for shear and friction forces in terms of the material properties and cutting process parameters. Give in detail the assumptions made while arriving at the final equations.

2.10 Derive the expression for the main cutting force in orthogonal cutting in terms of work material properties and cutting process parameters.

2.11 Derive the expression for the specific cutting energy in orthogonal cutting in terms of the shear angle and the mean shear strength of the work material. Assume Merchant's minimum energy principle holds good and that rake angle is zero.

2.12 A single point cutting tool has a zero rake angle and 2 degree clearance angle. By what percentage would the life of the tool between re-grinds be increased if a clearance angle of 8 degree was provided? State any assumptions made.

2.13 Derive the shear angle relationship based on Merchant's minimum energy principle. Discuss its validity from the experimental observations.

2.14 Discuss the importance of shear angle from the standpoint of metal cutting performance. What factors influence its value?

2.15 Discuss the nature of friction found in metal cutting. How do you explain the large value of apparent friction coefficient found in metal cutting?

2.16 What are the various forms of wear found in cutting tools? Show with a neat sketch.

2.17 How does the cutting process parameters affect the cutting tool wear in single point tools?

2.18 Show diagrammatically the variation of flank wear of cutting tool with time and explain its importance from the tool life point of view.

2.19 What are the types of cutting tool wear patterns observed in single point tools? How do they affect the metal cutting performance?

2.20 How do you define Tool Life? Explain the parameters that control the tool life of a single point cutting tool.

2.21 Which are the suitable tool failure criteria that are generally practised in industry? Explain your answers with examples.

2.22 What is the most generally used method for measuring average chip tool interface temperature? Explain its principle with a neat sketch.

2.23 What are the various methods available for measuring the cutting tool temperature? Explain their applications and disadvantages.

2.24 How are the cast cobalt alloys differ from cemented carbides in terms of material composition and machining performance?

2.25 What are the desirable characteristics of a cutting tool material? Explain how these are satisfied in the case of high speed steel tools.

2.26 How are the cemented carbides classified by ISO? Explain the general applications of each category.

2.27 How are the properties of cemented carbides affected by grain size, composition and amount of bonding material? Explain them with reference to metal cutting application.

2.28 How do you expect the coatings on carbides improve the machining performance?

2.29 Explain the advantages of coated carbides over the uncoated carbides. Name any three materials used for coatings.

2.30 How do you compare cutting tool made of CBN with that made by cemented carbides? Your answer should be based on tool material composition, structure, and cutting performance.

2.31 Why are synthetic diamonds preferred to natural diamonds as cutting tools?

2.32 What are the situations where diamonds are used as cutting tools?

2.33 What are the advantages and disadvantages of ceramics as cutting tool materials?

2.34 How are the ceramic tools made?

2.35 What are the main applications of cutting fluids?

2.36 What are the mechanisms that are generally responsible for the effectiveness of cutting fluid in improving machining performance?

2.37 What are the points from which a cutting fluid may be applied in a machining operation? Explain their relative merits.

2.38 Explain the basis for the selection of a specific cutting fluid for a given application. Take the example of turning, milling and grinding, and suggest the type of cutting fluid used.

2.39 What is the mechanism suggested for the cutting fluid to reach the chip-tool interface in spite of the adverse conditions of high pressure existing there? Explain with a neat sketch.

2.40 What is meant by machinability? Explain the method of representing the machinability.

2.41 What are the factors that control surface finish in turning? How do you select the cutting process parameters for finish turning?

2.42 Derive the relationship for the minimum cost cutting speed in single point turning of a cylindrical work piece. State the assumptions made.

2.43 Derive the relationship for the maximum production rate cutting speed in single point turning of a cylindrical work piece. State the assumptions made.

2.44 Derive the relationship for the minimum cost tool life from first principles in single point turning of a cylindrical work piece. State the assumptions made.

2.45 Derive the relationship for the maximum production rate tool life in single point turning of a cylindrical work piece. State the assumptions made.

2.46 Discuss the forms of tool life equations generally used with their applicability.

2.47 Suggest the grade of carbide you would use for i) rough milling of medium carbon steel, ii) finish turning of white cast iron rolls. Justify your recommendation.

2.48 Find the tool life for the minimum cost per piece in single pass turning given:

a = machine rate including labour and overhead,

b = tool cost per cutting edge, and

t = tool change time,

n = exponent in Taylor's tool life equation.

C = constant in Taylor's tool life equation.

Make any valid assumptions with justification.

2.49 Give a comparative evaluation of the various cutting tool materials.

2.50 What are the desirable properties of a cutting tool material?

2.51 What are the locations where heat is produced in an orthogonal cutting tool? Show their approximate percentages.

2.52 Explain how effective tungsten carbide is as a cutting tool material in comparison to the other cutting tool materials. What are the improvements caused by the coated carbides?

2.53 Explain the tool nomenclature as used in ASA and ORS systems.

2.54 Compare ASA and ORS systems of tool nomenclature.

2.55 Give the significance of various tool angles with a neat sketch.

2.56 What do you understand by the name tool signature? Give a typical tool signature in ASA system.

2.57 Explain the principles used in designing a metal cutting dynamometer.

2.58 Give any one type of metal cutting dynamometer design you are familiar with and explain its function.

Problems

2.1 A 100 mm bar is turned by means of a tool with a rake angle of 15° orthogonally. Depth of cut is 5 mm while the feed rate is 0.25 mm/rev. If the mean length of a cut chip representing one rotation of the work piece is 90.5 mm, find the shear plane angle. [17.63°]

2.2 During a metal cutting test under orthogonal conditions in a lathe with a tool of rake angle 20°, with a depth of cut of 3 mm and feed rate of 0.38 mm/rev, the following data is recorded.

Average chip thickness = 0.89 mm

Horizontal component of the cutting force = 1600 N

Vertical component of the cutting force = 2340 N

Calculate the following:

(a) coefficient of friction at the chip tool interface

(b) shear plane angle, and

(c) shear stress at the shear plane. [3.905, 25.17°, 168.95 MPa]

2.3 The following data was obtained from an orthogonal cutting test.

Rake angle = 20° Depth of cut = 6 mm

Feed rate = 0.25 mm/rev

Chip length before cutting = 29.4 mm

Chip length after cutting = 12.9 mm

Vertical cutting force = 1050 N

Horizontal cutting force = 630 N

Using Merchant's analysis, calculate
(a) magnitude of resultant force,
(b) shear plane angle, and
(c) friction force and friction angle. [1224.5 N, 25.88°, 1174 N, 78.3°]

2.4 In an orthogonal cutting of a steel component with carbide tool, the following data was obtained:

Tool rake angle = 10° Chip width = 6 mm

Uncut chip thickness = 0.10 mm

Chip thickness ratio = 0.33

Horizontal cutting force = 1290 N

Vertical cutting force = 1650 N

Sketch the force diagram and calculate the mean shear stress on the shear plane. [375.35 MPa]

2.5 In an orthogonal cutting test with a tool of rake angle 10°, the following observations were made:

Chip thickness ratio = 0.37

Horizontal component of the cutting force = 1000 N

Vertical component of the cutting force = 1500 N

From the Merchant's theory, calculate the various components of the cutting forces and the coefficient of friction at the chip tool interface. [1751.56 N, 426.66 N, 374.51 N, 1763.45 N, 4.105, 21.7°]

2.6 SAE 113 cold rolled steel is orthogonally cut on a lathe with a HSS tool having a rake angle of 20°. The following data is obtained during the test.

Width of cut = 5 mm Cutting speed = 8.5 m/min

Uncut chip thickness = 0.25 mm Cutting ratio = 0.351

Vertical cutting force = 1030 N

Horizontal cutting force = 550 N

Calculate the friction, shear and total energies consumed during the metal cutting process.
[1225.73 J/min, 3449.3 J/min, 4675 J/min]

2.7 Mild steel tube is being cut with a carbide tool having a rake angle 5°, orthogonally at a cutting speed of 250 m/min. Feed used is 0.21 mm/rev while depth of cut is 2 mm. The cutting ratio is 0.31. The vertical cutting force is 1030 N and the horizontal cutting force is 550 N. Calculate from the merchants theory, the various work done in metal cutting and shear stress.

[54.256 kJ/min, 83.235 kJ/min, 137.500 kJ/min, 153.17 MPa]

2.8 In an orthogonal cutting of C35 steel with HSS tool, the following conditions were noted.

Width of cut = 1.2 mm; Rake angle = 15° Cutting ratio = 0.35

Cutting force = 800 N; Thrust force = 800 N

Calculate the shear angle and other force components.

[17.59°, 520.8 N, 1004.4 N, 979.8 N, 565.7 N]

2.9 In an orthogonal cutting test on an alloy of aluminium, the following values were known or obtained from experimental data.

Rake angle = 20°	Thrust force = 340 N
Uncut chip thickness = 0.125 mm	Width of cut = 3.75 mm
Cutting speed = 0.5 m/s	Chip thickness = 0.51 mm

If the force in the cutting speed direction is 1.3 times that of the thrust force, calculate the average yield shear stress of the work material. State the assumptions made and derive any expression required for the calculations from the first principles. [115.42 MPa]

2.10 In an orthogonal cutting test, the following data were obtained.

Uncut chip thickness = 0.2 mm	Chip thickness = 0.5 mm
Chip width = 2.0 mm	Cutting force = 900 N
Thrust force = 600 N	Rake angle = 8°
Cutting speed = 0.6 m/s	

Calculate the average friction angle at the chip tool interface, shear stress and energies consumed in friction and shear. [64.3°, 198.06 MPa, 125.6 J/s, 233.93 J/s]

2.11 The following data was obtained in orthogonal cutting of mild steel.

Uncut chip thickness = 0.25 mm	Chip thickness = 0.75 mm
Chip width = 2.50 mm	Cutting force = 900 N
Thrust force = 450 N	Rake angle = 0°

Calculate a) the angle of friction along the tool rake face and b) shear strength of the work material. [63.43°, 138 MPa]

2.12 In an orthogonal cutting of C35 steel with HSS tool, the following conditions were noted.

Uncut chip thickness = 0.25 mm	Chip thickness = 1.0 mm
Width of cut = 2.5 mm;	Rake angle = –5°
Cutting force = 900 N;	Thrust force = 900 N

Calculate the shear angle, mean shear strength of the work material and other force components. [13.7°, 250.55 MPa, 818.1 N, 975 N, 661.2 N, 1087.5 N]

2.13 Show that in metal cutting, when the rake angle is zero in orthogonal cutting, the ratio of the shear strength of work material, τ, to the specific cutting energy, u_s, is given by

$$\frac{\tau_s}{u_s} = \frac{r(1-\mu r)}{1+r^2}$$

where μ is the coefficient of friction in the chip tool interface while r is the cutting ratio.

2.14 In an orthogonal cutting, if the frictional force, F on the tool rake face is equal to $K\,\tau A$, show that the following relationship between the mean coefficient of friction, μ and the shear angle, ϕ, is valid.

$$\mu = \frac{K \cos^2(\phi-\alpha)}{K \sin(\phi-\alpha)\cos(\phi-\alpha)+1}$$

where K is a constant, A is the area of cross section of the chip and α is rake angle.

2.15 If the shear angle relationship given by Lee and Shaffer $\left[\phi = \frac{\pi}{4} + (\alpha - \beta)\right]$ is valid, show that the specific cutting energy, u_s is given by

$$u_s = \tau(1 + \cot \phi)$$

where τ is the mean shear strength of the work material.

2.16 In an orthogonal cutting test the following values were known or obtained from experimentation.

Rake angle = 20°   Thrust force = 500 N

Uncut chip thickness = 0.14 mm   Width of cut = 5 mm

Cutting speed = 2 m/s   Chip thickness = 0.70 mm

If the force in the cutting speed direction is two times that of the thrust force, calculate the average yield shear stress of the work material. [16.52 MPa]

2.17 During a metal cutting test under orthogonal conditions in a lathe with a tool of rake angle 20°, with a depth of cut of 3 mm and feed rate of 0.38 mm/rev, the following data is recorded.

Average chip thickness = 0.89 mm

Horizontal component of the cutting force = 1000 N

Vertical component of the cutting force = 1840 N

Calculate the following:

(a) coefficient of friction at the chip tool interface
(b) shear plane angle, and (c) shear stress at the shear plane. [6.67, 25.2°, 45.35 MPa]

2.18 The following data was obtained from an orthogonal cutting test.

Rake angle = 20°   Depth of cut = 6 mm

Feed rate = 0.25 mm/rev   Cutting speed = 0.6 m/s

Chip length before cutting = 29.4 mm

Vertical cutting force = 1050 N

Horizontal cutting force = 630 N

Chip length after cutting = 12.9 mm

Using Merchant's analysis, calculate

(a) magnitude of resultant force,
(b) shear plane angle,
(c) friction force and friction angle, and
(d) various energies consumed. [1224.5 N, 25.88°, 1202 N, 79º, 61.5 J/s, 316.487 J/s]

2.19 Show that in an orthogonal cutting, when rake angle is 45° and the thrust and cutting forces are equal, the shear angle, ϕ is given by

$$\phi = \frac{\cos^{-1}\left[2 - \frac{2}{\gamma} - \frac{F_s^2}{F_H^2}\right]}{2}$$

where γ is the shear strain

F_s force along the shear plane

F_H cutting force in the direction of the cutting speed

State any relevant assumptions made.

2.20 In an orthogonal cutting test, the following data were obtained.

Uncut chip thickness = 0.2 mm Chip thickness = 0.5 mm

Chip width = 2.0 mm Cutting force = 900 N

Thrust force = 600 N Rake angle = 8°

Cutting speed = 0.6 m/s

Calculate average friction angle at the chip tool interface. [64.3º]

2.21 For a production turning operation, past records have shown that the tool life varies with cutting speed as shown in the following table:

Cutting speed, V, m/s	Tool Life, T, min
2.08	110
2.54	37

Estimate the tool life for this operation at a speed of 2.3 m/s. Outline all the assumptions used to obtain this estimate. [63.61 minutes]

2.22 An orthogonal cutting process is being carried out with the following process parameters:

Uncut chip thickness = 0.15 mm Cutting speed = 120 m/min

Rake angle = 10° Width of cut = 6 mm

The observed values of the other parameters are as follows:

Chip thickness = 0.225 mm Horizontal force = 550 N

Vertical force = 1230 N.

Calculate the percentage of the total energy that goes into overcoming the friction at the chip-tool interface. [83.87%]

2.23 If the Taylor's tool life constants for a given operation are specified as $n = 0.5$ and $C = 400$, what is the percentage increase in tool life when the cutting speed is reduced by half? [400]

2.24 In an orthogonal cutting operation, given the rake angle is 10°, what is the percentage change in the chip thickness when the friction angle changes from 30 to 50°? Do not use the shear angle relationship derived by the Merchant's minimum energy principle. [45%]

2.25 Using the Merchant's cutting mechanics analysis derive a relationship between shear energy and frictional energy in terms of rake angle, shear angle and friction angle. $\left[\frac{W_F}{W_S} = \frac{\sin\beta\tan\alpha}{\cos(\varphi+\beta-\alpha)}\right]$

2.26 The tool life of a high speed steel (HSS) tool and carbide tool have the same tool life of 60 minutes at a cutting speed of 75 m/min. The exponent of tool life in Taylor's equation (n) is 0.15 for HSS while it is 0.2 for carbide. Compare the life of the two tools at a speed of 90 m/min.

[17.28 minutes, 24.12 minutes]

2.27 An automatic lathe is to be used to machine brass component 75 mm long and 50 mm in diameter with a depth of cut of 1.25 mm and feed rate of 0.2 mm/rev. The lathe has 3 kW motor with drive efficiency of 70%. Select the cutting speed to give the minimum machining cost under the following conditions.

Operating cost of the lathe = ₹75 per hour

Regrinding cost of cutting edge = ₹15 per edge

Time to load and unload a component = 15 seconds

Tool change time = 5 minutes

Tool life constants $n = 0.2$; $C = 400$. [172 m/min]

2.28 In a normal turning operation the tool life varies with cutting speed as shown in the following table:

Cutting speed, V, m/min	Tool Life, T, min
25	30
70	2

Estimate the tool life for this operation at a speed of 60 m/min. [2.99 minutes]

2.29 A carbide cutting tool has tool life exponent $n = 0.27$. It gives a tool life of 60 minutes while machining a mild steel work piece at a cutting speed of 120 m/min. Compute the tool life if it is to be cut at a 20% higher cutting speed. Also what is the cutting speed if the tool life is to be doubled?

[30.39 minutes, 99.387 m/min]

2.30 Free cutting steel work pieces of 200 mm long and 100 mm in diameter are to be turned on a lathe using a feed of 0.15 mm/rev and a depth of cut of 2 mm. It is possible to use brazed and throw away type cemented carbide tools for the operation. The overhead cost is ₹80 per hour, while the tool life constants are $n = 0.25$ and $C = 200$. Compare the minimum cost and maximum productivity times and costs of these with the following data:

Brazed tools	Throw away tools
Tool cost = ₹90	₹30
No of regrinds = 10	No. of edges = 4
Regrinding cost = ₹15	
Tool change time = 3 min	1 min

2.31 In a metal cutting experimentation, the tool life were found to vary with cutting speed in the following manner:

Cutting speed, V, m/min	Tool Life, T, min
100	120
130	50

Derive the Taylor's tool life equation for this operation and estimate the tool life at a speed of 2.5 m/s. Also estimate the cutting speed for a tool life of 80 minutes. [31.1867 minutes, 113.072 m/min]

2.32 A bar of 150 mm long and 60 mm in diameter is to be turned with a feed rate of 0.25 mm/rev. The machine and labour overhead is ₹150 per hour. The tool cost is ₹15 per edge and the tool changing time is 1 minute. Two different work materials A and B satisfy the requirements of the operation whose tool life constants are given below:

Material	n	C
A	0.15	125
B	0.16	150

Compute the economical work material for this operation. [Material B]

Multiple Choice Questions

2.1 Rake angle of a cutting tool can be defined as
(a) Angle between rake face and flank face of a cutting tool
(b) Angle between rake face of the cutting tool and normal to the machined surface
(c) Angle between flank face of the cutting tool and normal to the machined surface
(d) Angle between flank face of the cutting tool and the machined surface

2.2 Clearance angle of a cutting tool can be defined as
 (a) Angle between rake face and flank face of a cutting tool
 (b) Angle between rake face of the cutting tool and normal to the machined surface
 (c) Angle between flank face of the cutting tool and normal to the machined surface
 (d) Angle between flank face of the cutting tool and the machined surface

2.3 Continuous chip can form during the cutting of
 (a) Ductile materials
 (b) Brittle materials
 (c) Any material at low cutting speeds
 (d) Any metal at high depths of cut

2.4 Discontinuous chip can form during the cutting of
 (a) Ductile materials
 (b) Brittle materials
 (c) Any material at high cutting speeds
 (d) Any metal at low depths of cut

2.5 Built-up-edge can form during the cutting of
 (a) Soft materials at high cutting speeds
 (b) Brittle materials at low cutting speeds
 (c) Hard material at low cutting speeds
 (d) Soft materials at low cutting speeds

2.6 An important assumption of Merchant's cutting force analysis is
 (a) The cutting tool is perfectly sharp
 (b) Rake angle of the cutting tool is positive
 (c) Rake angle of the cutting tool is negative
 (d) Chip thickness is proportional to the rake angle

2.7 A positive rake angle is generally preferred for
 (a) Brittle work piece materials to reduce cutting forces
 (b) Cutting tool materials that are hard and brittle
 (c) Ductile work piece materials to reduce cutting forces
 (d) Cutting tool materials that have poor thermal conductivity

2.8 Shear angle in orthogonal cutting is
 (a) The angle between the flank face and the shear plane
 (b) The angle between the rake face and the shear plane
 (c) The angle between the flank face and the machined surface
 (d) The angle between the rake face and the machined surface

2.9 A negative rake angle is generally preferred for
 (a) Brittle work piece materials to reduce cutting forces
 (b) Cutting tool materials that are hard and brittle
 (c) Ductile work piece materials to reduce cutting forces
 (d) Cutting tool materials that have higher shock resistance

2.10 *P* grade cemented carbide cutting tool bit is used for
 (a) Ferrous material with short chips
 (b) Ferrous material with long chips
 (c) Non-ferrous material with short chips
 (d) Any non-ferrous metal

2.11 Ceramic cutting tools should be used
 (a) With cutting fluid
 (b) With low cutting speeds because of their brittleness
 (c) With very high cutting speeds
 (d) With old machine tools

2.12 Tool life criterion normally used is
 (a) Crater wear
 (b) Flank wear
 (c) Crater wear and flank wear
 (d) Crater wear and nose wear

2.13 Cutting fluids are used during the machining operation to
 (a) Cool the work piece only
 (b) Cool the cutting tool and work piece
 (c) Clean the work piece
 (d) Clean the machine tool

2.14 Water soluble cutting fluids are mainly used to
(a) Cool the cutting tool and work piece
(b) Clean the work piece
(c) Clean the machine tool
(d) Lubricate the cutting tool and work piece interface

2.15 Why is the cutting speed of 150 m/min better than 30 m/min when using cemented carbide cutting tools?
(a) There would be better shaped chips
(b) Less heat is produced at 150 m/min than at 30 m/min
(c) The higher speed is less likely to burn the edge of a carbide tool
(d) The higher speed would produce a better finish

2.16 When using low cutting speeds and negative rake tools to cut soft metals, the result is often
(a) A long, and uniform coiled chip
(b) The metal splitting ahead of the tool
(c) A built-up edge
(d) A good surface finish

2.17 Cutting tools with negative rake angle require
(a) Frequent resharpening
(b) Less frequent resharpening
(c) More horse power for a given cut
(d) Less horse power for a given cut

2.18 Effect of hardness of work material on its machinability
(a) No effect
(b) Increases machinability
(c) Decreases machinability
(d) Very little effect on machinability

2.19 Effect of rake angle of cutting tool on machinability
(a) No effect
(b) Increases machinability up to a certain limit
(c) Decreases machinability
(d) Very little effect on machinability

2.20 The cutting speed for maximum profit rate should be chosen as
(a) Below the speed for minimum cost
(b) In between the speeds for minimum cost and maximum production rate
(c) Equal to the speed for minimum cost
(d) Higher than the speed for maximum production rate

Answers to MCQs

2.1 (b)	2.2 (d)	2.3 (a)	2.4 (b)	2.5 (d)
2.6 (a)	2.7 (c)	2.8 (b)	2.9 (b)	2.10 (a)
2.11 (c)	2.12 (b)	2.13 (b)	2.14 (a)	2.15 (d)
2.16 (c)	2.17 (c)	2.18 (c)	2.19 (b)	2.20 (b)

CASE STUDY

MACHINING

In a machine shop, annealed AISI 4140 steel rods of 150 mm diameter are to be turned to 149 mm. The hardness of the material is BHN 250. The length of the rods is 100 mm. The optimum cutting speed and feed can be obtained from Machinery's Handbook (30^{th} edition, p 1071) for different cutting tool materials. For example, the optimum cutting speeds for the following tool materials as in the table are 108, 192 and 375 m/min. The feed rates are 0.43, 0.43 and 0.25 mm/rev. These speeds and feeds are valid for a tool life of 15 minutes. In regular production, it means the tool need to be replaced every 15 minutes which adds a lot of idle time on the machine and the operator involvement. From the Taylor tool life equation it is noticed that for increasing the tool life the cutting speed needs to be reduced. For carbide tools, to increase the tool life the cutting speed is to be multiplied by 0.86 for 45 minute tool life, 0.78 for 90 minute tool life, 0.71 for 180 minute tool life. The factors for coated carbides are 0.80, 0.70 and 0.61 respectively. Similarly for ceramic tools are 0.89, 0.82 and 0.76 respectively. When the cutting speed is decreased, the machining time will increase. It may be noticed that the increase in tool life is not directly proportional to the reduction in cutting speed. The machining times can therefore be recalculated for all tool lives as given in the following table:

Tool Life, min	Machining Time, min		
	Uncoated Carbides Hard Grade	Coated Carbides Hard Grade	Ceramics, Hard
15	1.01	0.57	0.50
45	0.90	0.39	0.94
90	1.07	0.49	1.11
180	1.27	0.61	1.10

The relationship between tool life and the machining time can be better understood by actually comparing the number of parts that can be completely machined before a tool change is done in the machine. The data can be calculated with the results as follows:

Tool Life, min	Number of Parts Made before Tool Change		
	Uncoated Carbides Hard Grade	Coated Carbides Hard Grade	Ceramics, Hard
15	15	26	30
45	38	63	80
90	70	111	148
180	127	193	274

Assuming the tool cost of coated carbides and ceramic tools to be 1.4 times and 1.2 times respectively, compared to uncoated carbides, work out the economics for each tool/ work piece combination. One word of caution is that the relationship of the surface finish produced based on the cutting speed and feed used as well as the rigidity of the machine for suitability of the process was not considered in the above calculations.

CHAPTER 3

Machine Tools

Objectives

Machine tools are the main engines of the manufacturing industry. This chapter tries to cover a few of the details that are common to all classes of machine tools in this book. After completing this chapter, the reader will be able to

- Understand the classification of the various machine tools used in manufacturing industries
- Find the differences between generating and forming of surfaces
- Learn various methods used to generate different types of surfaces
- Distinguish the different accuracies and surface finishes that are achievable with different machine tools
- Understand the different components of the machine tool and their functions
- Learn about the different support structures used in the machine tools
- Understand various power transmission systems that are utilized in machine tools
- Understand various actuation systems that are useful to generate the required surfaces
- Learn the different types of guideways used in the machine tools
- Understand briefly about the work holding requirements

3.1 INTRODUCTION

The earliest known machine tools are the Egyptian foot-operated lathes. The machine tools were developed essentially to allow for the introduction of accuracy in manufacturing.

A machine tool is defined as one which while holding the cutting tools would be able to remove metal from a work piece in order to generate the requisite product of given size, configuration and finish. It is different from a machine, which essentially is a means of converting the source of power from one form to the other. The machine tools are the mother machines since without them no components could be produced in their finished form. They are very old and the industrial revolution owes its success to a very great extent to them.

A machine tool is required to provide proper support to the work piece and cutting tool as well as provide motion to one or both of them, in order to generate the required shape on the work piece. The form generated depends upon the type of machine tool. The details of these will be seen in this chapter.

In the last two centuries, the machine tools have developed to a very great extent, because of new inventions. The machine tool versatility has grown to cater to the varied needs of the new inventors who made major developments. For example, James Watt's steam engine could be proven only after a satisfactory method was found to bore the engine cylinder with a boring bar by Wilkinson around 1775.

A machine tool is designed to perform certain primary functions, but the extent to which it can be exploited to perform other (secondary) functions is a measure of its flexibility. Generally the flexibility of the machine tool is increased by the use of secondary functional attachments, such as radius (spherical) turning attachment for a centre lathe. Alternatively to improve productivity, special attachments are added which also reduce the flexibility.

3.2 CLASSIFICATION OF MACHINE TOOLS

There are many ways in which the machine tools can be classified. One such classification based on the production capability and application is shown below:

(1) General purpose machine tools (GPM) are those designed to perform a variety of machining operations on wide ranging types of components. By the very nature of generalisation, the general purpose machine tool though capable of carrying out a variety of tasks, would not be suitable for large production, since the time for setting-up such operations is large. Thus the idle time on the general purpose machine tool is more and the machine utilisation is poor. The machine utilisation may be termed as the percentage of actual machining (chip generating) time to the actual time available. This is much lower for the general purpose machine tools. They may also be termed as the basic machine tools.

Further skilled operators would be required to run the general purpose machine tools. Hence their utility is in job shops (catering to small batch, large variety job production) where the requirement is versatility rather than production capability. Examples are lathe, shaper and milling machine.

(2) Production machine tools are those where a number of functions of the machine tools are automated such that the operator skill required to produce the component is reduced. Also this would help in reducing the idle time of the machine tool thus improving the machine utilisation. It is also possible that a general purpose machine tool may be converted into a production machine tool by the utilisation of jigs and fixtures for holding the work piece. These have been developed from the basic machine tools. Some examples are capstan and turret lathes, automats and multiple spindle drilling machines. The time required for setting-up for a given job is more. Also tooling design for a given job is more time consuming and expensive. Hence these machines can only be justified for large volume production.

(3) Special purpose machine tools (SPM) are those machine tools where the setting operation for the job and tools is practically eliminated and complete automation is achieved. This would greatly reduce the cycle time (the actual manufacturing time) of a component and helps in the reduction of the costs. These are used for mass manufacturing. These machine tools are expensive compared to the general purpose machines since they are specifically designed for the given application, and also restrictive in their application capabilities. Examples are cam shaft grinding machine, connecting rod twin boring machine, and piston turning lathe.

(4) Single purpose machine tools are those, which are designed specifically for doing a single operation on a class of jobs or on a single job. This is the highest amount of automation and is used for really high

rate of production. These are used specifically for one product only and thus have the least flexibility. However these do not require any manual intervention and are most cost effective. Examples are transfer lines composed of unit heads for machining any given product completely.

The application of these four types can be shown graphically in Fig. 3.1.

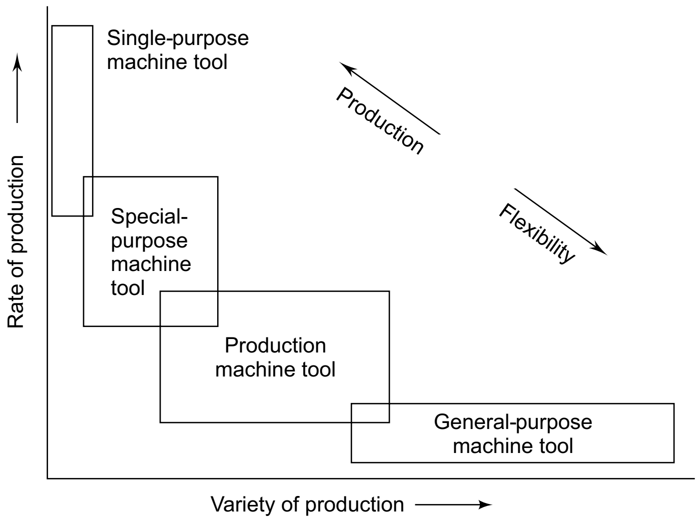


Fig. 3.1 *Application of machine tools based on the capability*

The other classification is based on the actual motions which is shown later in this chapter.

3.3 GENERATING AND FORMING

Generally the component shape is produced in machine tools by two different techniques, namely Generating and Forming.

Generating is where the required profile is obtained by manipulating the relative motions of the work piece and the cutting tool edge. Thus the obtained contour would not be identical to the shape of the cutting tool edge. This is generally used for a majority of the general profiles required. The type of surface generated depends upon the primary motion of the work piece as well as the secondary (feed) motion of the cutting tool.

For example, when the work piece is rotated and a single point tool is moved along a straight line parallel to the axis of rotation of the work piece, a helical surface is generated as shown in Fig. 3.2(a). If the pitch of the helix (feed rate) is extremely small then the surface generated may be approximated to a cylinder. This is carried out in lathes and called turning or cylindrical turning.

An alternate method of obtaining the given profile is called forming in which the shape of the cutting tool is impressed upon the work piece as shown in Fig. 3.2(b). Thus the accuracy of the shape obtained is dependent upon the accuracy of the form of the tool used.

However many of the machine tool operations are actually combinations of the above two. For example, when a dove tail is being cut, the actual profile is obtained by sweeping the angular cutter along the straight line. Thus, it involves forming (angular cutter profile) and generating (sweeping along a line) as shown in Fig. 3.3.

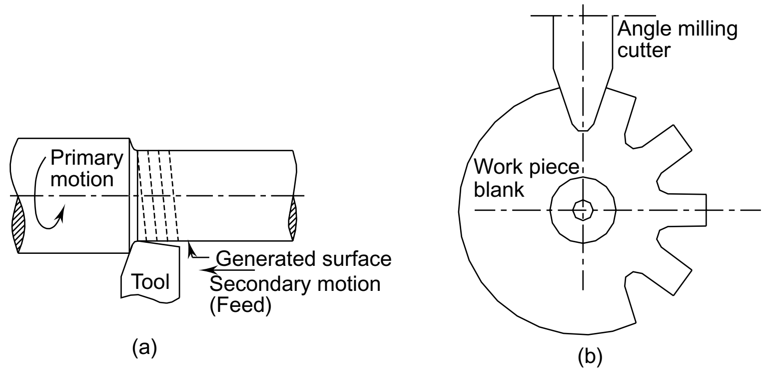


Fig. 3.2 *Generating and forming of surfaces by machine tools*

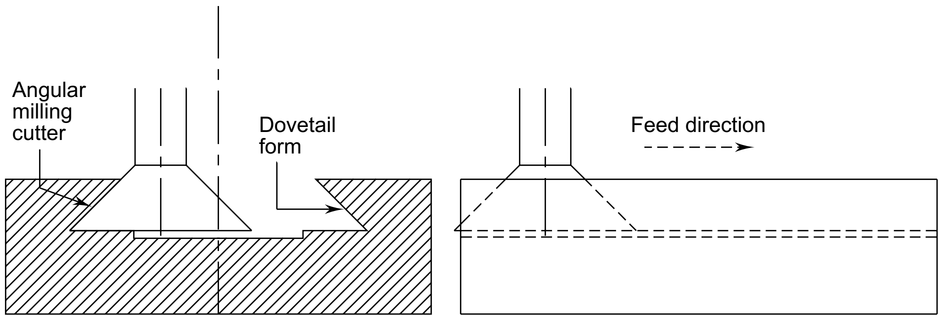


Fig. 3.3 *Generation of surface*

3.4 METHODS OF GENERATING SURFACES

A large number of surfaces could be generated or formed as the case may be with the help of the motions given to the tool and work piece. The shape of the tool also makes a very important contribution to the final surface obtained. Basically there are two types of motions in a machine tool. The primary motion given to the work piece or cutting tool constitutes the cutting speed which cause a relative motion between the tool and work piece such that the face of the cutting tool approaches the material to be removed. Usually, the primary motion consumes most of the cutting power. The secondary motion is which feeds the tool relatively past the work piece. The combination of the primary and secondary motions is responsible for the generation of specific surfaces, the details of which are discussed below. Sometimes there would be a tertiary movement in between the cuts for specific surfaces.

A classification of machine tools based on the motions is shown in Fig 3.4 for single point tools and Fig. 3.5 for multi-point tools. In the case of job rotation, cylindrical surfaces would be generated as shown in Fig. 3.6 when tool is fed in a direction parallel to the axis of rotation. When the feeding direction is not parallel to the axis of rotation, complex surfaces such as cones (tapers) (Fig. 3.7), or contours (Fig. 3.8) can be generated. The tools used in the above cases are of single point. If the tool motion is perpendicular to the axis of rotation, then a plane surface would be generated as shown in Fig 3.9. However, if a cutting tool of a given form is fed in a direction perpendicular to the axis of rotation (plunge cutting), then a contour surface of revolution would be obtained as shown in Fig. 3.10.

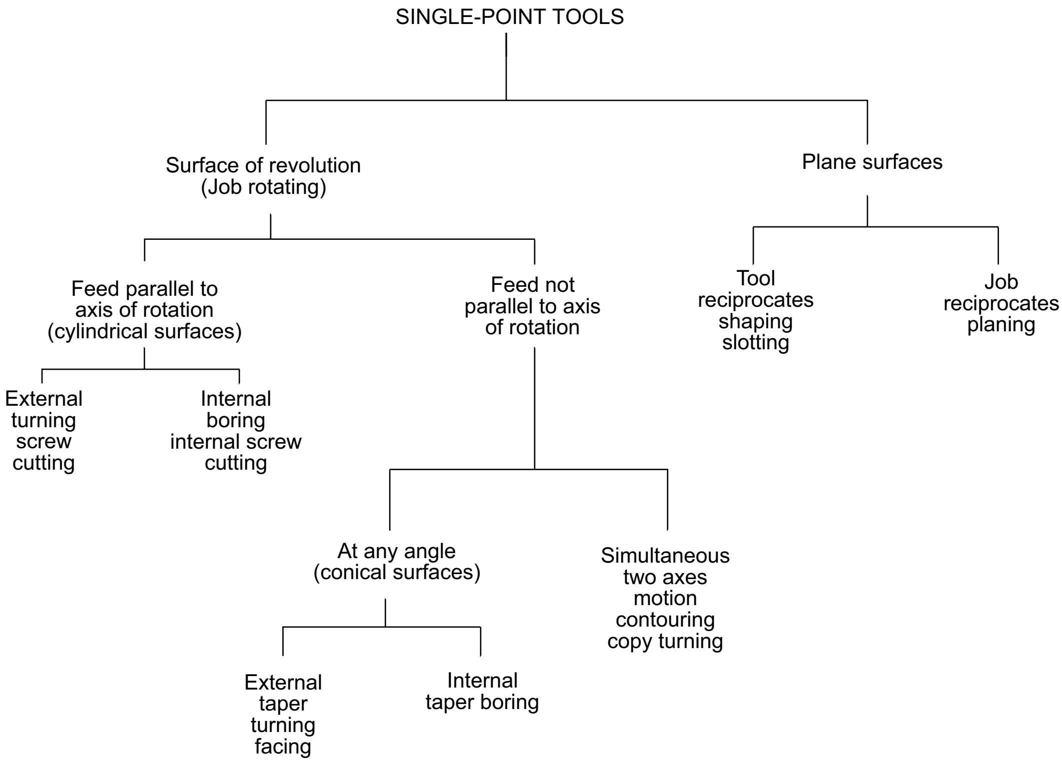


Fig. 3.4 *Classification of machine tools using single point cutting tools*

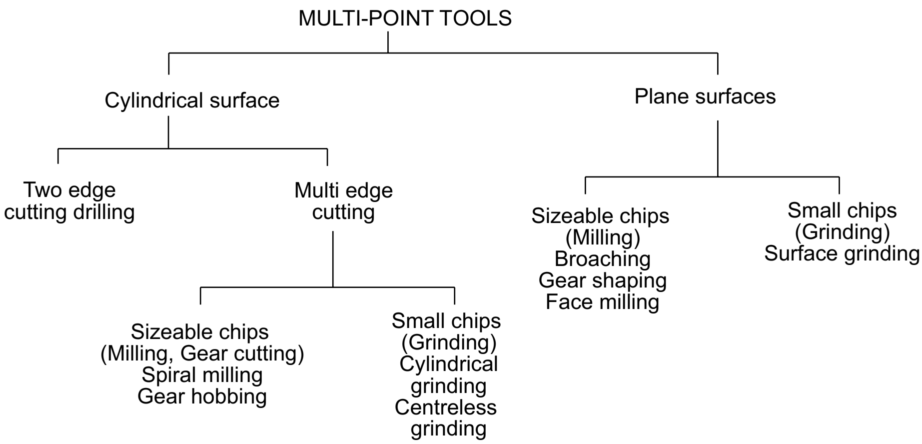


Fig. 3.5 *Classification of machine tools using multi-point cutting tools*

Plane Surface Generation in Shaping

Plane surfaces can be generated when the job or tool reciprocates the primary motion as shown in Fig. 3.11 without any rotation.

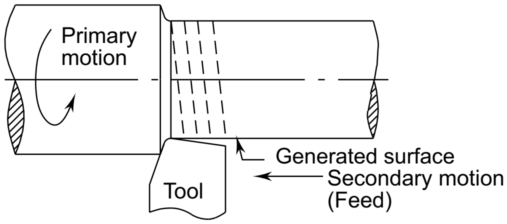


Fig. 3.6 *Generation of a cylindrical surface by a single point cutting tool*

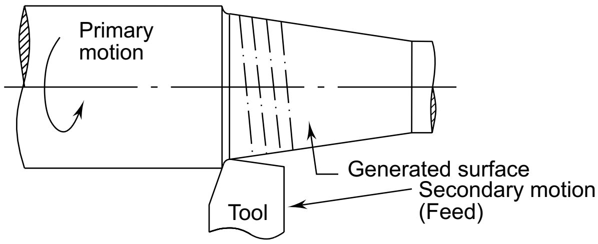


Fig. 3.7 *Generation of a conical surface by a single point cutting tool*

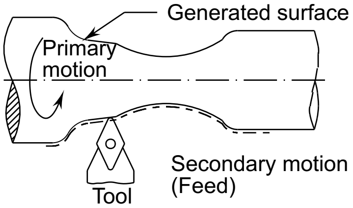


Fig. 3.8 *Generation of a contoured surface by a single point cutting tool*

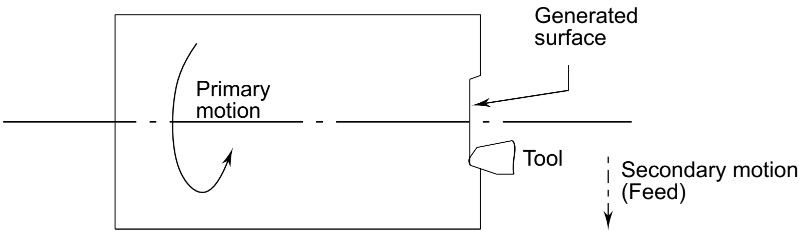


Fig. 3.9 *Generation of a flat surface by a single point cutting tool*

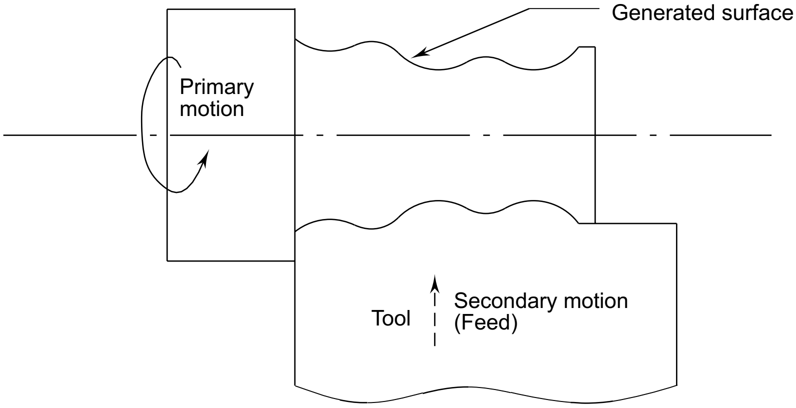


Fig. 3.10 *Forming a surface with a forming tool*

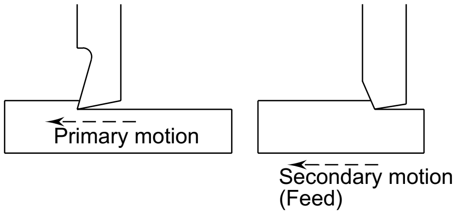


Fig. 3.11 *Generating a flat surface with linear motion of a single point cutting tool*

With the multi-point tool generally plane surfaces are generated as shown in Fig. 3.12. However, in this situation it is generally the combination of forming and generating to get a variety of complex surfaces, which are otherwise impossible to get through the single point tool operations. Some typical examples are the spur gear hobbing or spiral milling of formed cavities, which are described in later chapters.

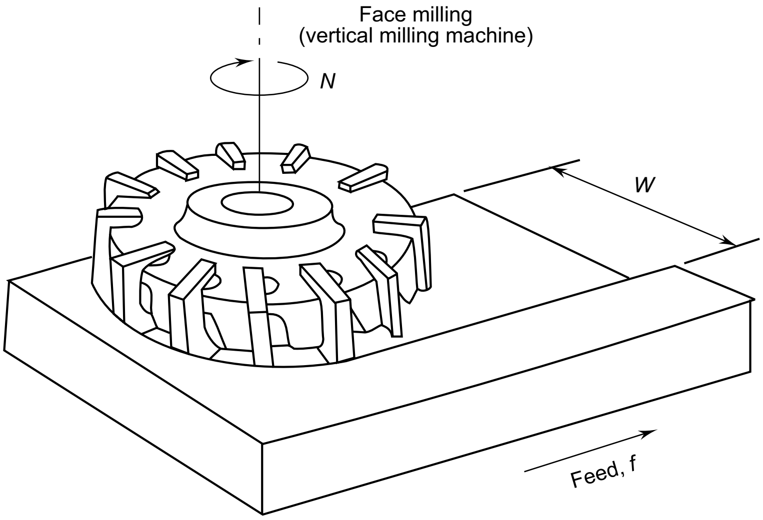


Fig. 3.12 *Multi-point cutting tool generating a plane surface*

3.5 ACCURACY AND FINISH ACHIEVABLE

It is necessary to select a given machine tool or machining operation for a job such that it is the lowest cost option. There are various operations possible for a given type of surface and each one has its own characteristics in terms of accuracy, surface finish and cost. At the time of process planning this selection is made. The obtainable accuracy for various types of machine tools is shown in the following Table 3.1. The surface finish expected from the various processes is shown in Fig. 3.13. The values presented in Table 3.1 and Fig. 3.13 is only a rough guide. The actual values are greatly varying depending upon the condition of the machine tool, cutting tool used and the various cutting process parameters as explained in Chapter 2.

TABLE 3.1 **Accuracies achievable in machining operations**

Machining Operation	Accuracy
Turning	± 25 μm
Shaping, Slotting	± 25 μm/side
Planing	± 65 μm/side
Milling	± 12 to 25 μm
Drilling in drill press—Location	± 250 μm
Hole	± 125 μm
Jig hole	± 50 μm
Drilling in lathe—Location	± 12 μm
Hole	± 2.5 μm
Boring	± 2.5 μm
Internal grinding	± 2.5 μm
Reaming	± 25 μm
Reaming with jig	± 12.5 μm
Jig boring—Hole	± 2.5 μm
Location	± 5 μm
Cylindrical and surface grinding	± 2.5 μm
Thread cutting products	± 50 μm
Broaching	± 12.5 μm
Lapping	± 5 μm
Honing	± 12.5 μm
Super finishing	± 0.5 μm

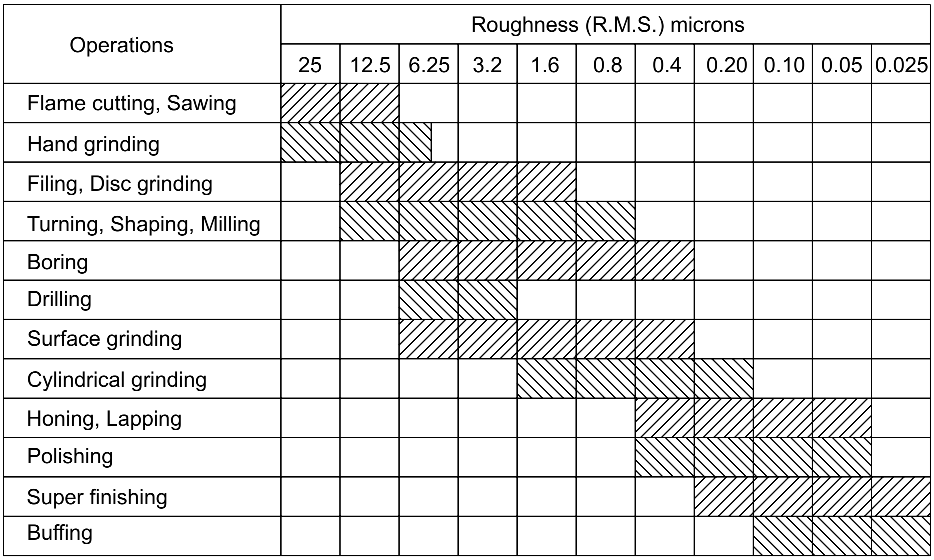


FIG. 3.13 *Achievable surface finishes by different machining processes*

3.6 BASIC ELEMENTS OF MACHINE TOOLS

The various components that are present in all the machine tools may be identified as follows:

Work holding device To hold the work piece in the correct orientation to achieve the required accuracy in manufacturing, e.g. chuck

Tool holding device To hold the cutting tool in the correct position with respect to the work piece and provide enough holding force to counteract the cutting forces acting on the tool, e.g. tool post

Work motion mechanism To provide the necessary speeds to the work piece for generating the requisite surface, e.g. head stock

Tool motion mechanism To provide the various motions needed for the tool in conjunction with work piece motion in order to generate the different surface profiles as desired, e.g. carriage

Support structure To support all the mechanisms as shown above and maintain their relative position with respect to each other and also allow for relative movement between the various parts to obtain the requisite part profile and accuracy, e.g. bed

The type of device or mechanism used varies, depending upon the type of machine tool and the function it is expected to serve. In this chapter some of the more common elements would be discussed. However, further details may be found in the chapters where the actual machine tools are discussed.

The two motions that need to be provided in the machine tool are cutting speed and feed. The range of speed and feed rates to be provided depends upon the capability of the machine tool and the range of work materials that are expected to be processed. Basically the actual speed and feed chosen depends upon the

- work material
- production rate desired
- surface finish required
- accuracy expected

The drive units in a machine tool are expected to provide the required speed and convert the rotational speed into linear motion as required. Details of these may be found in books dealing with machine tool design.

3.7 SUPPORT STRUCTURES

The broad categories of support structures found in various machine tools are shown in Fig. 3.14. They may be classified as beds (horizontal structures) or columns (vertical structures).

The main requirements of the support structure are:

- Rigidity
- Impact resistance
- Accuracy of guideways
- Wear resistance

Bed Bed provides a support for all the elements present in a machine tool. It also provides for the true relative positions of all units in machine tools. Some of these units may be sliding on the bed or fixed. For the purpose of sliding, accurate guideways are provided. Bed weight is approximately half the total of the machine tool weight.

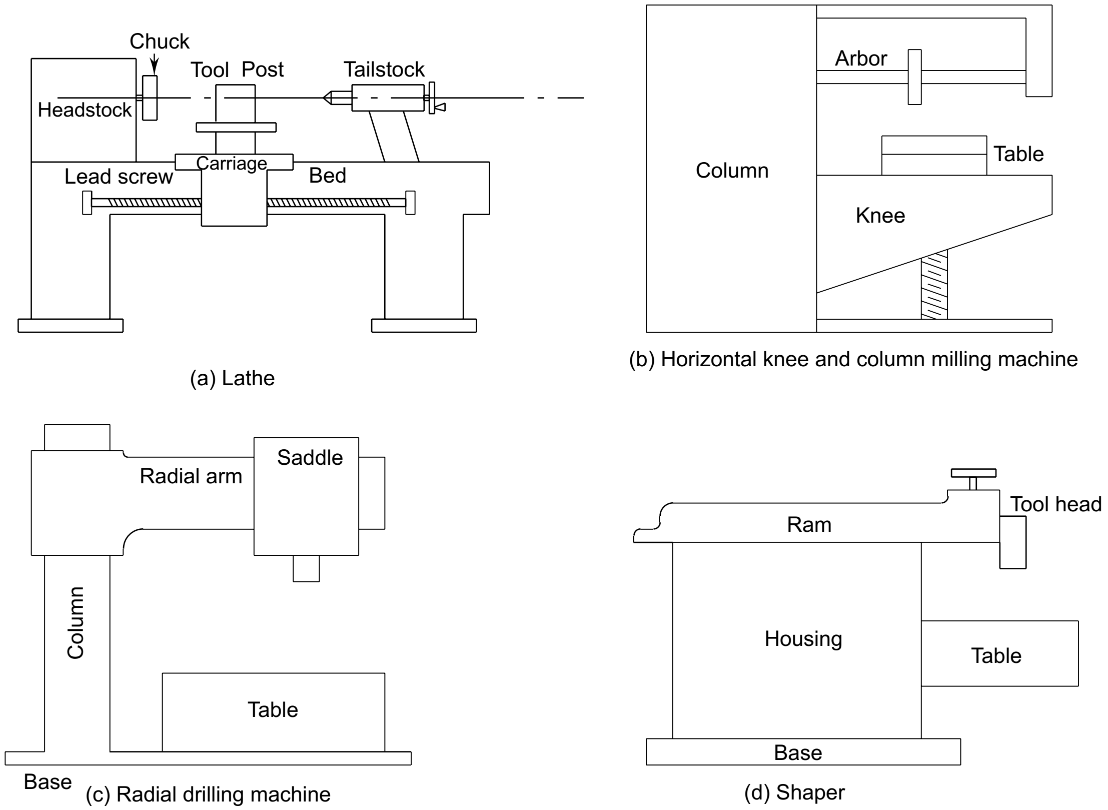


Fig. 3.14 *Different types of structures found in machine tools*

The basic construction of a bed is like a box to provide the highest possible rigidity with low weight. To increase the rigidity the basic box structure is added with various types of ribs as shown in Fig. 3.15. The addition of ribs complicates the manufacturing process for the beds.

Beds are generally constructed using cast iron or alloy cast iron consisting of alloying elements such as nickel, chromium and molybdenum. With cast iron, because of the intricate designs of the beds the casting defects may not be fully eliminated.

Alloy steel structure is also used for making beds. The predominant manufacturing method used is welding. The following advantages can be claimed for steel construction.

(a) With steels, the wall thickness can be reduced, thus greater strength and stiffness for the same weight would be possible with alloy steel bed construction.

(b) Walls of different thicknesses can be conveniently welded whereas in casting this would create problems.

(c) Repair of welded structures would be easier.

(d) Large machining allowances would have to be provided for casting to remove the defects and hard skin.

Concrete is also used as bed material. Its choice is mainly because of the large damping capacity. For precision machine tools and measuring machines granite is also used as the bed material.

The major types of bed styles used in the machine tools are shown in Fig. 3.16.

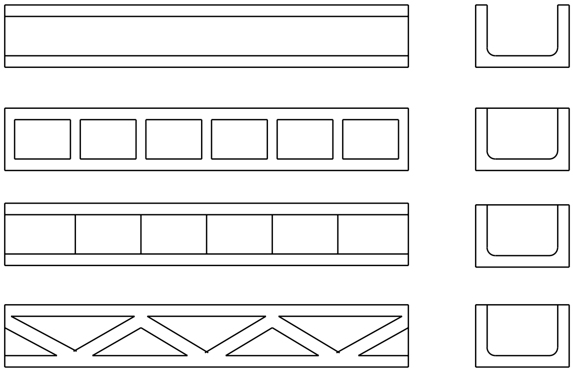


Fig. 3.15 *Different types of ribs used for strengthening machine tools beds*

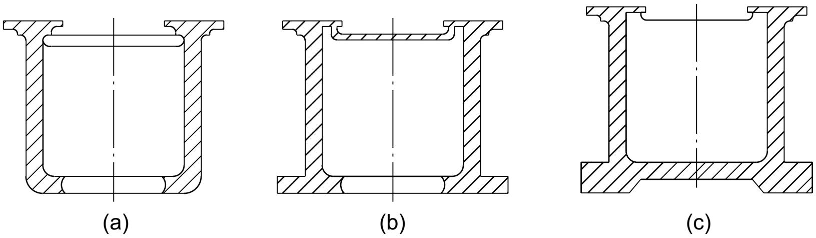


Fig. 3.16 *Different styles of beds used in machine tools*

Slant Bed Construction

With the advent of numerical control machines, slant bed construction for lathes has become more common. The slant bed as shown in Fig. 3.17 allows for better chip and coolant disposal besides better torsional rigidity compared to the conventional flatbed construction. It provides a better view of the machining area for the operator.

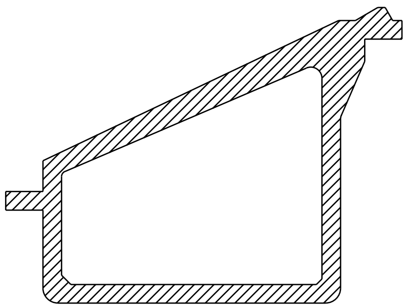


Fig. 3.17 *Typical slant bed construction used in lathes for easy disposal of chips*

3.8 POWER TRANSMISSION

The main power is provided by the spindle motor in most of the machine tools. From this source with a standard speed, the power need to be transmitted to run the spindle at different speeds and also provide feeds as required for differing machining situations. As seen earlier, speeds and feeds used during machining operations have a great effect on the tool lives and machining performance. The machine tool need to provide large variety of speeds and feeds to cater to the diverse range of materials that are cut.

Spindle Speeds

The choice of the range of speeds that need to be provided will depend upon the range of cutting speeds to be obtained along with the range of diameters of work pieces that need to be cut. The final speed at the spindle can be obtained in two ways:

(a) Stepped drive (Fixed speeds), or
(b) Step less (infinite speeds)

Stepped drives will have a fixed number of speeds that are generally obtained through multiple stages (generally 2 or 3). In the case of step less drives an infinite number of speeds can be obtained, either in single stage or multiple stages.

In either case when considering the design of the power transmission system it is important to take the following elements into consideration:

(a) Maximum spindle speed required (N_{max})
(b) Minimum spindle speed required (N_{min})
(c) Total number of speeds (n)
(d) The number of stages

The speed range, N_r is defined as the ratio of maximum to minimum spindle speeds to be used. Typical ranges of values for N_r are given in Table 3.2

Table 3.2 Speed ranges that are generally used in conventional machine tools

Machine Tool	Speed Range
Centre lathe	40 to 60
Milling machine	30 to 50
Drilling machine	20 to 30
Grinding machine	1 to 10

The cutting speed, V is related to the work piece diameter, D and the revolving speed of the work piece/tool, N as follows:

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

This equation can be rewritten as

$$N = \frac{1000\,V}{\pi\,D} \text{ rpm}$$

The maximum and minimum spindle speeds can be obtained from an idea of the maximum and minimum work piece that need to be considered assuming it to be a lathe. In the case of a milling machine it should be the smallest and largest size of the cutter diameters to be used. Hence

$$\text{Maximum spindle speed, } N_{max} = \frac{1000\,V_{max}}{\pi\,D_{min}} \text{ rpm}$$

$$\text{Minimum spindle speed, } N_{min} = \frac{1000\,V_{min}}{\pi\,D_{max}} \text{ rpm}$$

Where, V_{max} = maximum cutting speed to be used, m/min
V_{min} = minimum cutting speed to be used, m/min
D_{max} = maximum diameter of the work piece/cutter, mm
D_{min} = minimum diameter of the work piece/cutter, mm

Rather than selecting the individual cutting speeds between the maximum and minimum speeds arbitrarily, they are chosen following any one of the progressions such as arithmetic progression, geometric progression or logarithmic progression. A typical ray diagram of speeds is shown in Fig. 3.18. This is a plot of cutting speed vs. diameter. The full lines in the Fig. 3.18 represent the constant spindle speeds, N1, N2, etc. For example at a spindle speed of N1, the maximum and minimum diameters need to be between D1 and D2 to achieve the range of cutting speeds for which the machine tool is designed. It is interesting to note that as spindle speed is increased, the range of diameters decrease to maintain the range of cutting speeds.

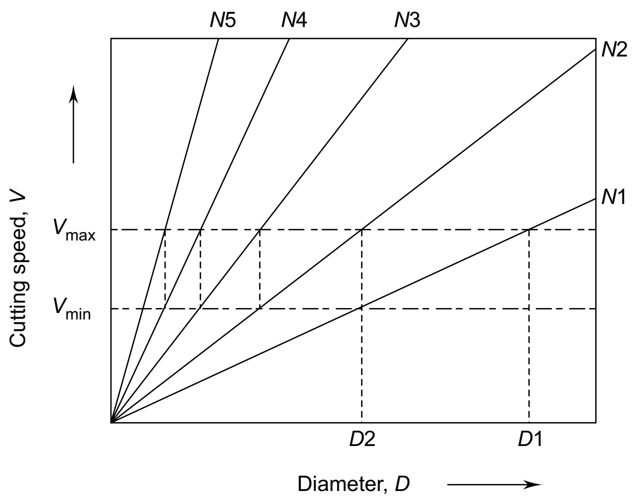


Fig. 3.18 *Typical ray diagram of spindle speed*

Arithmetic Progression

An arithmetic progression is a sequence in which each term (except the first term) is obtained from the previous term by adding a constant known as the common difference. For arithmetic progression the intermediate speeds can be obtained by using the following formula:

Speed, $N_i = N_{\min} + (i - 1)\, a$

$$N_{\max} = N_{\min} + (n - 1)\, a$$

Where, a = common difference,
n = number of speeds.

Hence, $$a = \frac{N_{\max} - N_{\min}}{n - 1}$$

For example, if 12 speed layout is to be chosen with minimum speed of 100 and maximum speed of 3400 rpm.

Then common difference, $a = \dfrac{3400 - 100}{12 - 1} = 300$ rpm

Thus the required speeds are: 100, 400, 700, 1000, 1300, 1600, 1900, 2200, 2500, 2800, 3100, and 3400.

Geometric Progression

Geometric progression is a sequence in which each term (except the first term) is derived from the preceding term by the multiplication of a non-zero constant, which is the common ratio. For geometric progression the intermediate speeds can be obtained by using the following formula:

Speed, $N_i = N_{min}\phi^{(i-1)}$

Maximum speed, $N_{max} = N_{min}\,\phi^{(n-1)}$

Or, $\phi = \sqrt[(n-1)]{\dfrac{N_{max}}{N_{min}}}$

Where ϕ = Common ratio.

For example, if 12 speed layout is to be chosen with minimum speed of 100 and maximum speed of 4050 rpm.

Then common ratio, $\phi = \sqrt[(12-1)]{\dfrac{4050}{100}} = 1.4$

Thus the required speeds are: 100, 140, 196, 274, 384, 538, 753, 1054, 1476, 2066, 2893, and 4050.

It can be noticed that the speed steps in the case of geometric progression are more dispersed at higher speeds, which allows for a range of work piece diameters to be used unlike in arithmetic progression where the range of work piece diameters that can be used at the higher spindle speeds is very limited. This limits the productivity of the machine tools. Hence geometric progression is generally preferred by the machine tool designers. The common ratios that are employed are 1.26, 1.41 and 1.58 for general work.

Logarithmic Progression

Another method that is possible for arranging range of spindle speeds is logarithmic progression (LP). Though the calculations involved are a little complex and utilizes trial and error procedures, LP reduces the crowding of speeds in the higher range while the gaps at the lower range are also narrowed down.

Step Less Drives

The problem with the stepped drives is due to the fixed spindle speeds, it is not possible to use optimum cutting speeds with any of the work piece diameters. This reduces the productivity of the machining operation since the operator will be working at non-optimum values most of the time. Also the speed change operation reduces the productivity further since the machine has to be stopped. The surface finish achieved will not be uniform as the cutting speed is likely to vary when the work piece diameter changes with fixed spindle speed.

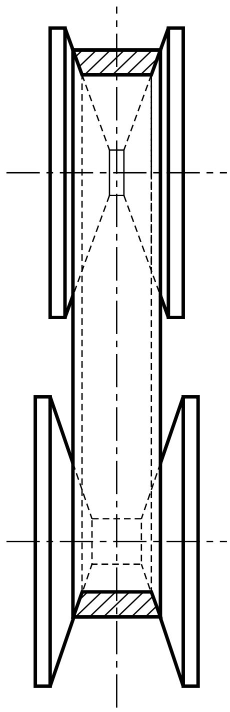


Fig. 3.19 *Step less drive utilizing V-belt*

One type of step less drive utilizing V-belts is shown in Fig. 3.19. This system consists of two pairs of conical discs that form V-belt like sheaves as shown in Fig. 3.19. The belt rides on the two conical discs. To change the speed ratio, the top discs are moved out (or in) while the bottom ones are drawn together (or out) by the same distance to maintain the constant belt length. Changing the axial distance of the discs will vary the point of contact between the belt and the discs. When the discs come closer the belt is pushed towards the periphery of the discs thereby increasing the effective diameter of the pulley. The reverse action reduces the effective diameter of the pulley by bringing the belt closer to the centre. Since the length of the belt is constant, there is a corresponding change in the diameter of the other pulley. This way it is possible to vary the speed ratios continuously. This process can be done without stopping the machine tool.

Another step less drive used is a positively-infinitely-variable or PIV drive as shown in Fig. 3.20. In this system a chain is used in place of the V-belt as shown earlier. Because of the use of a chain the transmission will be a positive torque transmission. This system consists of two pairs of conical wheels A, B, C and D, which are movable on their axis as shown in Fig. 3.20. These conical wheels are connected by means of a special chain. The conical wheel faces are radially serrated. The laterally movable slats of the chain engage the tooth spaces of the conical wheel face and transmit the torque positively. When the conical wheels move axially the effective diameter of engagement varies thereby changing the speed ratio. The chains are normally provided with an automatic chain tensioning device for positive power transmission.

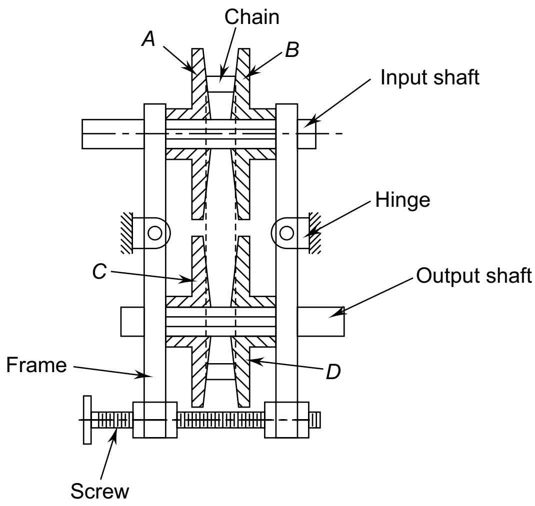


Fig. 3.20 *PIV drive*

3.9 ACTUATION SYSTEMS

Lead Screws

The rotary motion from the drive motor needs to be converted to linear motion to move the various axes of the machine tool. In conventional machine tools, the square (Acme) thread is normally used for this purpose. However, in view of the metal-to-metal sliding contact between the nut and the screw, the friction is very high. This results in greater power being utilised for the movement of the axes. Typical friction coefficients for these systems are shown in Table 3.3. Further, in view of the clearance provided between the nut and the screw in the case of Acme thread as shown in Fig. 3.21 to reduce friction, there is the problem of backlash whenever there is a reversal of motion. If any attempt is made to reduce the backlash, the friction increases. Hence most of the higher end machine tools use a lead screw with a recirculating ball nut.

Table 3.3 **Lead screw efficiencies**

Type	Efficiency (%)		
	High	Median	Low
Recirculating Ball screw - nut	95	90	85
Acme with metal nut*	55	40	35

* Since metallic nuts usually require a viscous lubricant, the coefficient of friction is both speed and temperature dependent.

In the case of recirculating ball screws, the nut is replaced by a series of balls, which circulate in the channel in the form of threads as shown in Fig. 3.22. This results in a highly efficient rolling motion of balls in the space between the screw shaft and nut. The balls at the end of the thread portion in the nut will be re-positioned into the beginning of the thread form by a deflector as shown in Fig. 3.22. The size of the nut being an internal return of the balls is small compared to the external return type using an external return tube.

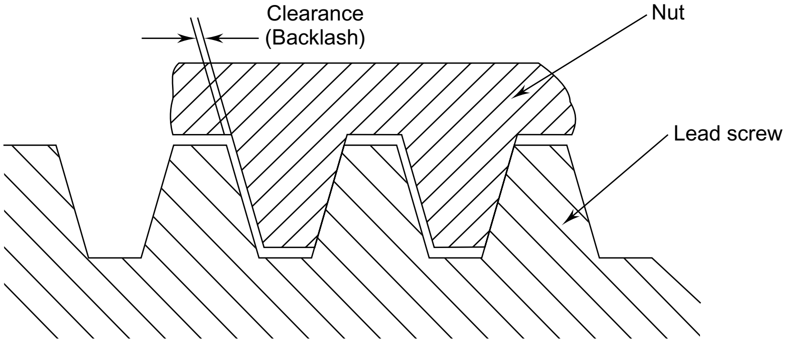


Fig. 3.21 *Lead screw with Acme nut*

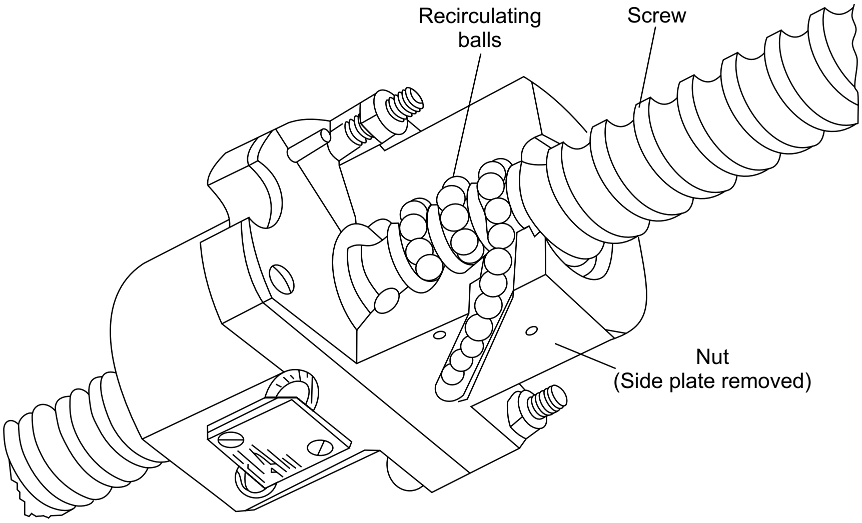


Fig. 3.22 *Recirculating ball screw*

Further, the ball screws can be preloaded to eliminate the axial displacement, which consequently also reduces the backlash. One of the methods followed for pre-loading is keeping a spacer between the two nuts as shown in Fig. 3.23. This increases the axial rigidity of the nut while decreasing the axial displacement.

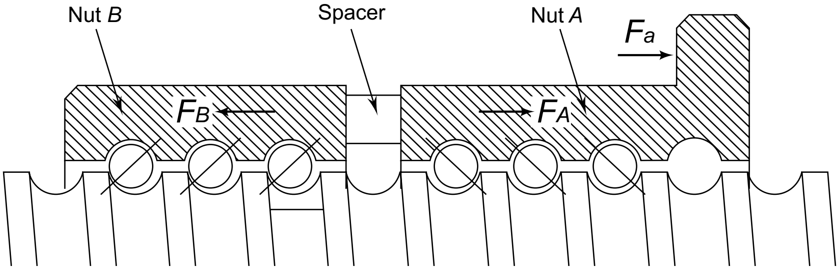


Fig. 3.23 *Pre-loading of the recirculating ball screw and nut arrangement*

The recirculating ball screws have a number of advantages compared to the conventional type of screws.

(a) They have a longer life.
(b) The wear of the screw is relatively small. Hence it will maintain accuracy through the entire life of the screw.
(c) The frictional resistance offered is small, hence can be used for carrying heavier loads at faster speeds.
(d) The power required for driving is small due to less friction.

3.10 GUIDEWAYS

Guideways (slideways) are linear bearings for translatory movement between two members of a machine tool, such as carriage and bed in a lathe. They should provide

- alignment and fit
- ample load carrying capacity
- absence of stick-slip jerk
- adjustment for wear and lubrication

When motion starts there should be instant and accurate response with repeatability both for feeding and pre-locating. The slide is at the end of a transmission train and the time lag between the input command and output response for a small effort will depend primarily upon the stiffness of the transmission, the friction, etc.

The requirements of the guideways are:

- High accuracy
- High surface finish
- Low value of frictional force
- Low wear rate

A close grained material is used along with inclusion of alloying elements. The bearing surfaces are flame and induction hardened for better wear characteristics. The bearing surfaces can be either ground or scraped. Grinding being high heat production process should be properly managed and the checking of the guideways for straightness should be done when it returns to normal temperature. Scraping when used should confirm to a specified number of high spots per unit area, the low spots forming minute oil wells. Flaring cup wheel grinding forms a multitude of oil channels, but the grain marks can abrade the mating surfaces. For very high accuracy such as jig boring, scraping is used.

The types of guideways may be classified as

- Guideways with sliding friction also termed as slide ways.
- Guideways with rolling friction also termed as anti-friction ways. In this there are two varieties with balls or rollers as anti-friction elements.

3.10.1 Types of Slideways

There are a number of types of slideways conventionally used in machine tools. They are (Fig. 3.24):

(a) V-slideways
(b) Flat slideways
(c) Round slideways
(d) Dovetail slideways

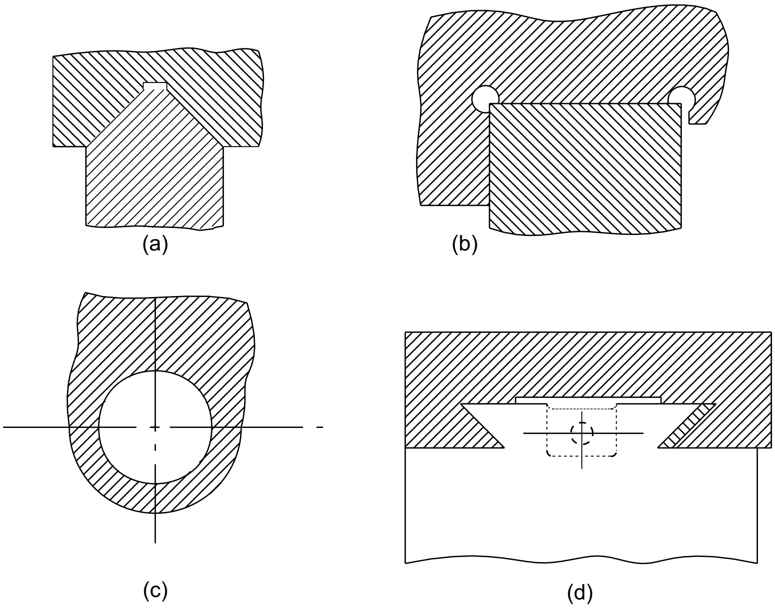


Fig. 3.24 *Different types of slideways used in machine tools*

The shape of slideway depends basically upon the function it is expected to serve. The following are some guiding principles in identifying the shape of a slideway.

- load carrying capacity
- ease of manufacture
- ease of chip disposal
- effective lubrication
- position of slideway

V-slideways

They can adjust clearances under the weight of the moving member. Machining accuracy is not greatly affected by the wear of guideways. The chips would not clog the slideways. However, they are difficult to manufacture and provide very small bearing area. They can be symmetrical as in Fig. 3.24(a) or more generally asymmetric as in Fig. 3.25. Symmetrical is good when the acting load is perfectly vertical since the load will be equally distributed among the faces of V. However in metal cutting, we have seen that tangential force is more compared to radial force. Thus, in asymmetrical V, the longer face is normally made perpendicular to the direction of the resultant force. This allows for uniform wear in both the faces.

Flat Slideways

Flat slideways [Fig. 3.24(b)] are easier to manufacture and are able to provide a large amount of bearing area. However, the locational accuracy and other advantages provided by the V-slideways are missing. Hence the flat slideways cannot be used on their own, but in conjunction with the V-slideways. The most common arrangement found in lathes is shown in Fig. 3.26. Similar arrangement is also used in planers and horizontal boring machines.

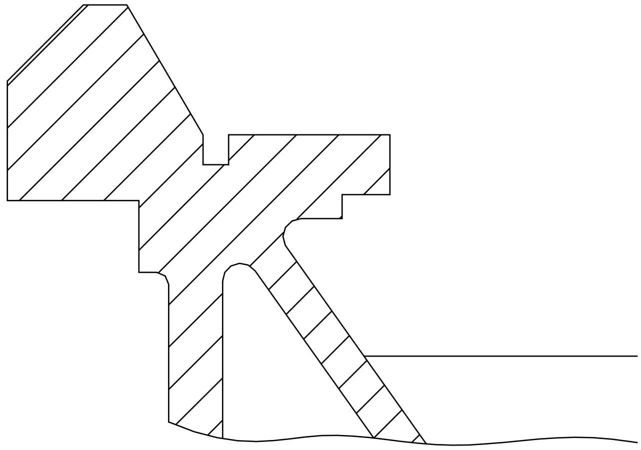


Fig. 3.25 *Single V-slideway used in machine bed*

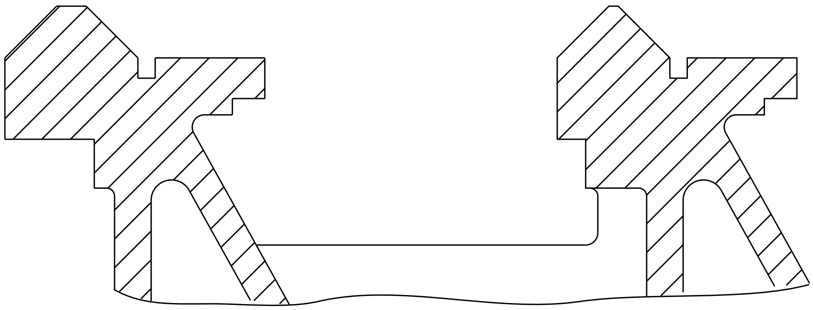


Fig. 3.26 *Asymmetric V-slideway used in machine tool bed.*

Round Slideways

Round slideways [Fig. 3.24(c)] are kinematically sound since they constrain all possible motions except the direction required, but are only used in vertical type of machines such as drilling (radial and pillar). Main reasons for their non-acceptance are:

- low rigidity
- difficulties in manufacture to the given accuracy
- difficulties in assembling and the resultant accuracy in motion

Dovetail Slideways

Dovetail slideways [Fig. 3.24(d)] are compact, but difficult to manufacture. These are generally used for vertical movement and where space is at a premium. However they provide good rigidity and alignment. Dovetails are generally used in carriage of lathe, for moving knee of a milling machine.

Materials for Slideways

Generally cast iron is used as the bed material and with integral slideways cast iron can also become the material for slideways. However cast iron is poor in wear resistance and hence has to be induction hardened.

Alternatively separate steel guideways can be used which can be welded in case of welded bed construction. Steel guideways have all the requisite properties and are easier to manufacture and maintain accuracy.

A very high level of friction is encountered in slideway systems as shown in the above varieties because of the metal to metal contact existing between the members. Also because of the large variation in the static and dynamic friction coefficient, stick slip phenomenon would be observed with metal to metal contact. This is undesirable, particularly for high speed operations such as those found in the modern machine tools.

A method which is often used to minimise the stick slip motion and provide some measure of damping capacity and improve the wear resistance, is when composite materials of small thickness are provided in between the moving members of a slideway system. The slideway composite materials are made from two or more materials so that one of them provides the friction reduction capability, and others increase the strength, wear resistance and load bearing capacity. Another advantage is when this material wears out, the composite material can be re-pasted. A slideway system with composite material is shown in Fig. 3.27. A number of composite materials are commercially available such as Turcite-B, SKC-3, and Ferobestos LA3.

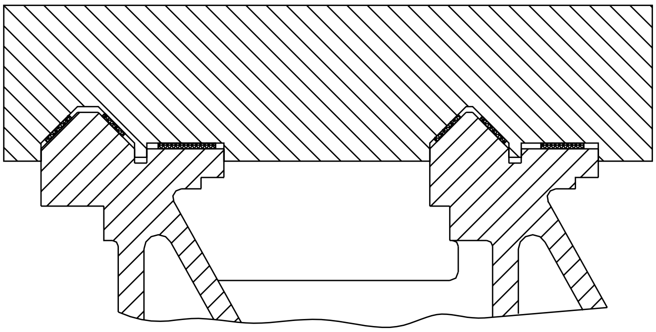


Fig. 3.27 *Anti-friction material used between the bed and the carriage of a lathe*

Anti-friction Slideways

The anti-friction guideways involve intermediate rolling members (balls or rollers) between the sliding members, thus reducing the friction as shown in Fig. 3.28.

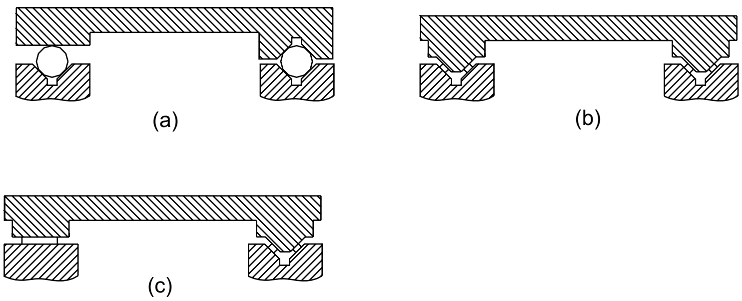


Fig. 3.28 *Different types of anti-friction guideways used in machine tools*

The use of anti-friction guideways because of the low friction during motion, and high stiffness, will provide very high velocities possible for slideway movement. These are generally used in high precision machine tools.

3.10.2 Linear Motion Elements

With the advent of the CNC machine tools, the need for decreasing the friction in linear motion has been felt to a very great extent. The concept of linear motion elements is to provide friction free (with rolling friction from the rolling balls or cylinders) and rigid motion with linear travel, e.g. for slides. A large variety of LM devices is available and is extensively used in CNC machine tools. The typical rapid positioning speeds as a result of the linear motion elements or sometimes termed as LM devices is an astounding 20 to 40 m/min.

Linear Motion Systems

Since the friction is high in the conventional slideways, the antifriction slideways are generally used which makes use of the rolling friction by the use of re-circulating balls. A typical linear motion guide using the rails is shown in Fig. 3.29. As shown in the cross-sectional view there are a number of re-circulating balls providing a rolling motion between the slider and the rail. At the end of the block there are end plates to ensure that the balls circulate through the rolling tracks. These provide a very high rigidity and very low friction for the movement of the axes. In view of the low friction, there is less wear and hence these systems are able to maintain the accuracy throughout its long life. A number of varieties of these LM guides are available off the shelf.

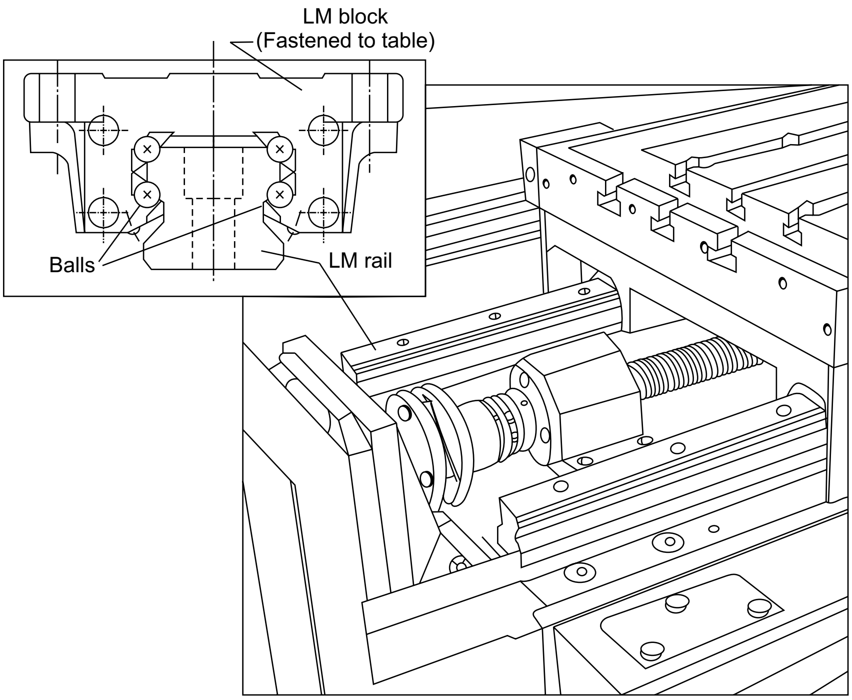


Fig. 3.29 *Linear motion system*

Another type of linear motion device is the use of a ball bush where the balls are arranged in a track inside of a bush, which can slide along a ground rod to provide the linear motion similar to a round slideway used in conventional machine tools.

3.11 GENERAL WORK HOLDING METHODS

Work holding is a very important function since the accuracy achieved depends upon the accuracy with which the component is held. Also a large number of work holding devices are required to cater to the range of work piece profiles present. The following are some of the types of generally used work holding devices:

- Vices
- Chucks
- Clamps

Vices

These are used for general purpose work holding on milling machines, shapers and grinding machines. The vice consists of a fixed jaw and moving jaw and is generally useful for jobs which are prismatic and with plane faces. The location available is the jaw surface which is generally a plane. However it is also possible to change the types of jaw to suit the external contour of the job. For example a VEE jaw can be used as for cylindrical external contours.

Chucks

Chucks are basically used for axi-symmetric jobs as well as for irregular surfaces where the job requires to be rotated during or in between the machining operation. They are generally used in lathes, cylindrical grinding machines and milling machines fitted with dividing heads.

The chucks may have two, three or four jaws depending upon the desired fixturing requirement. More details of chucks are given in Chapter 4.

Clamps

The table of machine tools generally have the flat surface with accurately machined T-slots, which are generally used for work holding purpose. Various clamps and locating elements are available to hold complex surfaces on these surfaces. These are discussed in later chapters that relate to specific machine tools.

SUMMARY

Machine tools are mainly used to provide higher accuracy that is essential in the manufacturing industry. A machine tool is able to hold a cutting tool and removes material from the stock to get the required part geometry.

- Machine tools are classified into different types based on their application, as general purpose, production and special purpose machine tools.
- Part geometry can either be generated or formed.
- Depending upon the surface a large number of different motions are used, based on the type of tools in operation.
- There is a large variation in accuracy and surface finish between the different operations.
- All the machine tools have work holding, tool holding, support, and work and tool motion mechanisms built in them.
- There is a variety of support structures used in machine tools depending upon the type of motions desired.
- Power transmission in machine tools is utilized to obtain a range of speeds and feeds that are required to cater to the different work materials and machining conditions.

- Lead screws are used in machine tools to convert rotary motion to linear motion, which is used to move the table or cutting tool depending upon the type of machine tool.
- Recirculating ball screws are used for more efficient load transmission.
- Antifriction guideways are used in a majority of the modern machine tools, while the conventional guideways are also used.
- A number of work holding systems are possible based on the part geometry.

Questions

3.1 How do you classify the various machine tools based on the motions used for generating the surfaces? Explain with the help of suitable block diagram.

3.2 What are the various methods available for generating plane (flat) surfaces with machine tools?

3.3 Compare Generating and Forming with reference to metal cutting machine tools.

3.4 Explain the term 'Machine Tool' and how is it different from machine.

3.5 Explain the classification of machine tools based on production capacity. Give the relative advantages with reasons and applications.

3.6 How do you select a machine tool for a given application? Give your answer with an application.

3.7 What are the various elements present in a machine tool for the purpose of generating any specified surface? Explain their relevance from the viewpoint of the final requirement.

3.8 What are the materials used for the manufacture of support structures in machine tools? Give their applications and disadvantages.

3.9 What are the types of slideways used in machine tools with particular reference to a centre lathe? Explain why such a geometry is preferred.

3.10 How do you classify various manufacturing machines based on the motions of the machine axes? Show the example surfaces generated for each of them.

3.11 What are the types of guide-ways used in machine tools? Explain about the various types generally used in lathe machines.

3.12 What is the function served by a lead screw in a machine tool?

3.13 Give the advantages derived by the use of re-circulating ball screws.

3.14 Write a brief note on linear motion elements used in machine tools.

Multiple Choice Questions

3.1 General purpose machine tools are used
(a) For high production rates
(b) In normal workshops and repair shops
(c) For large production volumes
(d) For automated production

3.2 Generating a surface compared to forming is
(a) Requires less power
(b) Less accurate
(c) More accurate
(d) Requires more power

3.3 Steel structure with welding is preferred to cast iron for machine tool structure because they
(a) Absorb impact loads
(b) Reduce noise
(c) Are heavier
(d) Repair of welded structure is easy

3.4 V-slideway is preferred in lathe bed because it
(a) Is easy to manufacture
(b) Adjusts clearance under the weight of the moving member
(c) Is a simple design
(d) No specific advantage

3.5 Speeds in a gear box of a machine tool are chosen to follow the geometric progression because it
(a) Allows to have fixed speed difference between any two consequent speeds
(b) Appears more geometrical
(c) Allows more uniform dispersion of speeds in the entire range
(d) Allows more speeds in the maximum speed range

3.6 Recirculating ball screws are used because
(a) They are easy to manufacture
(b) Power required for driving them is small due to small friction
(c) Frictional resistance is more compared to Acme threads
(d) Variable friction present due to the recirculating balls

3.7 The following system has the lowest frictional resistance
(a) Dove tail slideway
(b) Round slideway
(c) Linear motion device used in the slideway
(d) V-slideway

3.8 Ribs are provided in machine tool structures to
(a) Increase the weight of the structure
(b) Increase the rigidity of the structure
(c) Improve the appearance of the structure
(d) Store the chips during the machining operation

3.9 Flat surface can be produced on a machine tool
(a) Using a single point tool moving along the axis of rotation of work piece
(b) Using a single point tool moving perpendicular to the axis of rotation of work piece
(c) Using a single point tool moving at an angle to the axis of rotation of work piece
(d) Using a single point tool moving along the axis of rotation of work piece

3.10 The following process provides best surface finish
(a) Hand grinding
(b) Cylindrical grinding
(c) Cylindrical turning
(d) Milling

3.11 The following process provides highest dimensional accuracy
(a) Cylindrical turning
(b) Jig boring
(c) Shaping
(d) Milling

3.12 The most complete definition of a machine tool is that
(a) converts the energy from one type to other
(b) holds the work piece and provides rotary motion
(c) holds the work piece and the cutting tool to remove metal
(d) holds the cutting tool to remove metal

3.13 Special purpose machine tools (SPM) are used
(a) For small batch manufacture
(b) In normal workshops and repair shops
(c) For large production volumes
(d) For one off production of complex geometries

3.14 Production machine tools are
(a) Used for small batch manufacture
(b) Normal machine tools fitted with jigs or fixtures
(c) Used for large production volumes
(d) For one off production of complex geometries

3.15 Forming a surface by a machine tool is done by
(a) Controlling the motion of the cutting tool to generate the profile
(b) Transferring the shape of the cutting tool directly to the work piece
(c) Controlling the motion of the cutting tool and the work piece to generate the profile
(d) Controlling the motion of a general purpose cutting tool to generate the profile

Answers to MCQs

3.1 (b)	3.2 (c)	3.3 (d)	3.4 (b)	3.5 (c)
3.6 (b)	3.7 (c)	3.8 (b)	3.9 (b)	3.10 (b)
3.11 (b)	3.12 (c)	3.13 (c)	3.14 (b)	3.15 (b)

CHAPTER 4

Centre Lathe

Objectives

Centre lathe is the most common and versatile machine found in practically all machine shops. After completing this chapter, the reader will be able to

- Understand the importance of lathe and its many varieties
- Understand the basic structure of a centre lathe
- Choose various aids that are used to locate and support work pieces in a lathe
- Select the large variety of cutting tools that can be used in a lathe
- Practice a variety of common operations that can be performed in a lathe
- Understand the various methods used to carryout the taper turning operations in a lathe
- Understand the requirements and the methods to be used to cut precision threads in a lathe
- Estimate the machining time and power required for any given operation in a lathe
- Plan typical setups that can be done in a lathe

4.1 INTRODUCTION

Lathe is the oldest machine tool invented, starting with the Egyptian tree lathes. In the Egyptian tree lathe, one end of the rope wound round the work piece was attached to a flexible branch of a tree while the other end was pulled by the operator, thus giving the rotary motion to the work piece. This primitive device has evolved over the last two centuries to be one of the most fundamental and versatile machine tool with large variants to be used in practically all the manufacturing shops.

The principal form of surface produced in a lathe is the cylindrical surface. This is achieved by rotating the work piece while the single point cutting tool removes the material by traversing in a direction parallel to the axis of rotation as shown in Fig. 4.1 and termed as turning. The popularity of lathe is in view of the fact that a large variety of surfaces could be produced.

Considering the versatility, a large number of variants of lathes are used in manufacturing shops. The variations are:

1. Centre lathe
    - Bench Lathe
2. Tool room lathe

3. Special purpose lathes
   - Copying lathe
   - Gap bed lathe
   - Hollow spindle lathes
4. Capstan and turret lathes
5. Automatic lathes

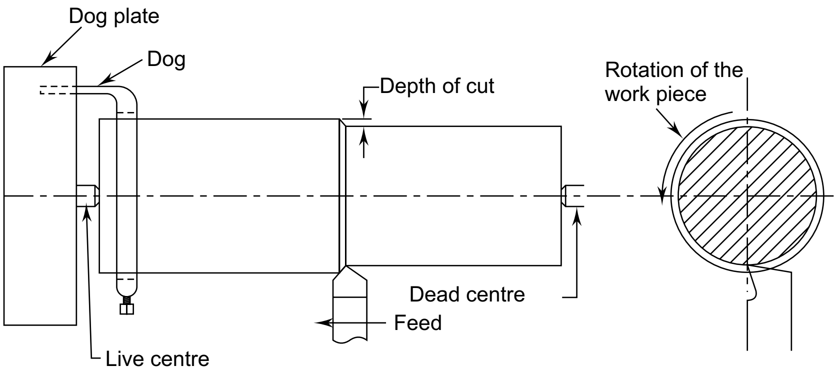


Fig. 4.1 *Cylindrical turning operation in a lathe*

The centre lathe is the most common of the lathes, which derives its name from the way a work piece is clamped by centres in a lathe, though this is not the only way in which the job is mounted. This is sometimes also called engine lathe in view of the fact that early lathes were driven by steam engines. This is used for more general applications and thus the construction of the machine tool is more rigid. This is discussed in greater detail in this chapter.

The tool room lathe is generally meant for applications of tool making, where the accuracy desired is much higher than is normally required for general production work. Also the range of sizes and materials handled would normally be large. Thus the machine would have a higher range of speeds and feeds along with greater rigidity. Also the range of accessories and attachments would generally be larger.

The special purpose lathes are developed from the centre lathe, to cater to special forms of application which cannot be handled by the conventional centre lathe.

Capstan and turret lathes and automatic lathes are the form of lathes to cater for high rate production and thus would be used for very special applications. These have the special features to help in improving the production rate and also work unattended if necessary. These are discussed in the next chapter.

4.2 CONSTRUCTIONAL FEATURES OF A CENTRE LATHE

The typical centre lathe is shown in Fig. 4.2. The major elements present in the lathe are:

The headstock houses the power source, all the power transmission, gear box and the spindle. The headstock is fixed at the left-most end on the bed. The spindle is hollow and should be sufficiently rigid to provide accurate rotary motion and maintains perfect alignment with the lathe axis. A live centre fits into the Morse taper in the spindle hole for the purpose of locating the work piece axis.

The main gear box provides the necessary spindle speeds considering the range of materials to be turned in the lathe. The headstock also houses the feed gear box to provide the various feed rates and thread cutting ranges.

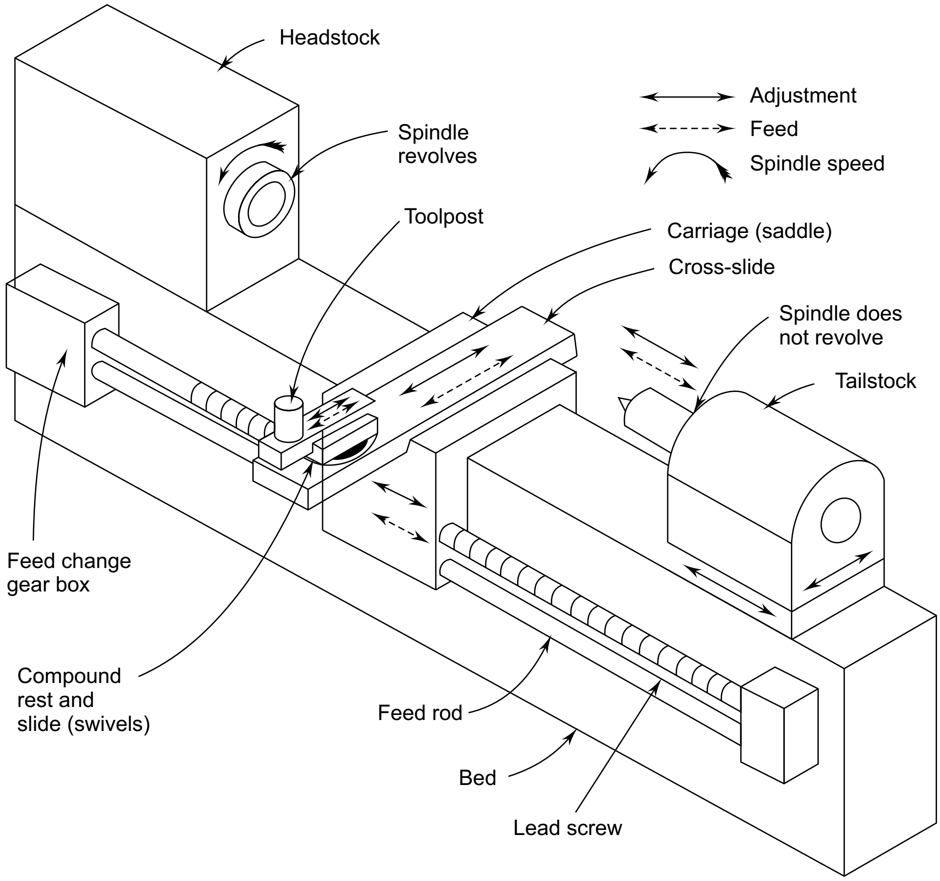


Fig. 4.2 *General view of a centre lathe showing various mechanisms and features*

The tailstock is towards the right-most end on the bed, and houses the tailstock spindle for the purpose of locating the long components by the use of centres. The tailstock is movable on the inner guideways provided on the bed to accommodate the different lengths of work pieces. It also serves the purpose of holding tools such as centre drill, twist drill, reamer, etc. for making and finishing holes in the components, which are located in line with the axis of rotation.

The third major element in the lathe mechanism is the carriage, which provides the necessary longitudinal motion for cutting tool to generate the necessary surfaces. This also houses three parts: the cross slide for giving motion (cross feed) to the cutting tool in a direction perpendicular to the axis of rotation, the compound slide which provides an auxiliary slide to get the necessary special motion for specific surface generations, and the tool post which allows for the mounting of the cutting tool.

The motion from the spindle motor is communicated to the carriage through a lead screw. Engagement of the lead screw with the carriage is through the use of a half nut. Though the lead screw could be used for feeding the cutting tool in a direction parallel to the axis of rotation, many a times a separate feed rod is provided for this function. The main reason is that the lead screw would be more accurate and would be sparingly used only for thread cutting, such that it maintains its accuracy. For routine feeding, the feed rod is used.

4.2.1 Lathe Specifications

In order to specify a lathe, a number of parameters could be used based on the specific application. However, the major elements used for specification should invariably be based on the components that would be manufactured in the lathe. Thus the following are the basic elements generally specified for the capability of the lathe machine (Fig. 4.3).

- distance between centres—this specifies the maximum length of the job that can be turned in the lathe
- Swing over the bed—this specifies the maximum diameter of the job that can be turned in the lathe machine, generally restricted to small length jobs.
- Swing over the cross slide—this specifies the maximum diameter of the job that can be turned in the lathe machine with the job across the cross slide, which is generally the case.

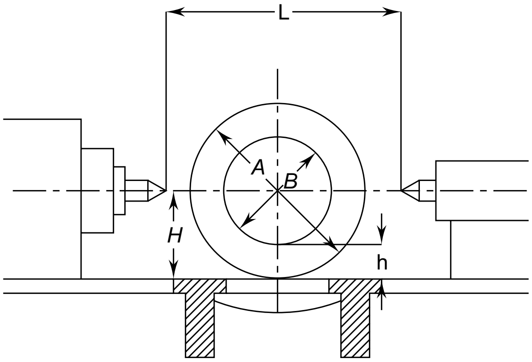


Fig. 4.3 *Capacity specifications for a lathe*

Though the above give the basic capacity of the machine as shown in Fig. 4.3, there are a number of other factors that should also be specified to fully describe the lathe machine. They are:

- horse power of the motor
- cutting speed range
- feed range
- screw cutting capacity
- Accuracy achievable
- spindle nose diameter and hole size

Typical specifications of some centre lathes are given in Table 4.1.

Table 4.1 **Specifications of Centre Lathes**

Centre Height, mm	250,300	375,450	525,600	750,900
Bed Width, mm	325,375	450,550	650,750	900,1050
Sizes Available, mm	1650 to 4200	2400 to 9600	2400 to 9600	2400 to 9600
Distance Between Centres (mm)	500 to 3100	1000 to 8200	800 to 8000	
Power Capacity, H.P.	3	5	7.5/10	10/15
Spindle speed Range	30 to 550	30 to 350	15 to 200	15 to 200

Further specifications would be based on the accessories used with the machine tool and their respective capabilities.

4.3 AIDS FOR SUPPORT AND LOCATION

The work holding devices normally used should have the following provisions:

- suitable location
- effective clamping
- support when required

The most common form of work holding device used in a lathe is the chuck. Chucks come in various forms with a varying number of jaws. Of these the three jaw chuck or the self-centring chuck as shown in Fig. 4.4 is the most common one. The main advantage of this chuck is the quick way in which the typical round job is centred. All the three jaws would be meshing with the flat scroll plate. Rotating the scroll plate through a bevel pinion would move all the three jaws radially inward or outward by the same amount. Thus, the jaws will be able to centre any job, whose external locating surface is cylindrical or symmetrical, like hexagonal. Though it is good for quick centring, it has limitations in terms of the gripping force and also the accuracy is gradually lost due to the wear of the mating parts.

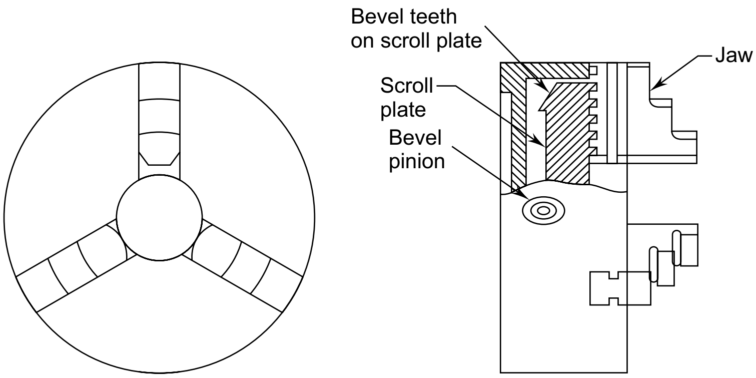


Fig. 4.4 *3-jaw chuck and principle of operation*

The independent jaw chuck has four jaws, which can be moved in their slots independent of each other (Fig. 4.5), thus clamping any type of configuration. Since each of these jaws could move independently any irregular surface could be effectively centred. Better accuracy in location could be maintained because of the independent movement. However more time is spent in fixturing a component in a 4-jaw chuck compared to the 3-jaw chuck. This is generally used for heavy work pieces and for any configuration.

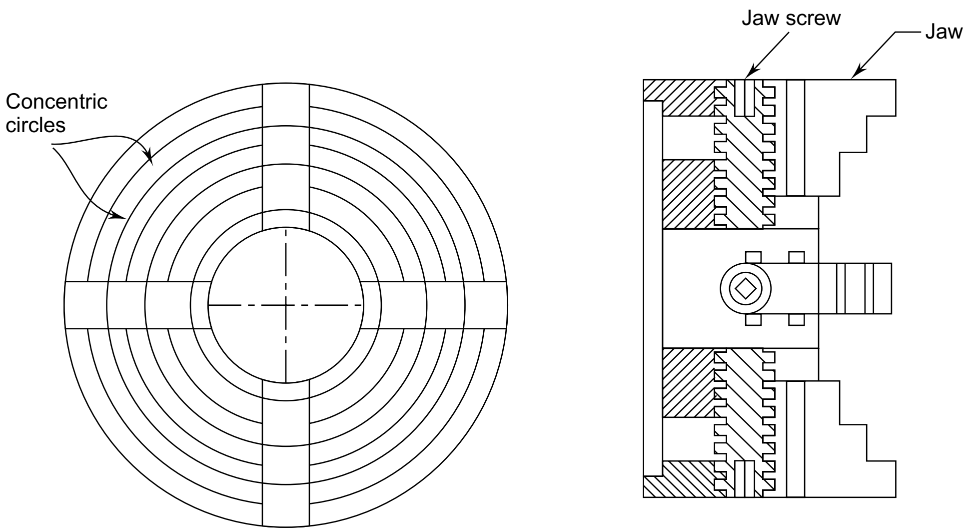


Fig. 4.5 *4-jaw chuck*

The jaws in a 4-jaw chuck could be reversed for clamping large diameter work pieces as shown in Fig. 4.6. The soft jaws are sometimes used in these chucks for clamping surfaces of a component whose surface is already finished and which is likely to be disfigured by the hard surface of the normal jaws used in them.

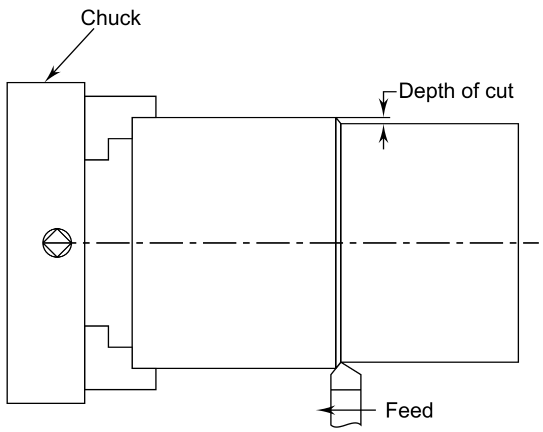


Fig. 4.6 *Chuck and reverse jaw usage*

The 3-jaw and 4-jaw chucks would normally be suitable for short components. However in the case of long components, supporting is done at only one end, because in the case of chucks would make it deflect under the influence of the cutting force. In such cases the long work pieces are held between the centres. The work piece ends are provided with a centre hole as shown in Fig. 4.7. Through these centre holes the centres mounted in the spindle and the tailstock would rigidly locate the axis of the work piece.

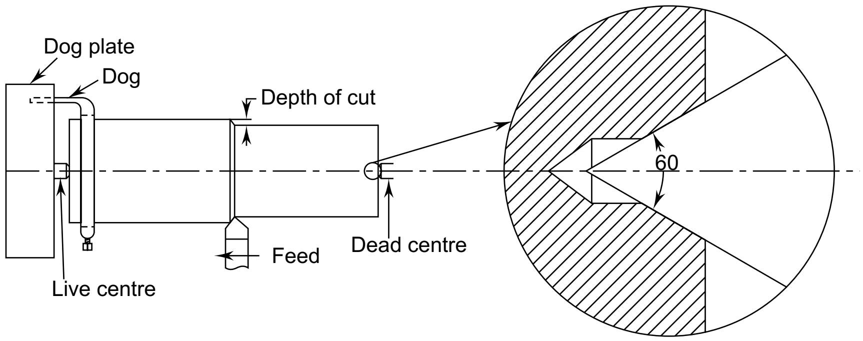


Fig. 4.7 *Centre hole, locating between centres*

Centres as shown in Fig. 4.7 would be able to locate the central axis of the work piece, however would not be able to transmit the motion to the work piece from the spindle. For this purpose, generally, a carrier plate and a dog, as shown in Fig. 4.8, are used. The centre located in the spindle is termed live centre while that in the tailstock is termed the dead centre. The shank of the centre is generally finished with a Morse taper which fits into the tapered hole of the spindle or tailstock.

Live centre rotates with the work piece, and hence it remains soft. Whereas the dead centre does not rotate, it is hardened as it forms the bearing surface. However, in case of heavier work pieces the relative movement between the work piece and the dead centre causes a large amount of heat to be generated. In such cases, a revolving centre is used. In this the centre is mounted in roller bearings thus it rotates freely reducing the heat generation at the tailstock end. In cases where a facing operation is to be carried out with centres, a half centre would sometimes be used.

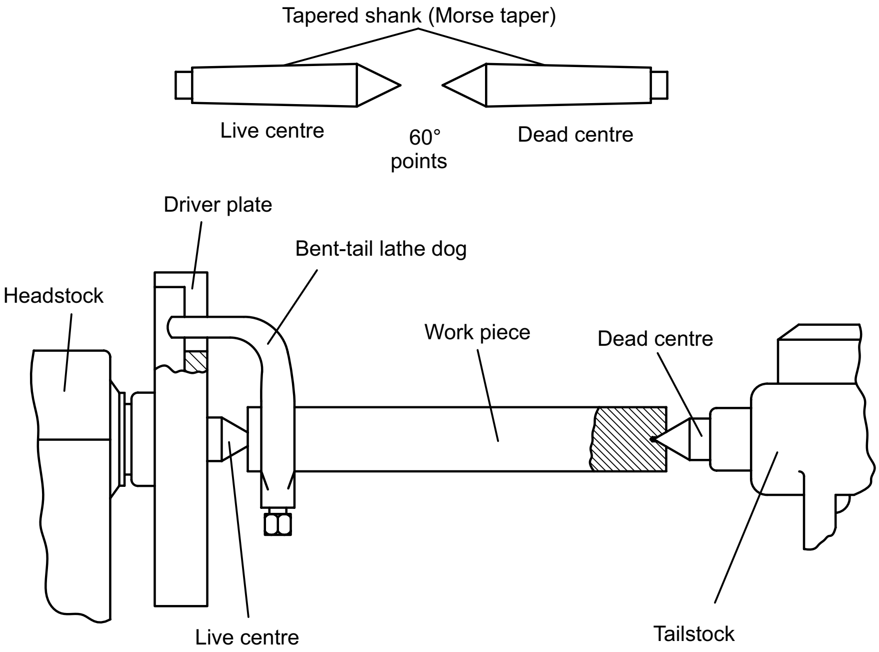


Fig. 4.8 *Dog carrier and revolving centre, half centre, and steady*

Some of the precautions to be observed during the use of centres are:

- The centre hole in the work must be clean and smooth and have an angle of 60° bearing surface, large enough to be consistent with the diameter of the work. For heavier work this may be changed to 75° or 90°.
- The bearing must take place on the countersunk surfaces and not on the bottom of the drilled hole.

When the job becomes very long, it is likely to deflect because of its own weight as well as due to the cutting force acting away from the supports provided at both the ends. A steady is used for supporting the work piece at the maximum deflection point. Sometimes a steady is fixed to the carriage, so that it moves with the tool thus effectively compensating for the acting cutting force.

For odd shaped components a faceplate is more widely used where the locating and clamping surfaces need not be circular. This has radial slots on the plate as shown in Fig. 4.9 for the purpose of locating the component and clamped by means of standard clamps. The method is somewhat similar to the clamping of work pieces on a milling machine table using the T-slots on the table. However, in view of the fact that the faceplate rotates, it is possible that the component is off centre. This would cause vibrations due to the mass unbalance. A balancing mass would therefore be provided as shown in Fig. 4.9. Sometimes angle plates along with the faceplate may have to be used for typical components where the locating surface is perpendicular to the plane of the faceplate as shown in Fig. 4.9.

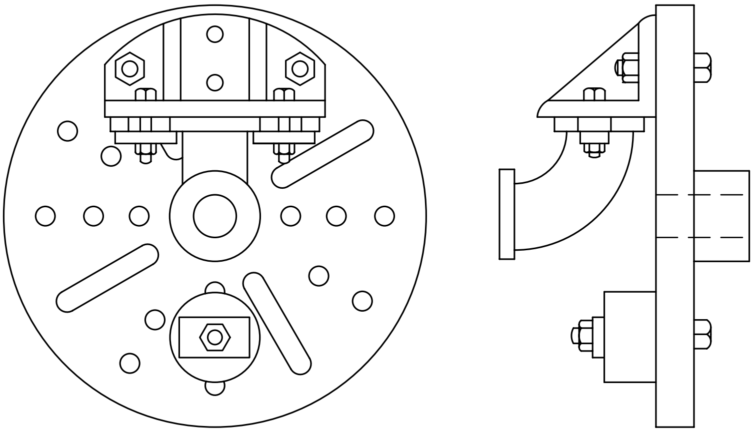


Fig. 4.9 *Face plate*

For holding components with locating holes, for the purpose of generating external surfaces, a mandrel is generally used. Various types of mandrels used are shown in Fig. 4.10.

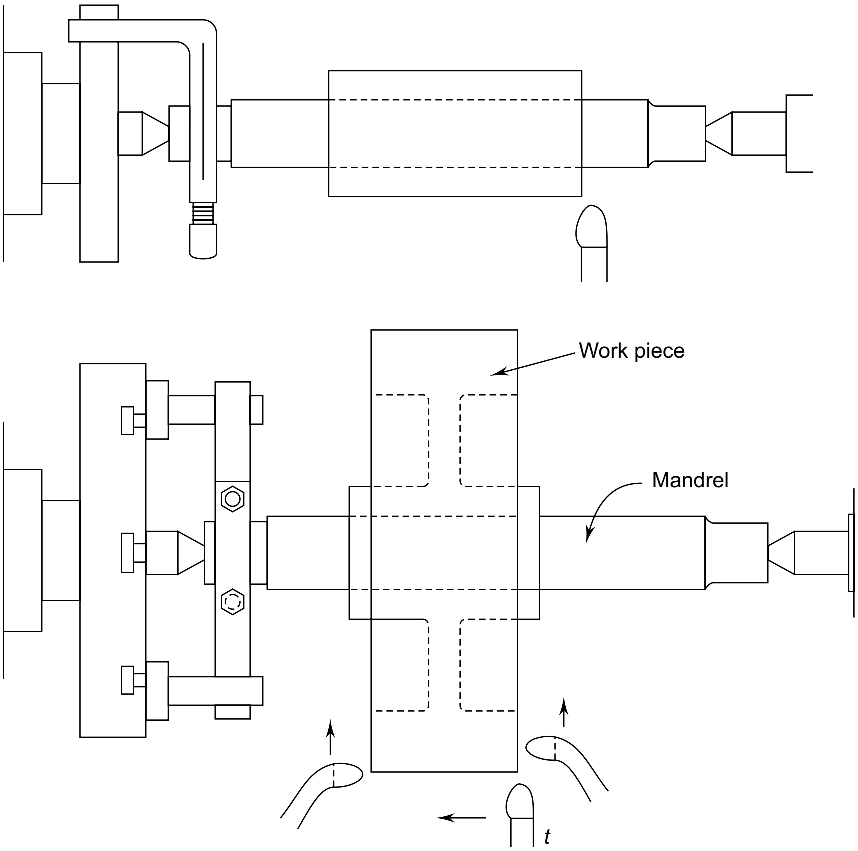


Fig. 4.10 *Type of mandrels used for work holding*

The given work holding devices are more useful for general purpose work. However, for production work such devices would mean that considerable time is spent in locating and holding individual work pieces. In production machine tools it is therefore necessary to use work holding devices which require very less time for clamping purpose. Collet is one such device, which provides good clamping accuracy with very little time required for clamping and unclamping.

Collet has a sleeve as the holding part, which is slit along the length at a number of points along the circumference as shown in Fig. 4.11. When uniform pressure is applied along the circumference of the sleeve, these segments would elastically deflect and clamp the component located inside. Since the deflection of the sleeve is in the elastic range, it would spring back once the clamping pressure is removed, thus releasing the component located inside. This clamping method is very accurate and fast in operation, in addition to the fact that it would hold the work uniformly over the entire circumference. However, the size range in which a collect becomes operational is very small in view of the limit on the elastic deformation allowed. Thus a large number of chucks need to be maintained in the inventory to cater to the variety of diameters to be worked in the machine tool. These are normally used for large scale production where saving in terms of the locating and clamping time would be desirable.

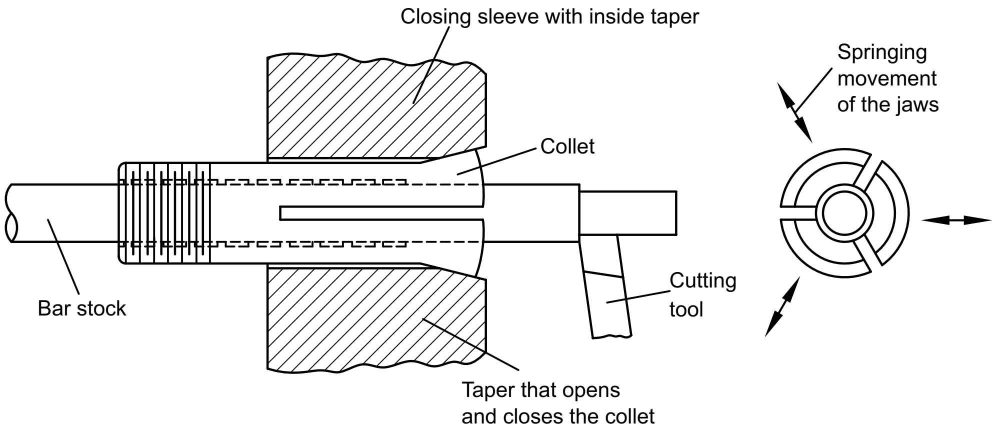


Fig. 4.11 *Collet chuck principle*

Steady Rest

The steady rest, also called a centre rest, should be used when turning or boring long work pieces. It is also used for internal threading operations where the work piece projects a considerable distance from the chuck or faceplate. The steady rest (Fig. 4.12) is clamped to the lathe bed at the desired location (where the maximum deflection is likely to occur) and supports the work piece with three adjustable jaws. The jaws must be carefully aligned to properly locate the axis of rotation of the work piece. The area of contact must be lubricated frequently. The top section of the steady rest swings away from the bottom section to permit removal of the work piece without disturbing the jaw setting. For machining with very high cutting speeds, steady rests as shown will generate substantial heat, so they will be provided with a ball or roller bearings built into the jaws. Another problem with this type of rest is that since the carriage cannot pass it, the work piece needs to be turned in two set-ups by reversing after the first portion is machined.

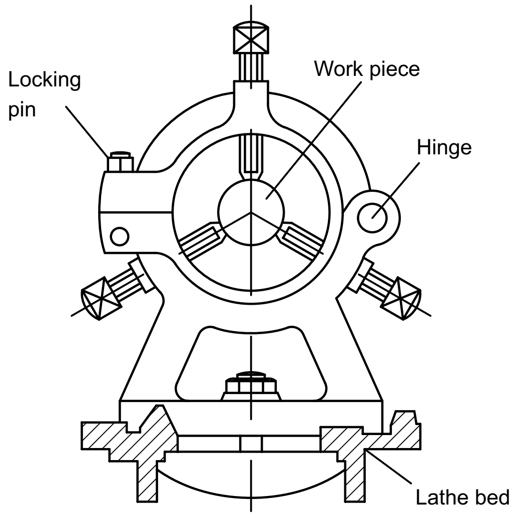


Fig. 4.12 *Steady rest used to support long, slender work pieces in centre lathe*

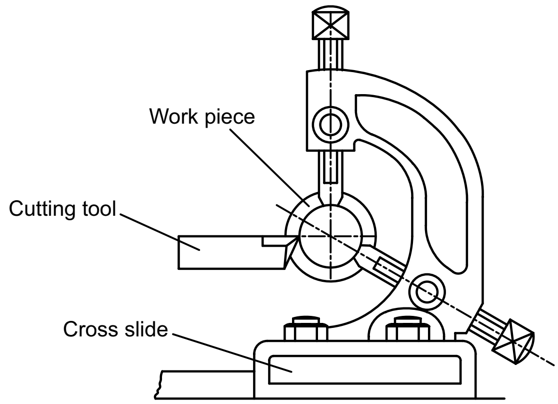


Fig. 4.13 *Follower rest used to support long, slender work pieces in centre lathe which moves with the carriage*

Follower Rest

The follower rest (Fig. 4.13) on the other hand has two jaws that bear against the work piece. The follower rest is fastened to the lathe carriage so that it will follow the cutting tool and bear upon the portion of the work piece that has just been turned. The cut must first be started and continued for a short longitudinal distance before the follower rest may be applied. The rest is generally used only for straight turning and for threading long, thin work pieces. Steady rests and follower rests can be equipped with ball-bearing surfaces on the adjustable jaws. These types of rests can be used without excessive lubricant or having to machine a polished bearing surface.

4.4 CUTTING TOOLS

4.4.1 Cutting Tool Geometry

The size of the tool is generally square or rectangular in cross section. The shank is that part of the tool on one end of which the cutting point is formed. It is supported in the tool post of the lathe. The base is that part of the shank which bears against the support and bears the tangential force of the cut.

In Chapter 2 the orthogonal cutting tool geometry is discussed in a simplified manner to suit the discussion on metal cutting principles. The detailed turning tool geometry as given in Fig. 4.14 has a number of angles that fully describe it. The details are given in Chapter 2.

These individual angles have considerable influence on the cutting performance. They have to be judicially chosen for a given application. For example the side cutting edge angle controls the width and thickness of the chips produced. A very large angle means that the uncut chip thickness reduces resulting in higher specific cutting resistance. When it approaches zero, the radial component of the cutting force is minimum while the axial component is maximum. This is generally the preferred condition since the vibration resistance is at its best in this condition. The recommended tool angles for various types of work and tool material combinations are given in Tables 4.2 and 4.3.

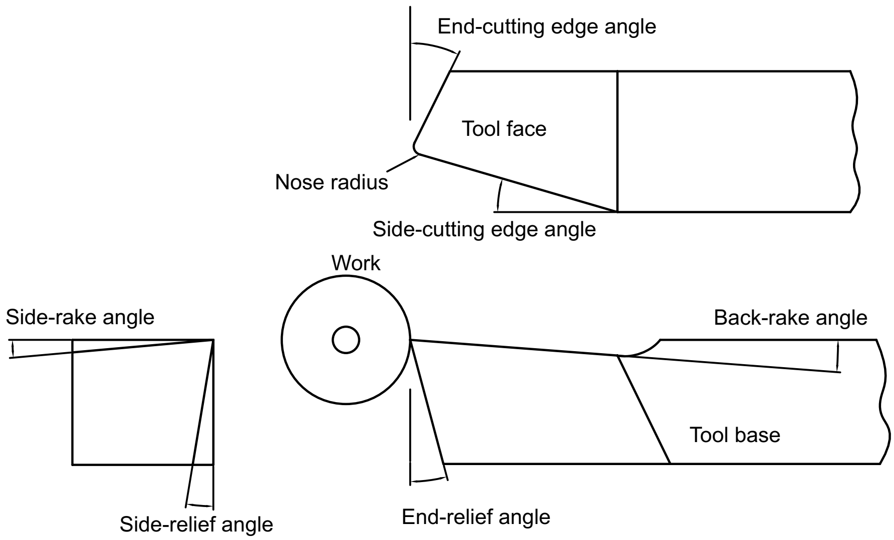


Fig. 4.14 *ASA Turning tool geometry*

Table 4.2 **Recommended tool angles in degrees for high speed steel cutting tools**

Work Material	Back Rake Angle	Side Rake Angle	Side Relief Angle	Front Relief Angle	Side Cutting Edge Angle	End Cutting Edge Angle
Steel	8 – 20	8 – 20	6	6	10	15
Cast steel	8	8	6	6	10	15
Cast iron	0	4	6	6	10	15
Bronze	4	4	6	6	10	10
Stainless steel	8 – 20	8 – 20	6	6	10	15

Table 4.3 **Recommended tool angles in degrees for carbide cutting tools**

Work Material	Back Rake Angle	Side Rake Angle	Side Relief Angle	End Relief Angle
Aluminium and magnesium alloys	0 – 10	10 – 20	6	6
Copper	0 – 4	15 – 20	6 – 8	6 – 8
Brass and bronze	0 – 5	–5 – 8	6 – 8	6 – 8
Cast iron	–7 – 0	–7 – 6	5 – 8	5 – 8
Plain carbon steels	–7 – 0	–7 – 6	5 – 8	5 – 8
Alloy steels	–7 – 0	–7 – 6	5 – 8	5 – 8
Stainless steels	–7 – 0	–7 – 6	5 – 8	5 – 8
Titanium alloys	–5 – 6	–5 – 0	5 – 8	5 – 8

4.4.2 Different Types of Tools Used

A large variety of tools are used in centre lathes in view of the large types of surfaces that are generated. The actual type of tool used depends upon the surface of job being generated as well as the work piece. A variety of tools used for normal generation of external surfaces are shown in Fig. 4.15. Similar tools will be available for generating the internal surfaces as well. The choice of a particular tool depends upon the actual surface to be generated as shown in Fig. 4.16.

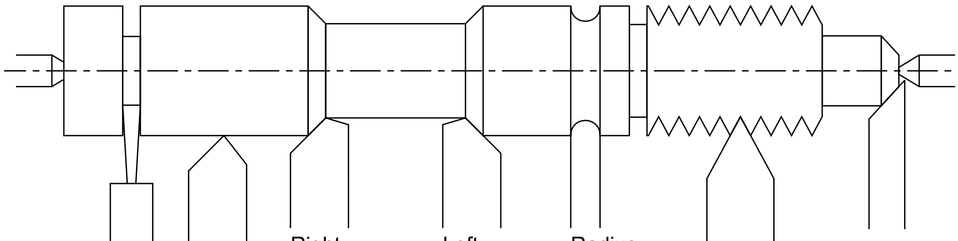


Fig. 4.15 *Different kinds of tools used for external surfaces*

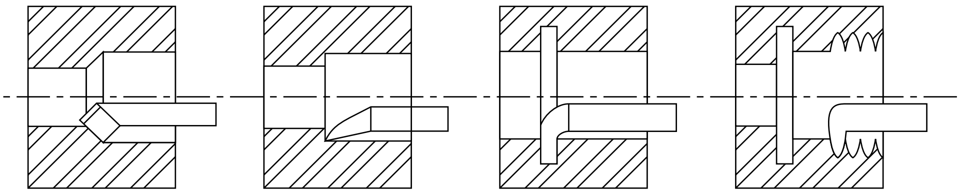


Fig. 4.16 *Different kinds of tools used for internal surfaces*

The tools have primary cutting edge by means of which the direction of the movement of the tool for removing the metal is indicated. The direction is termed as right or left depending upon the movement direction. The tool is termed right when it cuts during the movement towards the head stock. It is derived by the fact that when the right palm is placed on the tool, the direction of the thumb indicates the direction of tool motion. Similarly the left hand tool cuts during its motion in the direction of tailstock.

The variations in the type of tools are indicative of the variety applications for which these tools are used. The large variety is needed because of the large number of surfaces to be generated. For example, by the side cutting edge angle of the tool, it is possible to know the application of pockets for which the tool could be used. Similarly some tools would be required for facing applications while others are used for boring.

Form tools are generally used for machining short profiles. Typical form tools are shown in Fig. 4.17. The Fig. 4.17 shows two types of form tools that are generally used. The circular form tool shown in Fig. 4.17(a) is held in a holder mounted on the cross slide. The centre of the tool should be mounted slightly above the centre of the work piece in order to get a clearance angle such that the tool will not rub the work. The circular form tool has a long life as it can be continuously sharpened as shown in the Fig. 4.17(a) over 270° of the tool. The straight or flat form tool shown in Fig. 4.17(b) is the simplest type. It is sharpened by grinding the top face that reduces the strength of the tool.

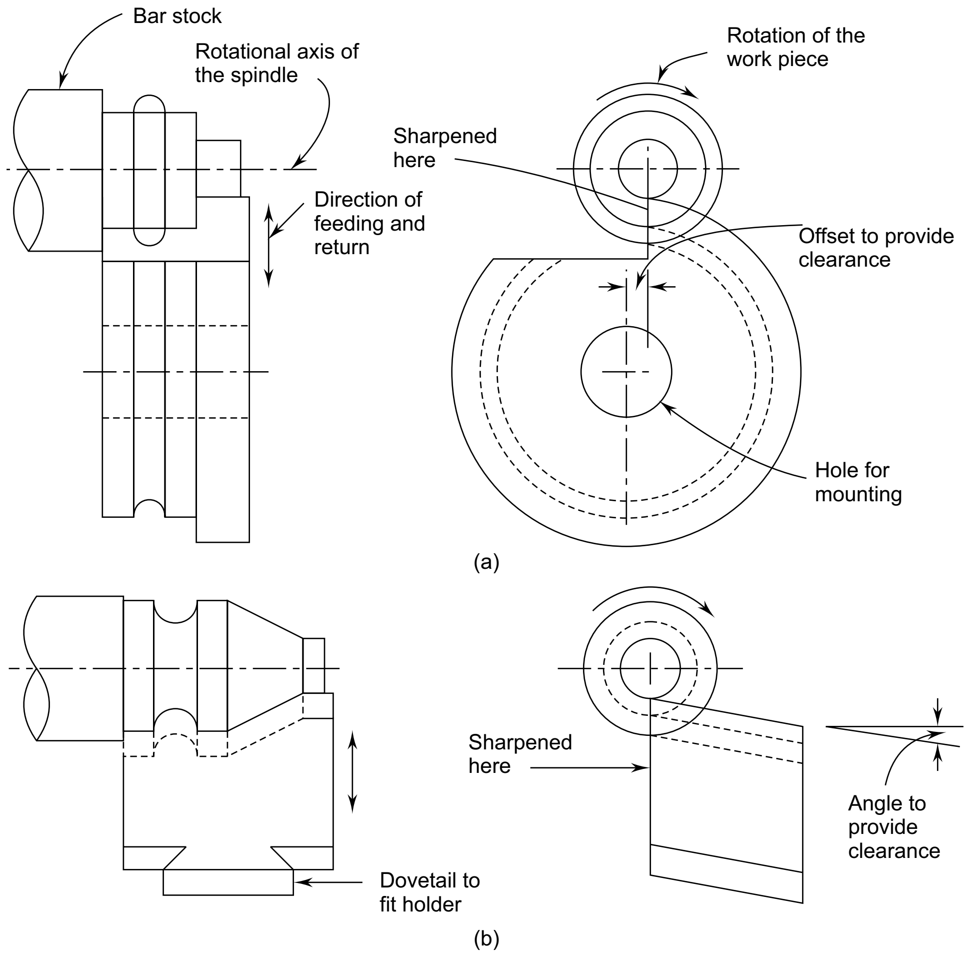


Fig. 4.17 *Form tool types used in centre lathe, (a) Circular form tool, (b) Straight form tool*

As of now a majority of the tools used are of the cemented carbide type with indexable insert type. It therefore becomes necessary to understand the ISO coding systems for these to be able to easily make the selection. The ISO coding system (as per ISO 1832–1991) for tungsten carbide inserts and external turning tools is shown in extracted form in Fig. 4.18 and 4.19.

The actual selection of the tools for a particular application has to carefully match the geometry. Generally the manufacturer's catalogue provides such application information. For example, referring to Fig. 4.20 the use of PDJNR type tools is for the external turning requires that the maximum contour angle be 30° as shown in Fig. 4.20. It also shows the other type of features that can be machined by the tool.

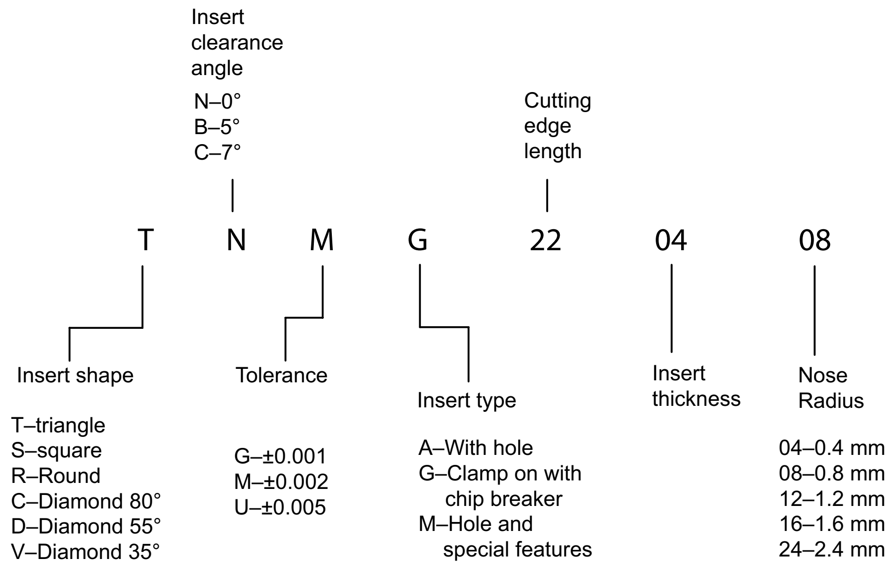


Fig. 4.18 *The ISO coding system for tungsten carbide inserts used in turning*

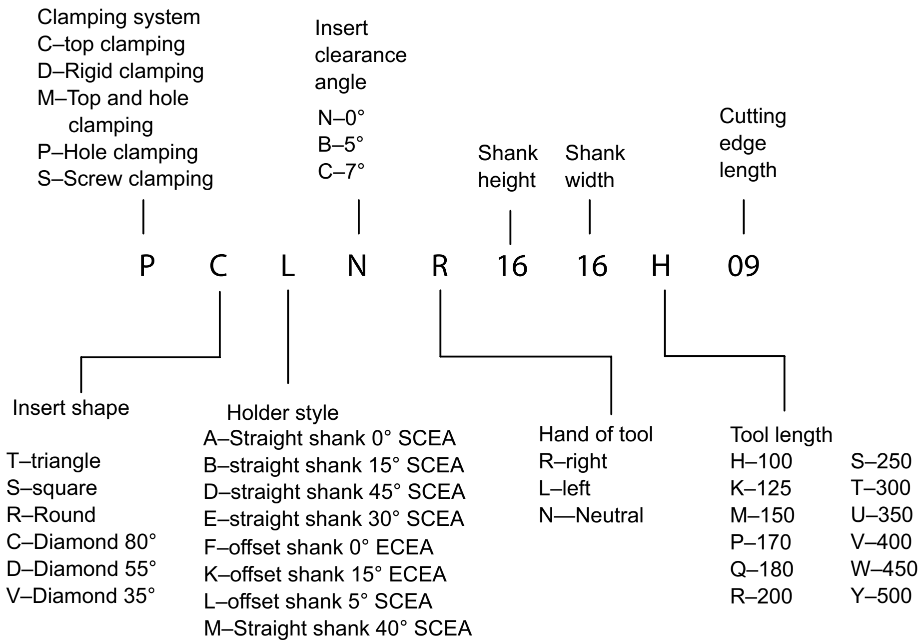


Fig. 4.19 *The ISO coding system for tungsten carbide turning tool holders used in external turning (SCEA - side cutting edge angle, ECEA - end cutting edge angle)*

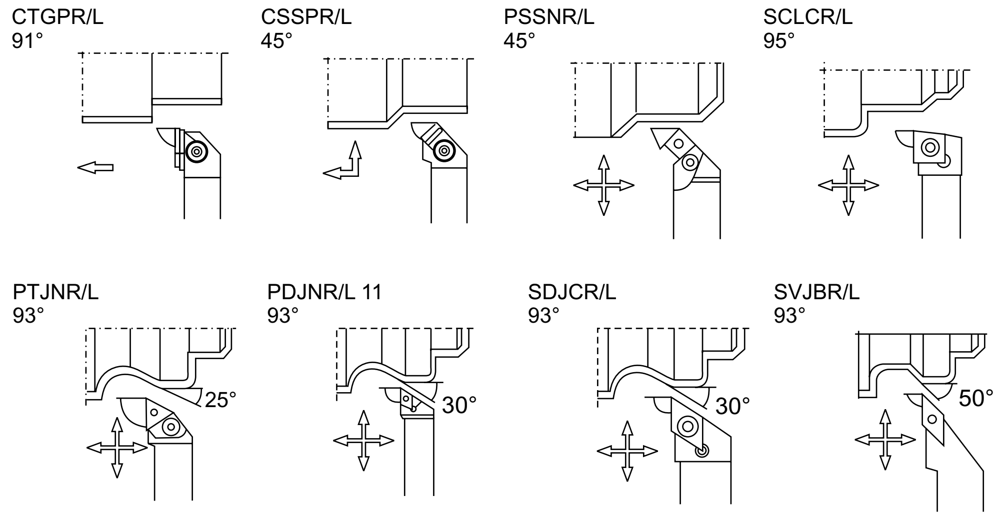


Fig. 4.20 *Typical contour capability of external turning tools (Courtesy Seco Tools, Germany. Redrawn from Seco Catalogue)*

4.4.3 Errors in Tool Setting

The tool should be set exactly at the centre of the work piece for proper cutting. If the tool is kept below or above the work piece, the tool geometry gets affected as shown in Fig. 4.21. For example, if the tool is set at a position below the work piece centre by a distance h, then the expected changes are:

$$R = \sqrt{r^2 - h^2} \tag{1}$$

where R is the actual radius of the component produced, and
r is the radius set

$$\alpha = \sin^{-1}\left(\frac{h}{r}\right) \tag{2}$$

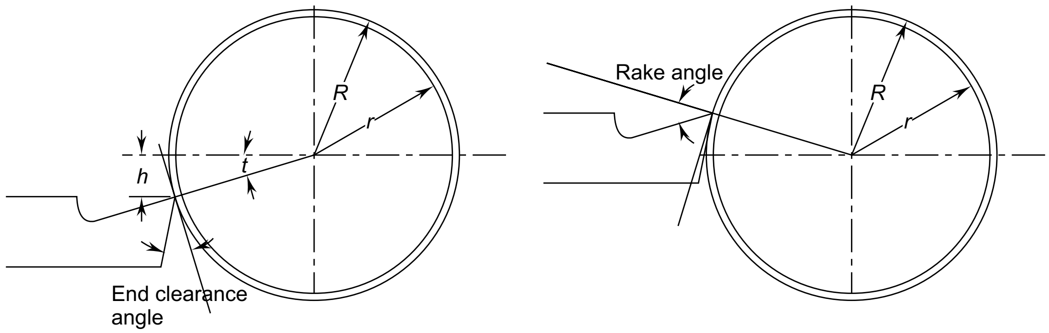


Fig. 4.21 *Tool setting errors*

The above angle decreases the actual rake angle a while the clearance angle increases by the same amount. Thus the cutting forces increase because of the reduction in the rake angle. In the case of setting the tool above the centre, it causes the rake angle to increase, while the clearance angle reduces. This would cause for more rubbing to take place in the flank face.

The following example shows the magnitude of error that is involved in a cutting tool setting.

Example 4.1

While turning a diameter of 90 mm, the turning tool is set below centre line by an amount equal to 5 mm. Find out the change in the effective cutting tool geometry.

Solution $$\alpha = \sin^{-1}\left(\frac{5}{45}\right) = 6.37937° \qquad (3)$$

If the actual back rake angle is 10°, the effective rake angle would be (10 – 6.38) = 3.62°.

Similarly if the clearance angle is 5°, the effective clearance angle would be (5 + 6.38) = 11.38°.

If the tool is set at a point above the centre line, there would be a corresponding increase in the rake angle by the same amount, while the clearance angle would be (5 – 6.38) = –1.38°, which would cause rubbing (interference) with the work piece.

4.4.4 Types of Tool Posts

The turret tool post is a swivelling block that can hold many different tool bits or tool holders. Each cutting tool can quickly be swivelled into cutting position and clamped into place using a quick clamping handle. The turret tool post is used mainly for high-speed production operations. The heavy-duty or open-sided tool post is used for holding a single carbide-tipped tool bit or tool holder. It is used mainly for very heavy cuts that require a rigid tool holder.

4.4.5 Chip Control

Chips produced with some ductile materials become very long and often become hazardous to the operator as well as the surface finish of the part produced. Hence most of the tools need a mechanism by which these long chips can be broken. If the rake surface of the tool is flat, the chip will slide over the surface and will not have a chance to break as shown in Fig. 4.22(a). However if a groove is made on the surface as shown in Fig. 4.22(b), then the chip will curl like the figure 9 and then will be broken by the extra curl. An example of the type of chip breaking groove provided for the carbide tools is shown in Fig. 4.23.

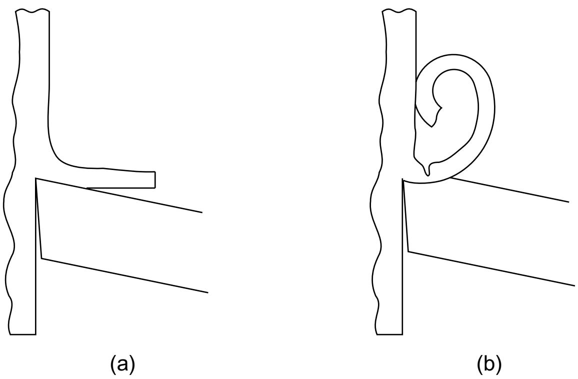


Fig. 4.22 *Chip breaking groove, (a) Cutting with a tool without chip breaking groove; (b) Cutting with chip breaker. Notice the chip being broken by the extra curl taken by the chip.*

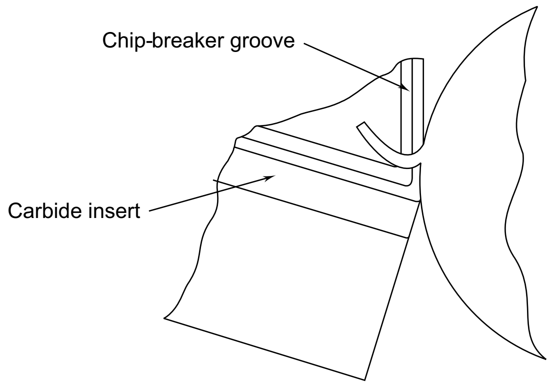


Fig. 4.23 *Chip breaker groove on a tungsten carbide tool bit*

4.5 OPERATIONS PERFORMED IN A CENTRE LATHE

As discussed before, in a lathe, it is possible to achieve a large number of surfaces based on the various settings that are possible.

4.5.1 Turning

Turning is by far the most commonly used operation in a lathe. In this the work held in the spindle is rotated while the tool is fed past the work piece in a direction parallel to the axis of rotation. The surface thus generated is the cylindrical surface as shown in Fig. 4.1.

4.5.2 Facing

Facing is an operation for generating flat surfaces in lathes as shown in Fig. 4.24. The feed in this case is given in a direction perpendicular to the axis of revolution. The tool used should have a suitable approach angle so that it would not interfere with the work piece during the tool feeding.

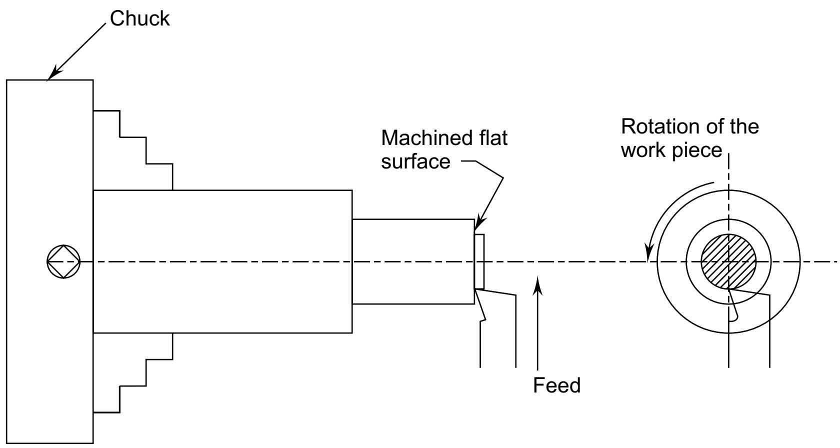


Fig. 4.24 *Facing*

Also the radius of work piece at the contact point of the tool varies continuously as the tool approaches the centre, thus the resultant cutting speed also varies in facing, starting at the highest value at the circumference to almost zero near the centre. Since the cutting action and the surface finish generated depend upon the actual cutting speed, the finish becomes very poor as the tool approaches the centre. Also while choosing the rotational speed of the work piece due care has to be taken of this fact.

4.5.3 Knurling

Knurling is a metal working operation done in a lathe. In this a knurling tool having the requisite serrations is forced on to the work piece material, thus deforming the top layers as shown in Fig. 4.25. This forms a top surface, which is rough and provides a proper gripping surface.

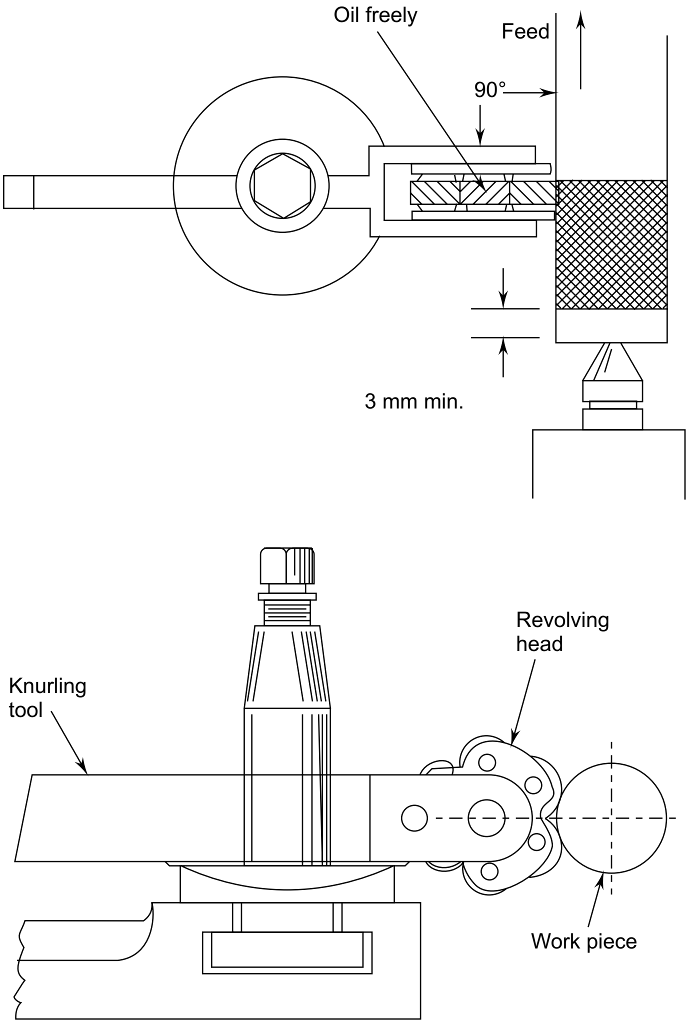


Fig. 4.25 *Knurling*

4.5.4 Parting

Parting and grooving are similar operations. In this a flat nosed tool would plunge cut the work piece with a feed in the direction perpendicular to the axis of revolution as shown in Fig. 4.26. This operation is generally carried out for cutting off the part from the parent material. When the tool goes beyond the centre, the part would be severed. Otherwise a rectangular groove would be obtained. It is also possible, in similar operation, to use a special form of tool to obtain the specific groove shape.

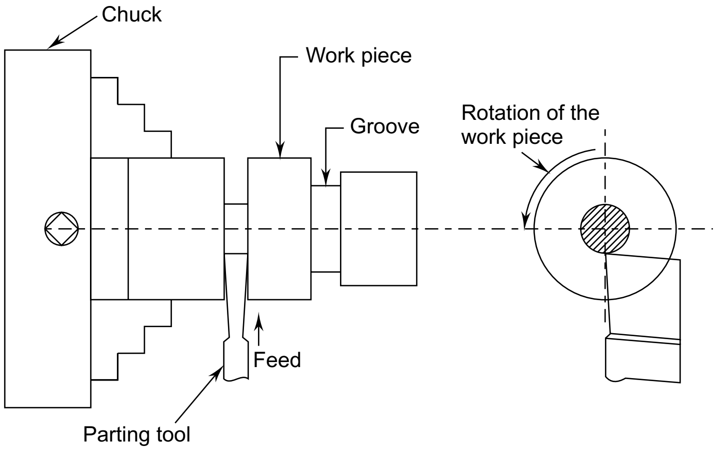


Fig. 4.26 *Parting tool in operation*

4.5.5 Drilling

Drilling is the operation of making cylindrical holes into the solid material as shown in Fig. 4.27. A twist drill is held in the quill of the tailstock and is fed into the rotating work piece by feeding the tailstock quill. Since the work piece is rotating, the axis of the hole is very well maintained, even when the drill enters at an angle initially. The same operation can also be used for other hole making operations such as centre drilling, counter sinking and counter boring. This operation is limited to holes through the axis of rotation of the work piece and from any of the ends.

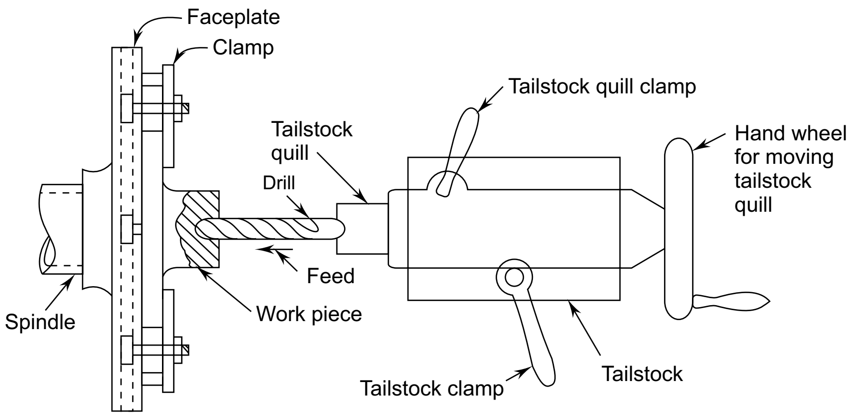


Fig. 4.27 *Drilling operation in a lathe*

4.5.6 Boring

Boring is the operation of enlarging a hole already made by a single point boring tool termed as boring bar as shown in Fig. 4.28. The operation is somewhat similar to the external turning operation. However, in view of the internal operation, it is more restricted. The cutting forces experienced are somewhat more than the external operation. Also the tool used is less rigid compared to turning tool and as a result it cannot withstand the large cutting forces. Thus the process parameters used are somewhat lower than those used for turning. Boring is used for generating an accurate hole with good surface finish.

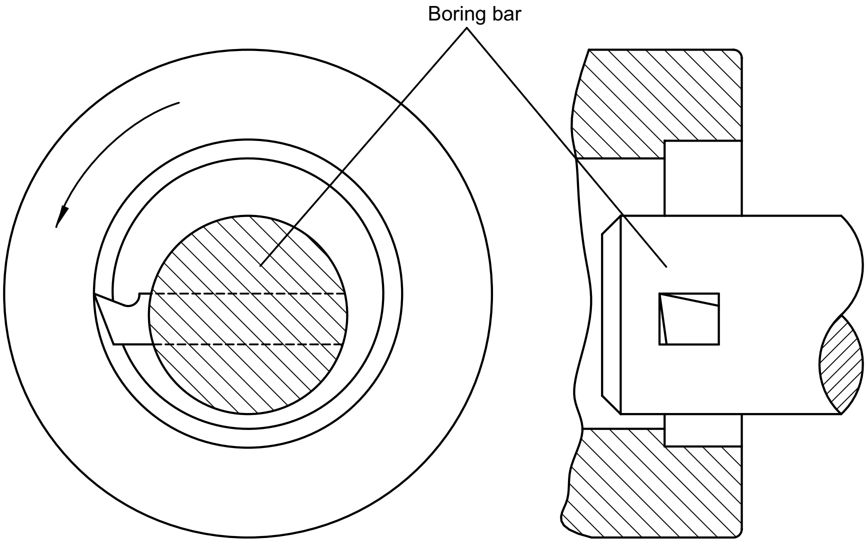


Fig. 4.28 *Boring operation in a lathe*

4.6 TAPER TURNING METHODS

Cutting tapers on a lathe is one of the most common applications. A number of methods are available for cutting tapers in a lathe. They are:

- using a compound slide
- using form tools
- offsetting the tailstock
- using taper turning attachment

The compound slide is an auxiliary slide underneath the tool post and above the carriage. It is possible to swivel the compound slide to the desired angle of the taper as shown in Fig. 4.29 for cutting the tapers. The tool is then made perpendicular to the work piece and feed is given manually by the operator. Some of the features of this method are:

- Short and steep tapers can be easily done.
- Limited movement of the compound slide.
- Feeding is by hand and is non-uniform. This is responsible for low productivity and poor surface finish.

Another method that is normally used for production applications is the use of special form tool for generating the tapers as shown in Fig. 4.30. The feed is given by plunging the tool directly into the work. This method is useful for short tapers, where the steepness is of no consequence, such as for chamfering.

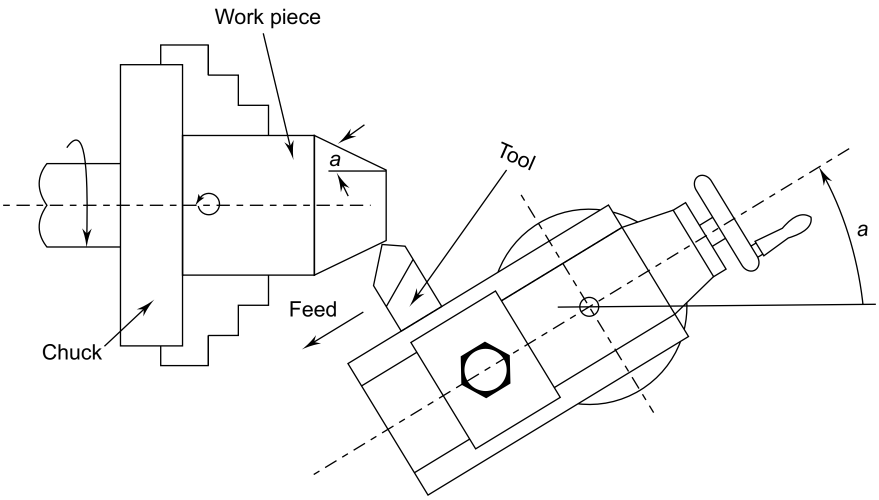


Fig. 4.29 *Compound slide method for taper turning*

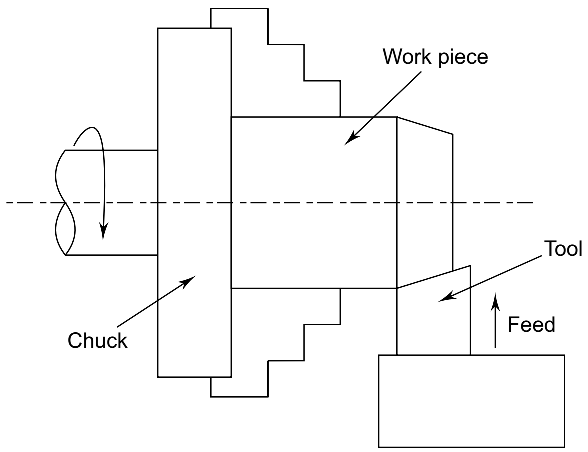


Fig. 4.30 *Taper turning using Form tools*

When turning long tapers with form tools, the tool is likely to chatter (vibrate) resulting in poor surface finish. However, care of the form tools, particularly for regrinding is a careful exercise. The cutting edge should be perfectly straight for good accuracy.

Still another method sometimes used is the method of offsetting the tailstock from the centre position. By offsetting the tailstock, the axis of rotation of the job is inclined by the half angle of taper as shown in Fig. 4.31. The feed to the tool is given in the normal manner parallel to the guideways. Thus the conical surface is generated.

Referring to Fig. 4.31, the offset can be calculated as follows:

$$\sin \alpha = \frac{BC}{AB} \tag{4}$$

$$S = AB \sin \alpha = L \sin \alpha \tag{5}$$

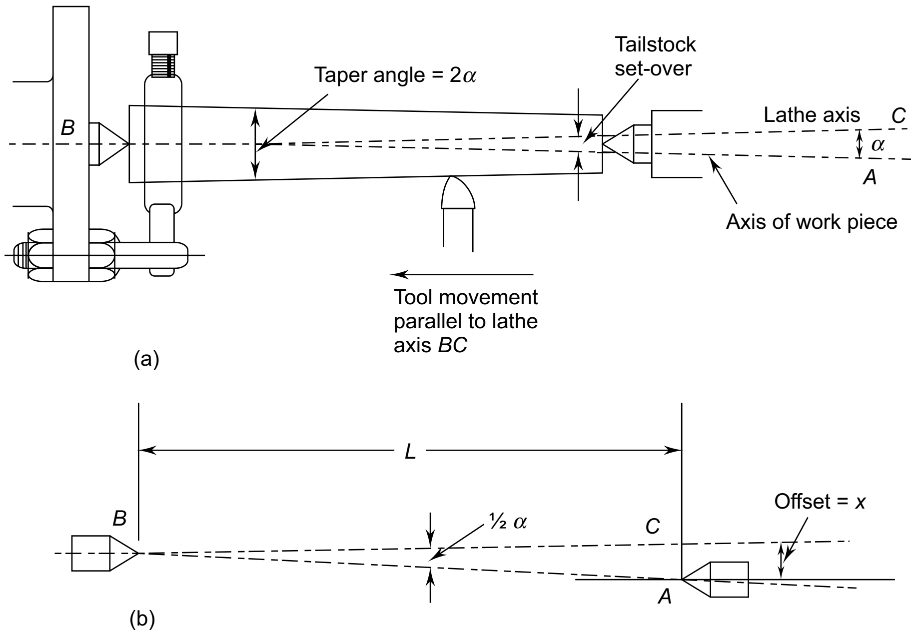


Fig. 4.31 *Tail stock offset*

If α is very small, then we can approximate

$$\sin \alpha \approx \tan \alpha \approx \frac{D-d}{2l} \tag{6}$$

$$\therefore \quad S = L\frac{(D-d)}{2l} \tag{7}$$

This is the most general situation where the taper is to be obtained over a small portion of the length, l of the job while the actual length of the work piece, L could be long. However when they are equal

$$\text{if} \quad L = 1, \text{ then } S = \frac{(D-d)}{2} \tag{8}$$

The offset, S that is possible is generally limited, and as such this method is suitable for small tapers over a long length. The disadvantage is that the centres are not properly bearing in the centre holes and as such there would be non-uniform wearing taking place.

Example 4.2

Find the setting required turning a taper of 85 mm diameter to 75 mm diameter over a length of 200 mm, while the total length of the job is 300 mm between centres. Tailstock offset is to be used for generating the prescribed taper.

Solution $$S = 300\frac{(75-60)}{2 \times 200} = 11.25 \text{ mm} \tag{9}$$

Still another method for turning tapers over a comprehensive range is the use of taper turning attachment. In this method a separate slideway is arranged at the rear of the cross slide. This slide can be rotated at any angle to be setup. The block that can slide in this taper slide way is rigidly connected to the cross slide as shown in Fig. 4.32. The cross slide is made floating by disconnecting it from its lead screw. As the carriage moves for feeding, the block moves in the inclined track of the slide, it gets the proportional cross movement perpendicular to the feed direction, the cross slide and in turn the cutting tool gets the proportional movement. Thus the tool tip follows the taper direction set in the attachment. However, in this condition, the cross slide cannot be used for normal turning operations.

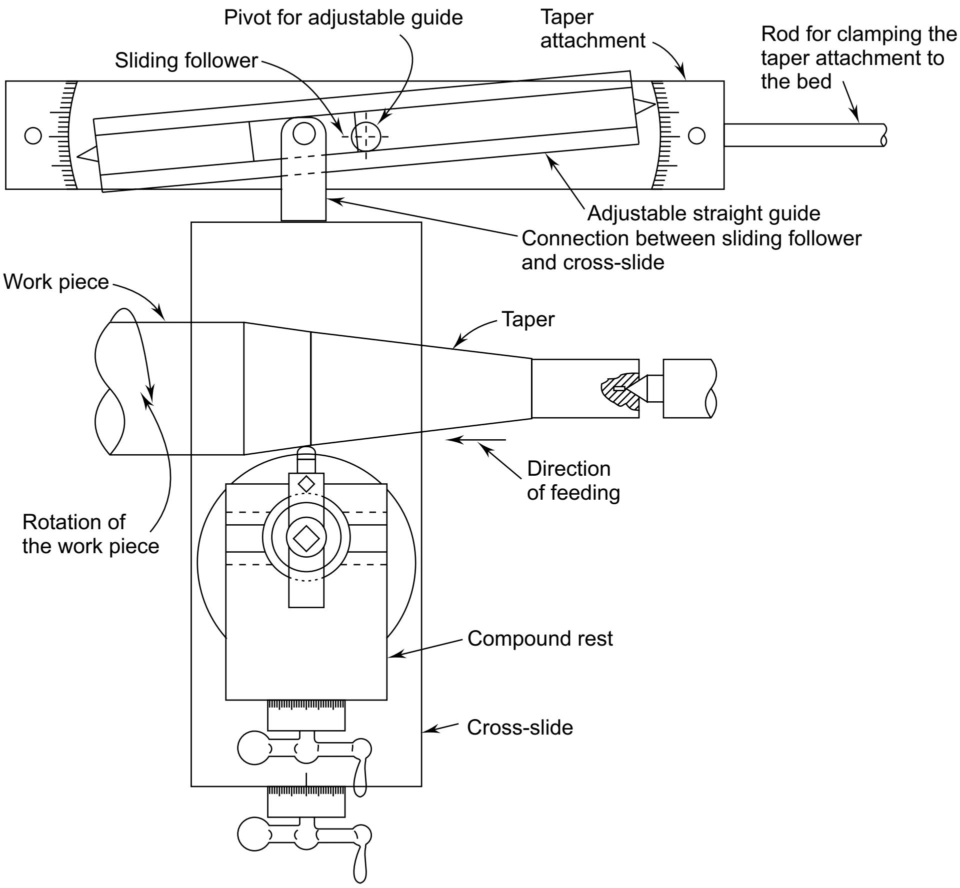


Fig. 4.32 *Taper turning attachment*

This method is most commonly used for a range of tapers. Setting of the cutting tool is most important, since the form of the taper cut would be affected by the position of the cutting tool with respect to the work piece. For example consider the case of a cutting tool set below the centre line of the work piece as shown in Fig. 4.33. The work piece is to have *r* as radius at the small end and *R* as the radius at the big end, with *L* as the length of the taper. The original taper angle, a to be produced is given by

$$\tan \alpha = \frac{R - r}{1} \tag{10}$$

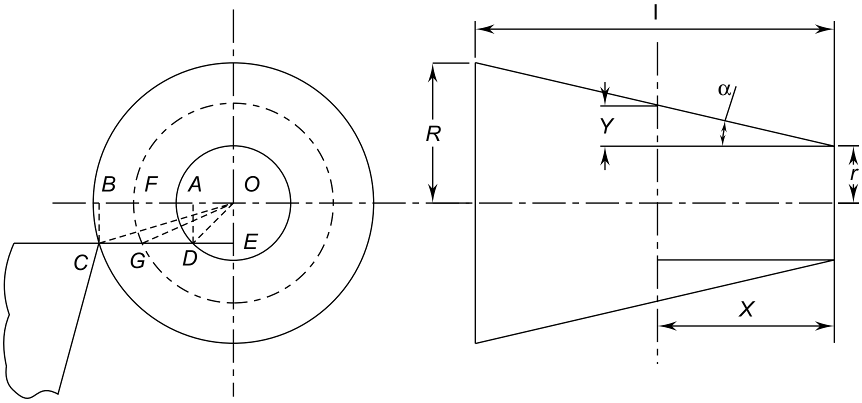


Fig. 4.33 *Possible error in taper turning using taper turning attachment because of the wrong positioning of the cutting tool.*

where R = radius of the work at the big end
r = radius of the work at the small end
l = length of the taper

$$DE = \sqrt{OD^2 - OE^2} = \sqrt{r^2 - h^2} \quad (11)$$

If we consider a section along the length at a distance of x, from the small end, the radius, y at that point is given by

$$y^2 = r^2 + x^2 \sin^2 \alpha \quad (12)$$

This can be rearranged as

$$\frac{y^2}{r^2} - \frac{x^2 \tan^2 \alpha}{r^2} = 1 \quad (13)$$

This is the equation of the taper produced, which is a hyperbola. Thus, the surface produced by a tool located away from the centre is hyperboloid.

To find the taper angle, we may calculate the radius at the big end for a corresponding small end diameter r as follows. From the Fig. 4.33

$$OC^2 = OE^2 + CE^2 = OE^2 + (CD + DE)^2 \quad (14)$$

$$= h^2 + \left[1 \tan \alpha + \sqrt{r^2 - h^2}\right]^2 \quad (15)$$

Since OC = OB = Radius of big end

AB = OB – r

The taper angle generated is given by

$$\tan \alpha_1 = \frac{AB}{1} \quad (16)$$

$$\tan \alpha_1 = \frac{\sqrt{(1 \tan \alpha + \sqrt{r^2 - h^2}) + h^2}}{1} \quad (17)$$

Example 4.3

While turning a taper using taper turning attachment, the setting was done for 4°, but the tool is set 3 mm below the centre. If the work piece diameter at the small end is 40 mm, calculate the actual taper produced.

Solution The taper angle is 4°, and the big end of the work piece considering the length of the taper to be 100 mm.

$$R = 100 \tan 4° = 6.9927 \text{ mm}$$

$$OC^2 = 3^2 + \left[100 \tan 4 + \sqrt{20^2 - 3^2}\right]^2 = 725.44 \quad (18)$$

$$OC = \sqrt{725.44} = 26.933 \text{ mm} \quad (19)$$

The produced taper angle is therefore

$$\alpha = \tan^{-1}\left[\frac{26.933 - 20}{100}\right] = 3.9657°$$

The error produced is

$$\text{Error in half taper} = 4 - 3.9697 = 0.0303° = 2'$$

4.7 THREAD CUTTING METHODS

Cutting screws is another of the most important tasks carried out in lathes. A typical thread form is shown in Fig. 4.34. There are a large number of thread forms that can be machined in lathe such as Whitworth, Acme, ISO metric, etc.

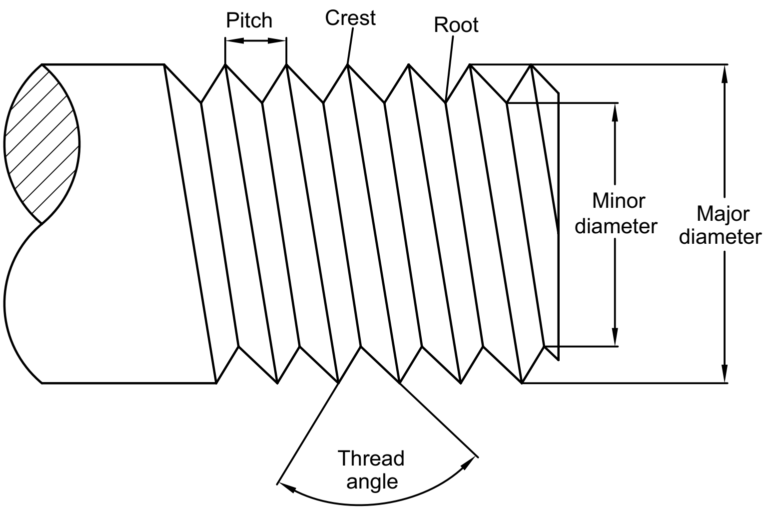


Fig. 4.34 *Simple thread definition*

Thread cutting can be considered as another form of turning since the path to be travelled by the cutting tool is helical. However, there are some major differences between turning and thread cutting. Where as in turning the interest is in generating a smooth cylindrical surface, in thread cutting the interest is in cutting a helical thread of a given form and depth which can be calculated from the formulae for different forms of threads as given in Table 4.3.

TABLE 4.3 Formulae for some common thread forms

Thread Form	Formulae for Calculating the Parameters
British Standard Whitworth (BSW)	Depth = 0.6403 × Pitch Angle = 55° in the plane of the axis Radius at the crest and root = 0.137329 × Pitch
British Association (BA)	Depth = 0.6 × Pitch Angle = 47.5° in the plane of the axis Radius at the crest and root = $\frac{2 \times \text{Pitch}}{11}$
International Standards Organisation (ISO) metric thread	Max. Depth = 0.7035 × Pitch Min. Depth = 0.6855 × Pitch Angle = 60° in the plane of the axis Root radius Maximum = 0.0633 × Pitch Minimum = 0.054 × Pitch
American Standard ACME	Height of thread = 0.5 × Pitch + 0.254 mm Angle = 29° in the plane of the axis Width at tip = 0.3707 × Pitch Width at root = 0.3707 × Pitch – 0.132 mm

The shape of the cutting tool is of the same form as the thread to be generated.

For the purpose of feeding the tool for generating the thread, the feed is given by the lead screw. Feed is same as the lead of the pitch to be generated. In normal turning the thickness of the uncut chip is same as the feed rate chosen, whereas in the thread cutting case it is controlled by the depth of cut, *d*, in view of the thread form being generated as shown in Fig. 4.35(a). The uncut chip thickness, t_u, can be shown to be

$$t_u = 2 \times d \times \tan \alpha \tag{20}$$

The depth of cut in the case of thread cutting can be given in two ways: plunge cutting as shown in Fig. 4.36(b) or compound cutting as in Fig. 4.35(a).

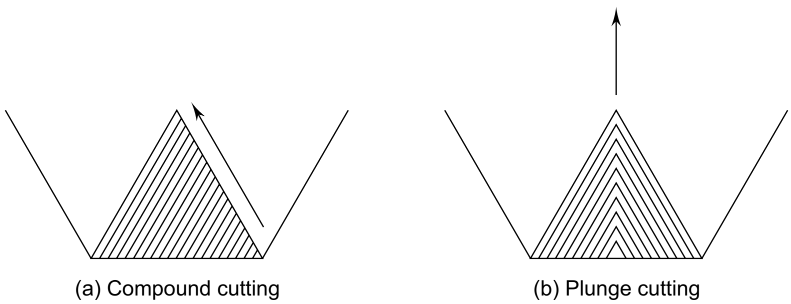


FIG. 4.35 *Depth of cut in screw cutting*

In the case of plunge cutting, the cutting of the thread takes place along both the flanks of the tool. This would mean that the cutting tool would have to be provided with a zero or negative rake angle. In addition the relief along the cutting edges cannot be provided in view of the form to be achieved. Cutting is also taking place along a longer length of the tool. This gives rise to difficulties in machining in terms of higher cutting forces and consequently chattering (violent vibrations). This results in poor surface finish and lower tool life, thus this method is not generally preferred.

With the compound feeding, the tool needs to be moved in both the directions (along the bed as well as a direction perpendicular to it) simultaneously to position the tool tip along one flank of the thread. This configuration helps in smoother flow of chips as the cutting takes place only along one cutting edge. This method therefore is much preferred compared to the earlier method. Only problem is to move the tool for giving the depth of cut along the flank of the thread, which can be achieved by the use of compound slide for giving the depth of cut as shown in Fig. 4.36, while feed is given by the carriage in the conventional manner.

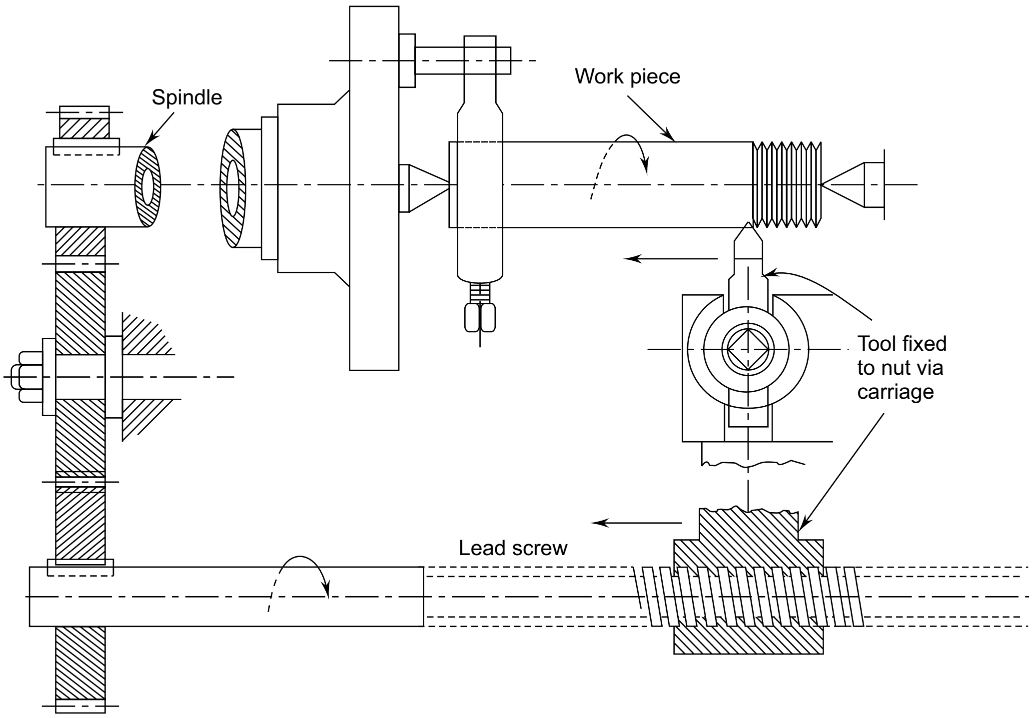


Fig. 4.36 *Thread cutting using compound slide*

The compound slide is rotated by the half angle of the thread, and the cutting tool is adjusted to make it perpendicular to the work piece surface. For this purpose a thread setting gauge which contains the required form of the thread being cut is kept perpendicular to the surface of the work piece, and the tool tip is set as shown in Fig. 4.37.

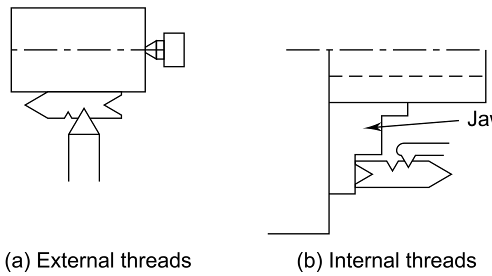


Fig. 4.37 *Setting-up the cutting tool for thread cutting in a lathe*

The next important consideration for thread cutting is the feeding of the tool along the helical path. For this purpose, the lead screw is normally employed for feeding the tool along the length of the job. In turning the engaging of tool at any point would be of no consequence since the surface to be generated is cylindrical. However in thread cutting it is essential that the tool tip should always follow the same thread profile generated in the first cut, otherwise no thread would be generated.

One of the methods that can be followed in this case is to reverse the spindle while retaining the engagement between the tool and the work piece. The spindle reversal would bring the cutting tool to the starting point of the thread following the same path in reverse. After giving a further depth of cut the spindle is again reversed and the thread cutting is continued in the normal way. This is easy to work and is somewhat more time consuming due to the idle times involved in the stopping and reversing of the spindle at the end of each stroke.

Another alternative method is to use a chasing dial to help in following the thread. As shown in Fig. 4.38 the chasing dial consists of a worm gear located inside the carriage in mesh with the lead screw. A vertical shaft connected with the worm gear has a dial with separate markings to indicate equal divisions of the circumference. Since the worm gear is in continuous contact with the lead screw, which is in continuous engagement with the spindle, markings on its surface indicate the precise position of the thread being cut on the work piece. Thus it is possible to engage with the work piece at any desired location.

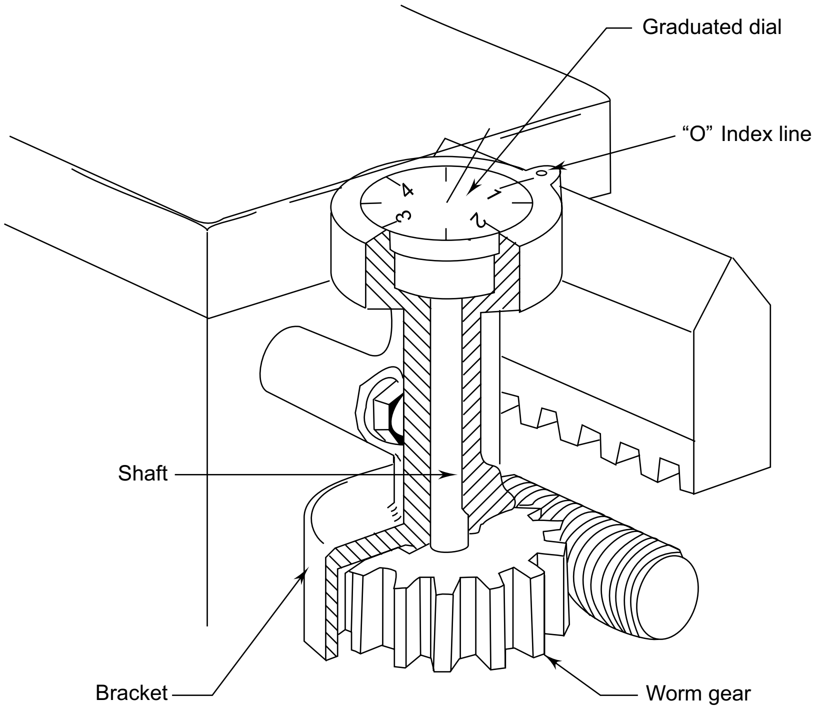


Fig. 4.38 *Chasing dial principle*

It is also possible to cut threads on a tapered surface by combing the thread cutting concepts as explained above along with the taper turning attachment as shown in Fig. 4.39.

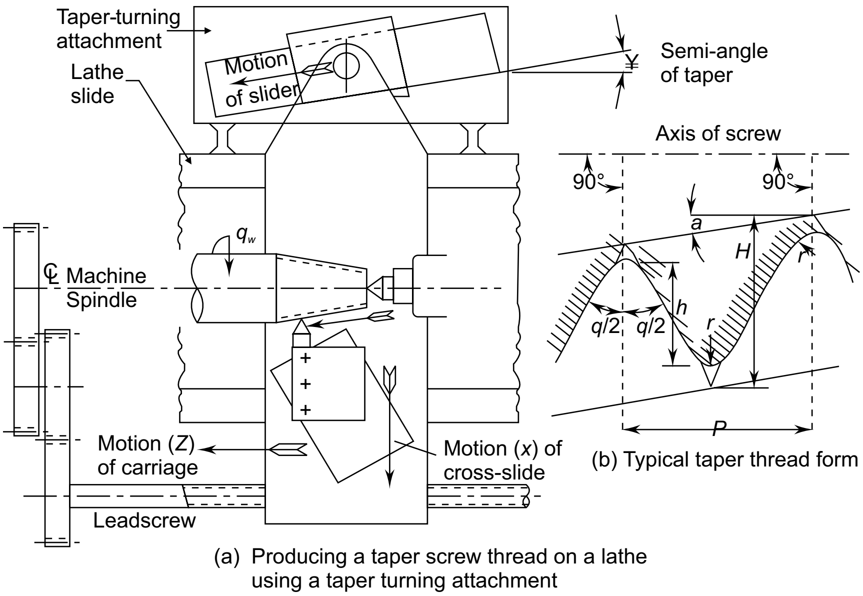


Fig. 4.39 *Setup for thread cutting on a taper*

4.8 SPECIAL ATTACHMENTS

Most of the details discussed so far allow for general purpose machining operations to be carried using a centre lathe. In addition, it is possible to provide special enhancements to the capability of a lathe whereby it could be used for special applications using special attachments. One such attachment discussed earlier is the taper turning attachment for obtaining cylindrical tapers or conical surfaces.

4.8.1 Milling Attachment

This is an attachment used in lathes to carry out milling operations. The attachment is provided with a separate spindle (Fig. 4.40) where the milling cutters could be located, and is attached to the cross slide, replacing the compound slide. The work is held between the lathe centres as in normal centre lathes. The milling cutter can normally be fed in all the three directions, thus permitting any type of milling operation.

4.8.2 Grinding Attachment

Similar to the milling attachment, a grinding attachment is used to finish a part in the lathe by completing the required grinding operations without disturbing the setup. It can be mounted in place of the tool post on the compound slide. These have two or three-axis movement and will be able perform a number of grinding operations.

4.8.3 Copy Turning Attachment

Many a times the need exists for machining complex contours, which require the feeding of the tool in two axes (*X* and *Y*) simultaneously, similar to taper turning. For such purposes copy turning is to be used. In this, the cross slide is directly driven by a stylus which can trace a master for the actual contour to be produced. The cross slide is made similar to the taper turning attachment.

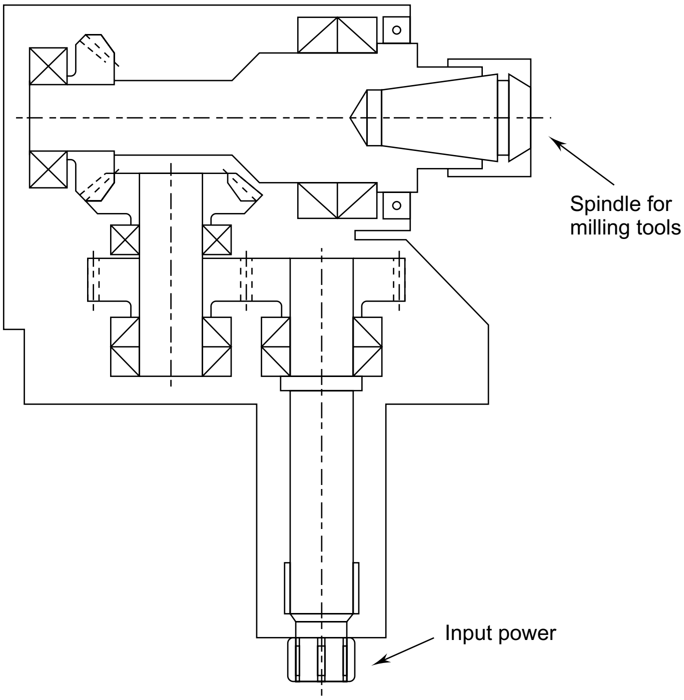


Fig. 4.40 *Schematic of a milling attachment for lathe*

4.8.4 Radius Turning Attachment

In this attachment, the cross slide is attached to the bed by means of a radius arm whose length is same as the radius of the spherical component to be produced as shown in Fig. 4.41. The radius arm couples any movement of the cross slide or the carriage and hence the tool tip traces a radius R as shown in Fig. 4.41.

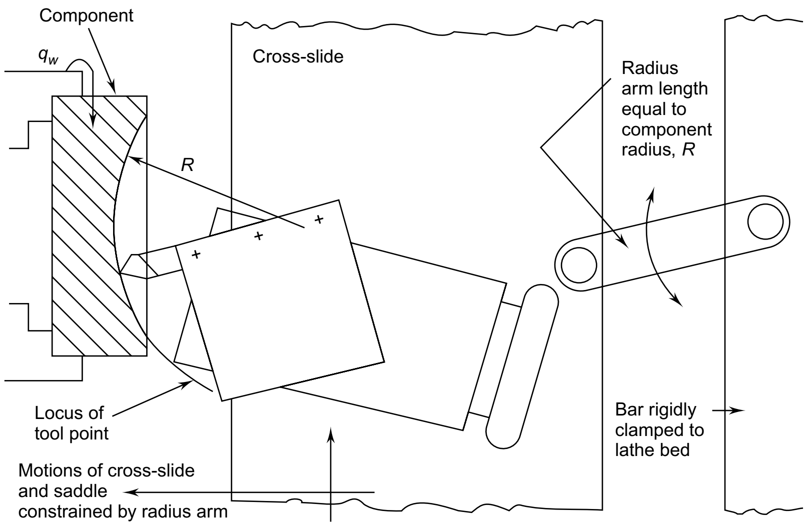


Fig. 4.41 *Radius turning attachment*

4.9 MACHINING TIME AND POWER ESTIMATION

To estimate the machining times, it is necessary to select the proper process parameters. For this purpose it is necessary to know the work piece material and the cutting tool material combinations to arrive at the right combination of the process parameters, cutting speed, feed and depth of cut. Their choice is somewhat difficult and a lot depends upon the shop practices as well as the experience of the operator/planner.

Some typical values of these parameters are given in Table 4.4 for the materials that are generally used. These should be considered as starting values and should be modified further based on the shop experience. Some of the problems that are likely to be noticed during machining process are identified and shown in Table 4.5 with the possible causes. If any of these problems are observed during the actual machining, then the operator should take care of it by solving the root problem as given in the table.

Table 4.4 Suggested cutting process parameters for turning

Work Material	Hardness	High Speed Steel Tool		Carbide Tool	
	BHN	Speed m/min	Feed mm/rev	Speed m/min	Feed mm/rev
Grey cast iron	150–180	30	0.25	140	0.30
Grey cast iron	220–260	20	0.25	90	0.30
Malleable Iron	160–220	33	0.25	50	0.25
Malleable Iron	240–270	—	—	45	0.30
Cast steel	140–180	40	0.25	150	0.30
Cast steel	190–240	26	0.25	125	0.30
C20 Steel	110–160	40	0.30	150	0.38
C40 Steel	120–185	30	0.30	145	0.38
C80 Steel	170–200	26	0.30	130	0.30
Alloy Steel	150–240	30	0.25	110	0.38
Alloy Steel	240–310	20	0.25	100	0.30
Alloy Steel	315–370	15	0.25	85	0.25
Alloy Steel	380–440	10	0.20	75	0.25
Alloy Steel	450–500	8	0.20	55	0.25
Tool Steel	150–200	18	0.25	70	0.25
Hot Work die steel	160–220	25	0.25	120	0.25
Hot Work die steel	340–375	15	0.25	75	0.25
Hot Work die steel	515–560	5	0.20	23	0.20
Stainless Steel	160–220	30	0.20	120	0.25
Stainless Steel	300–350	14	0.20	70	0.25
Stainless Steel	375–440	10	0.20	30	0.25
Aluminium Alloys	70–105	210	0.30	400	0.38
Copper Alloys	120–160	200	0.25	300	0.25
Copper Alloys	165–180	85	0.25	230	0.25

Table 4.5 Possible causes for the turning problems

Observed Problem	Possible Cause
Vibration	Work or chuck out of balance
Chatter	Work improperly supported Feed rate too high Tool overhang too large Tool is not properly ground or length of tool edge contact is high
Work piece not turned straight	Headstock and tailstock centres not aligned Work improperly supported Tool not in the centre
Work piece out of round	Work loose between centres Centres are excessively worn Centre out of round

Turning:

The cutting speed in turning is the surface speed of the work piece. Thus,

$$V = \frac{\pi DN}{1000} \tag{21}$$

where, V = cutting speed (surface), m/min

D = diameter of the work piece, mm

N = rotational speed of the work piece, rpm

The diameter, D to be used can be either the initial diameter of the blank or the final diameter of the work piece after giving the depth of cut. However, there is practically not much change in the values obtained by using either of the values. To be realistic, the average of the two diameters would be better.

From the above equation, we get

$$N = \frac{1000\,V}{\pi\,D} \tag{22}$$

The rpm obtained from the previous equation may not be an exact value of the speed available on the lathe machine, since any lathe would only have limited range of rpms available. It therefore is necessary to adjust the values obtained to that available in the speed range considering the work and tool material combination. This is demonstrated later using an example.

The time, t for a single pass is given by

$$t = \frac{L + L_o}{f\,N} \tag{23}$$

where L = length of the job, mm

L_o = over travel of the tool beyond the length of the job to help in the setting of the tool, mm

f = feed rate, mm/rev

The over travel to be provided depends upon the operator's choice but usual values could be 2 to 3 mm on either side.

The number of passes required to machine a component depends upon the left-over stock (stock allowance). Also depending upon the specified surface finish and the tolerance on a given dimension, the choice would have to be made as to the number of finishing passes (1 or 2) while the rest of the allowance is to be removed through the roughing passes. The roughing passes, P_r is given by

$$P_r = \frac{A - A_f}{d_r} \tag{24}$$

where A = Total machining allowance, mm

A_f = Finish machining allowance, mm

d_r = Depth of cut in roughing, mm

The value calculated from the above equation is to be rounded to the next integer.

Similarly the finishing passes, P_f is given by

$$P_f = \frac{A_f}{d_f} \tag{25}$$

where d_f = Depth of cut in finishing, mm

Example 4.4

Estimate the actual machining time required for the component (C40 steel) shown in Fig. 4.42. The available spindle speeds are, 70, 110, 176, 280, 440, 700, 1100, 1760 and 2800. Use a roughing speed of 30 m/min and finish speed of 60 m/min. The feed for roughing is 0.24 mm/rev while that for finishing is 0.10 mm/rev. The maximum depth of cut for roughing is 2 mm. Finish allowance may be taken as 0.75 mm. Blank to be used for machining is 50 mm in diameter.

Fig. 4.42 *Machining time example 1*

Solution Stock to be removed = $\frac{50 - 42}{2} = 4$ mm

Finish allowance = 0.75 mm

Roughing:

Roughing stock available = 4 – 0.75 = 3.25 mm

Since maximum depth of cut to be taken is 2 mm, there are 2 roughing passes.

Given cutting speed, V = 30 m/min

$$\text{Average diameter} = \frac{50 + 42}{2} = 46 \text{ mm}$$

$$\text{Spindle speed, } N = \frac{1000 \times 30}{\pi \times 46} = 207.59 \text{ RPM}$$

The nearest RPM available from the list is 176 RPM as 280 is very high compared to 207 as calculated.

$$\text{Machining time for one pass} = \frac{(120+2)}{0.24 \times 176} = 2.898 \text{ minutes}$$

Finishing:

Given cutting speed, $V = 60$ m/min

$$\text{Spindle speed, } N = \frac{1000 \times 30}{\pi \times 42} = 439.05 \text{ RPM}$$

The nearest RPM available from the list is 440 RPM.

$$\text{Machining time for one pass} = \frac{(120+2)}{0.10 \times 440} = 2.77 \text{ minutes}$$

Total machining time = $2 \times 2.888 + 2.77 = 8.546$ minutes

Facing:

In facing the choice of the spindle speed is affected by the fact that the cutting tool is engaged with the work piece at a gradually changing radius. As a result the actual cutting speed changes from the highest value at the surface to almost zero at the centre. Thus the diameter used for calculating the rpm in Eq. 20 should be the average of blank diameter and the lowest diameter (zero in case of complete facing) of the face being generated.

Taper turning:

The time calculation of taper turning depends upon the method used for the purpose. In the case of taper turning attachment, the calculation is similar to turning, as the feed motion is given by the carriage parallel to the axis of rotation. However, when the tailstock offset method is used, the motion of the tool is parallel to the actual taper surface generated and then that length should be used in Eq. 21.

All the other operations are similar to turning where care has to be taken to find the actual distance travelled in the operation.

Example 4.5

In Fig. 4.43 a component is shown to be machined from a stock of CRS C40 steel, 40 mm in diameter and 75 mm long. Calculate the machining times required for completing the part with (a) HSS tool and (b) Carbide tool.

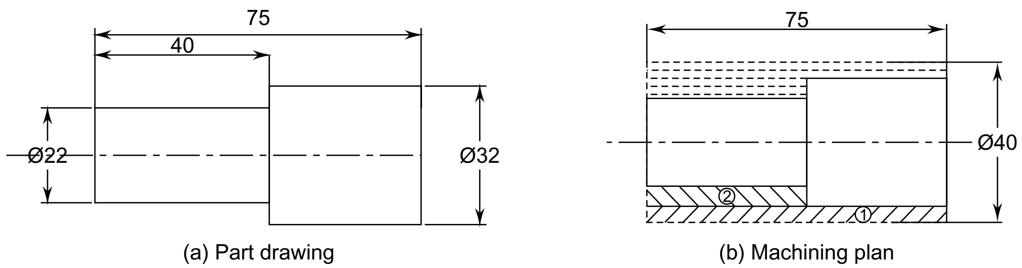


Fig. 4.43 *Machining time Example 2*

The machining is to be carried out in two stages as pockets marked in Fig. 4.43(b) as 1 and 2.

Pocket 1: HSS Tool

Assume Cutting speed, $V = 30$ m/min

Feed rate, $f = 0.30$ mm/rev.

Depth of cut = 2 mm

$$\text{Spindle speed} = \frac{1000 \times 30}{\pi \times 36} = 265.25 \text{ RPM} \approx 265 \text{ RPM}$$

$$\text{Time for machining one pass} = \frac{75 + 2}{0.30 \times 265} = 0.9686 \text{ minutes}$$

$$\text{Number of passes required} = \frac{40 - 32}{2 \times 2} = 2$$

Carbide Tool

Assume Cutting speed, $V = 145$ m/min

Feed rate, $f = 0.38$ mm/rev.

Depth of cut = 2 mm

$$\text{Spindle speed} = \frac{1000 \times 145}{\pi \times 36} = 1282.05 \text{ RPM} \approx 1280 \text{ RPM}$$

$$\text{Time for machining one pass} = \frac{75 + 2}{0.38 \times 1280} = 0.158 \text{ minutes}$$

Pocket 2: HSS Tool

$$\text{Number of passes required} = \frac{32 - 22}{2 \times 2} = 2.5 \approx 3$$

$$\text{Spindle speed} = \frac{1000 \times 30}{\pi \times 27} = 353.677 \text{ RPM} \approx 355 \text{ RPM}$$

$$\text{Time for machining one pass} = \frac{40 + 2}{0.30 \times 355} = 0.394 \text{ minutes}$$

Carbide Tool

$$\text{Spindle speed} = \frac{1000 \times 145}{\pi \times 27} = 1709.44 \text{ RPM} \approx 1710 \text{ RPM}$$

$$\text{Time for machining one pass} = \frac{40 + 2}{0.38 \times 1710} = 0.065 \text{ minutes}$$

Total machining time:

For HSS tool = $2 \times 0.9686 + 3 \times 0.394 = 3.1192$ minutes

For Carbide tool = $2 \times 0.158 + 3 \times 0.065 = 0.511$ minutes

The above gives only the estimation of actual machining time taken by the machine (tool in contact with the work piece). In addition to this there are certain other times, often termed as idle times, associated with the machining which needs to be estimated. Some representative values of these elemental times are given in Table 4.6.

Table 4.6 Idle time elements for lathe

Time Element		Time Seconds
Load and unload between centres with dog carrier with a work piece weight	1 kg 2 kg 4 kg 8 kg	20 30 40 50
Load and unload in 3-jaw chuck with a work piece weight	1 kg 2 kg kg 8 kg	30 40 50 60
Start/stop spindle		2
Reverse spindle		2
Change speed		3
Change feed		4
Engage/disengage feed		1
Index tool post		2
Approach tool, set to mark, return in turning		3
Approach tool, set to mark, return in boring		5
Move cross slide upto 100 mm		6
Move saddle upto 150 mm		3
Move saddle upto 500 mm		6
Fit/remove tool in tailstock		10
Approach/remove tailstock		15
Advance/return tailstock quill upto 50 mm		8

Power required in Turning: The power required at the spindle for turning depends upon the cutting speed, depth of cut, feed rate and the work piece material hardness and machinability. The power required depends upon the cutting force, which was shown in Chapter 2 to be a power function of feed rate, f and depth of cut, d. However, for the sake of gross estimation it can be safely assumed that

Cutting force, $F = K \times d \times f$

where K is a constant depending on the work material, which is given in Table 4.7.

Table 4.7 Constant *K* for power calculation

Material being Cut	K (N/mm^2)
Steel, 100–150 BHN	1200
Steel, 150–200 BHN	1600
Steel, 200–300 BHN	2400
Steel, 300–400 BHN	3000
Cast Iron	900
Brass	1250
Bronze	1750
Aluminium	700

Then Power, $P = F \times V$

Combining the above two equations
Power, $P = K \times d \times f \times V$

Example 4.6

Calculate the power required for roughing and finishing passes in Ex. 4.4.

Solution

Roughing:
Given feed rate, $f = 0.24$ mm/rev
Depth of cut, $d = 2$ mm

$$\text{Cutting speed, } V = \frac{\pi \times 176 \times 46}{1000} = 25.43 \text{ m/min}$$

The value of K from Table 4.7 = 1600 N/mm^2

$$\text{Power} = \frac{1600 \times 25.43 \times 0.24 \times 2}{60} = 325.5 \text{ W} = 0.326 \text{ kW}$$

Finishing:
Given feed rate, $f = 0.10$ mm/rev
Depth of cut, $d = 0.75$ mm

$$\text{Cutting speed, } V = \frac{\pi \times 440 \times 43.5}{1000} = 60.13 \text{ m/min}$$

$$\text{Power} = \frac{1600 \times 60.13 \times 0.10 \times 0.75}{60} = 120.26 \text{ W} = 0.120 \text{ kW}$$

Example 4.7

Calculate the power required for roughing and finishing passes in Ex. 4.5.

Solution

Pocket 1: HSS Tool
Assume Cutting speed, $V = 30$ m/min
Feed rate, $f = 0.30$ mm/rev.
Depth of cut = 2 mm
The value of K from Table 4.7 = 1600 N/mm^2

$$\text{Power} = \frac{1600 \times 30 \times 0.3 \times 2}{60} = 480 \text{ W} = 0.48 \text{ kW}$$

Carbide Tool
Assume Cutting speed, $V = 145$ m/min
Feed rate, $f = 0.38$ mm/rev.
Depth of cut = 2 mm

$$\text{Power} = \frac{1600 \times 145 \times 0.38 \times 2}{60} = 2939 \text{ W} = 2.94 \text{ kW}$$

For the second pocket also, the power required remains the same since the processing parameters did not change.

4.10 TYPICAL SETUPS

It is necessary to consider all the factors while planning for a job on any machine tool. A few examples of tool layouts are discussed that can be machined in a centre lathe.

In the examples the components are shown as a general cross-section of the type that would be machined on centre lathes. The examples are shown in the form of process diagrams where the material to be removed is shown in the form of a closed pocket which is hatched. The type of tool to be chosen is based on the type of pocket to be machined.

Example 1 The example is a cylindrical pin which can be turned from a long bar, with each part being parted off. The typical sequence in which the material is produced is shown in Fig. 4.44. In the first operation, the external diameter of 16 mm is produced to the required length using a right hand turning tool. The chamfering form tool to produce the right hand chamfer in operation 2 will follow this. In the third operation the undercut is produced again by using a form tool with plunge feed. Then the part will be parted off from the bar stock with a parting tool. In the next operation, the part will be held from the other side and then the facing and chamfering will be done to complete the job.

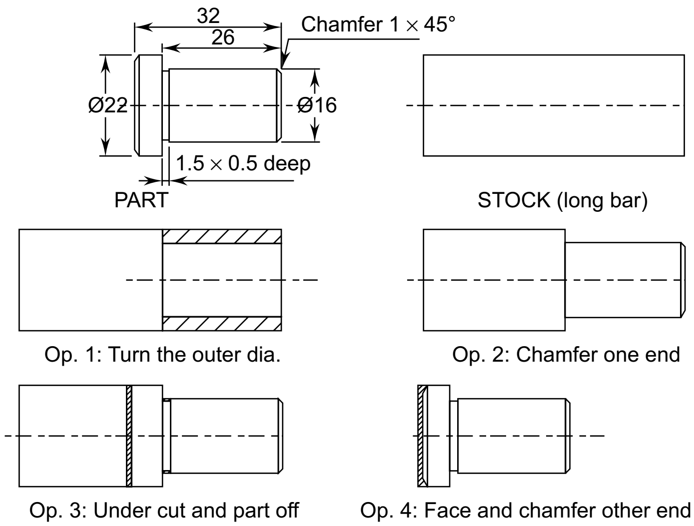


Fig. 4.44 *Typical process pictures for machining a pin from a cylindrical bar on a lathe involving only external features*

Example 2 The second component shown in Fig. 4.45 is a bit more complex for processing in view of the close tolerances specified on some dimensions.

As shown the close tolerances cannot be achieved by turning and hence a grinding allowance is left on the 32 mm diameter portion and the rest is completed in the centre lathe as shown in Fig. 4.46.

In this case, since the large diameter of the component is in the middle, the part needs to be machined at least in two setups, with the part being clamped from both the sides alternatively. First the part is held from the right side and the outer diameter is turned leaving the allowance for finishing using a right hand turning tool. In the second operation the part is gripped from the turned side and the two steps are created on the other side using a right hand turning tool, leaving the allowance for the next operation. In the next operation the two shoulders are faced to get the squareness desired in the part. Finally the undercut is made with a form tool and the part leaves the centre lathe for finishing.

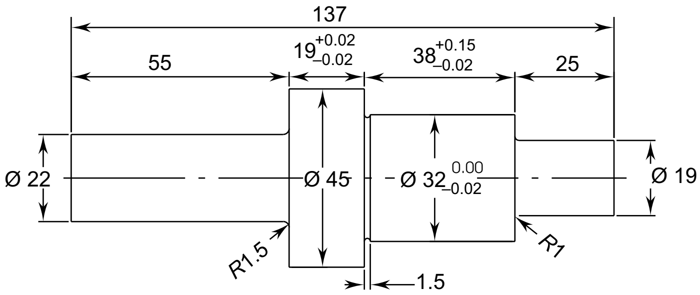


Fig. 4.45 *Typical component (pin) normally produced on a lathe*

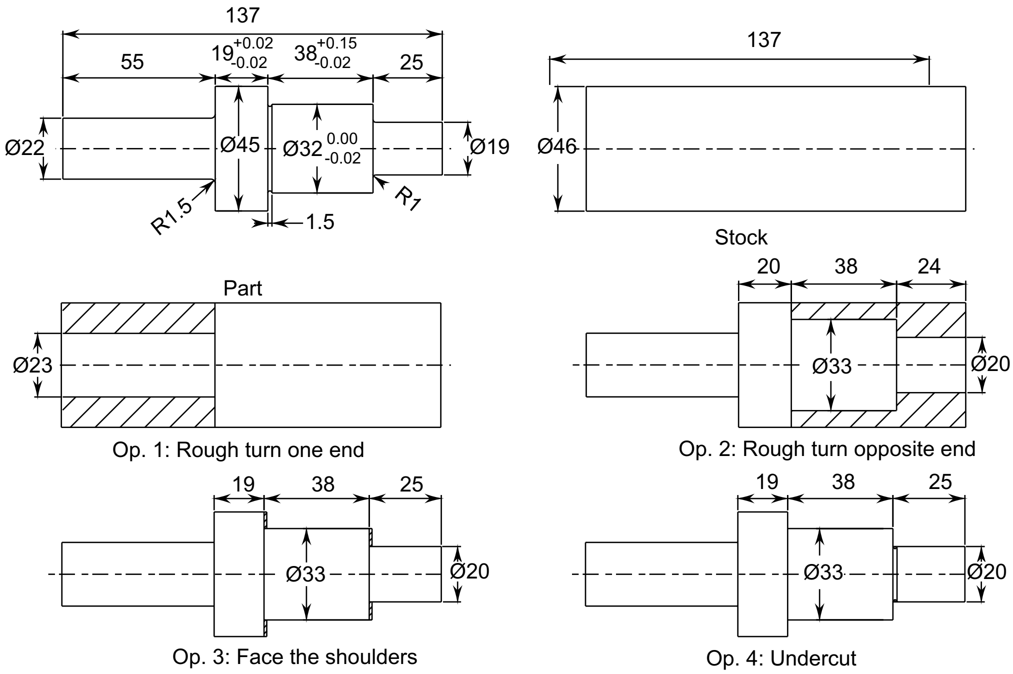


Fig. 4.46 *Typical process pictures for machining a pin on a lathe*

Example 3 Next example is a component with both external and internal details as shown in Fig. 4.47. In this though the part diameter is toleranced, the accuracy is achievable in a centre lathe. In the first operation the part is drilled to make the central hole with a twist drill held in the tailstock. After this the part is turned to get the outside diameter using a right hand turning tool. In the next operation, boring of the hole is done to get the necessary finish and accuracy. After the boring operation the facing of the shoulders is done to complete the processing of operation from one end. Then the part is reversed in the chuck and turned and faced on the other side to complete the machining of the part.

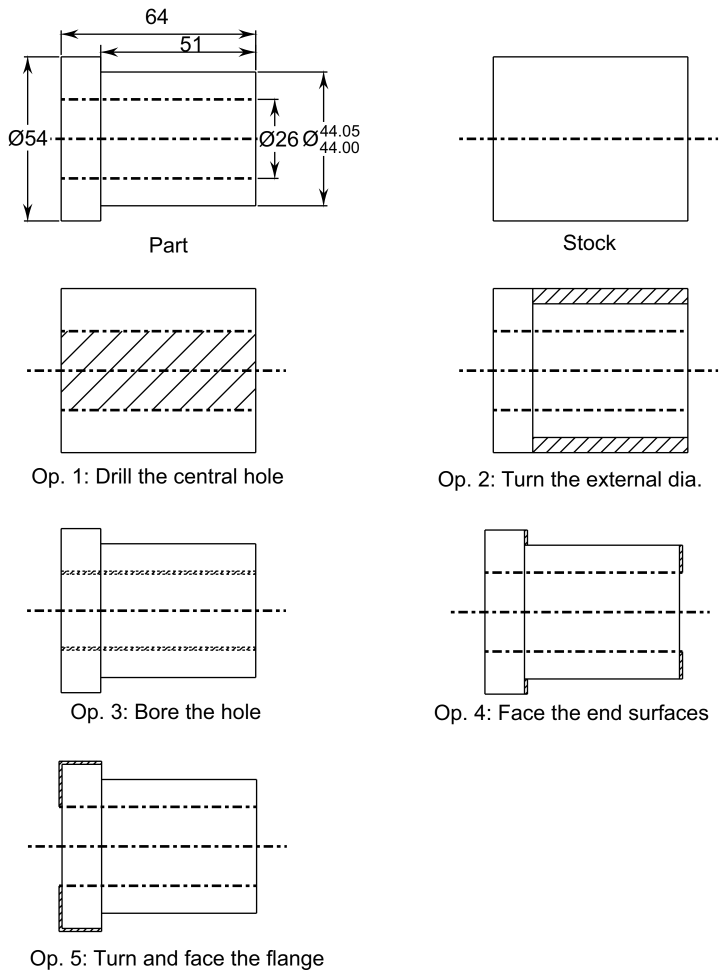


Fig. 4.47 *Typical process pictures for machining a bush on a lathe involving external and internal features*

SUMMARY

Lathe is the most important and common machine tool found in practically all machine shops. It is one of the oldest known machine tool, though major developments have appeared in the last two centuries.

- A large variety of lathes have been developed to cater to the different processing requirements.
- A lathe consists of a bed, headstock, tailstock and a carriage as major components along with a few other items that provide the necessary support and motions.
- A variety of chucks such as universal 3-jaw, independent 4-jaw, and faceplate, are used to locate and support work pieces in a lathe for common machining applications.
- Though single point cutting tools are used for most of the operations in a lathe, a variety of them are available, depending upon the type of surface that needs to be generated.

- There are a large variety of operations such as turning, facing, knurling, contouring, etc. that can be carried out in a lathe. In fact practically all types of surfaces can be generated in a lathe.
- Taper turning is a special type of operation that requires the tool to be moved in two different directions simultaneously to generate the surface. For this purpose, a variety of methods are used in a lathe such as compound slide, tailstock offset or a special attachment.
- Precision threads can be cut in a lathe using the lead screw and special methods.
- There are various special attachments such as a milling attachment, grinding attachments, etc. that enhance the range of surfaces that can be generated in a lathe.
- Machining time for different operations can be estimated using the cutting process parameters and the geometry of the part.

Questions

4.1 What type of work holding devices are generally used in a lathe? Give the typical applications, comparative accuracies, precautions and disadvantages for each type of work holding device.

4.2 How is a lathe specified? Explain with a neat sketch the relevance of each of the specification points.

4.3 What type of work holding devices are generally used in a lathe? Give the typical applications, comparative accuracies and disadvantages.

4.4 What are the applications, achievable (comparative) accuracies and disadvantages of the following type of work holding device:
(a) 3-jaw chuck (b) 4-jaw chuck

4.5 Compare the applications, accuracies and disadvantages of 3-jaw and 4-jaw chucks.

4.6 What are the types of surfaces that can be generated in a centre lathe? Show with the help of sketches how these are achieved?

4.7 Explain the methods used for the generation of threads in lathe.

4.8 How is thread chased in a lathe?

4.9 What are the precautions and settings to be done for cutting single start threads on a centre lathe? Give a practical method that will allow the cutting tool to follow the thread.

4.10 What are the various methods available for taper turning in a lathe? Explain their specific advantages and limitations.

4.11 Describe the method of generation of a long and small taper in a centre lathe.

4.12 Explain the procedure for turning a job, which is very long. List the various operations and tools (cutting and holding) required for the operation.

4.13 What are the various methods available for taper turning on a centre lathe? Explain in detail with a sketch the method used for machining steep tapers of short length.

4.14 What are the various methods available for supporting long components and frail components in a lathe? Explain with sketches.

4.15 Draw a setup required for cutting external threads on a centre lathe. Explain clearly the various methods that are followed for the purpose of engaging the cutting tool for giving the depth of cut in threading.

4.16 What are the types of lathes you are familiar with and give their applications?

4.17 What are the applications, achievable (comparative) accuracies and disadvantages of the following type of work holding device when used in a centre lathe? State the any precautions to be used with their application.
(a) Face plate (b) 4-jaw chuck

4.18 Explain the steady rest and follower rest as used in engine lathes. Give their functions.

Problems

4.1 A grey cast iron shaft is machined in a centre lathe in 1 minute with a single cut. The shaft is 100 mm long and 75 mm in diameter. If the feed used is 0.30 mm/revolution, what cutting speed was used? [7.854 m/min]

4.2 In a centre lathe equipped with a taper turning attachment, a 3° taper is to be produced. The small end of the work piece is 25 mm in diameter. The taper attachment is set at 3° but the tool is set 3 mm below centre. Calculate the error in taper due to the incorrect setting of the tool. [0.018°]

4.3 Calculate the maximum possible error (distance from the centre of the work piece) in setting the turning tool with a clearance angle such that the tool starts rubbing at the clearance face (clearance angle become zero). The work piece is 50 mm in diameter. If the rake angle is 10°, what is the effective rake angle for this condition? [5°]

4.4 A high speed steel tool has a tool life of 105 minutes while turning cast iron at a cutting speed of 20 m/min. If the tool life is given by the Taylor's tool life equation as $VT^{0.1} = C$, calculate the tool life for a cutting speed of 15 m/min. [1862.9 minutes]

4.5 While taper turning a taper of 1 in 6, the tool is wrongly set at a distance 4 mm below the work piece centre. If the small end of the work piece is 35 mm in diameter, calculate the actual taper obtained. [9.42°]

4.6 The taper turning attachment of a lathe is set to turn a taper of 1 in 6. The larger end of the work piece is 50 mm in diameter and the length is 100 mm. If the tool is set 5 mm below the centre of the work piece, calculate the actual taper produced. [9.425°]

4.7 A work piece of 30 mm diameter is being turned on a centre lathe. The tool angles are a rake of 12° and clearance of 7°. If the tool is set at a distance of 0.8 mm above the centre line, calculate the resultant tool angles. [3.94°]

4.8 A work piece with 25 mm diameter is turned in a centre lathe with a back rake angle of 12° and front relief angle of 7°. Calculate the actual cutting angles if the cutting tool is set (a) 0.8 mm above the centre line of the work piece, and (b) 0.8 mm below the centre line of the work piece. [3.33°, 10.67°]

4.9 A work piece with 50 mm diameter is turned in a centre lathe with a back rake angle of –12° and front relief angle of 7°. Calculate the actual cutting angles if the cutting tool is set (a) 1.2 mm above the centre line of the work piece, and (b) 1.2 mm below the centre line of the work piece. [4.25°, 9.75°]

4.10 A work piece with 100 mm diameter is turned in a centre lathe with a back rake angle of 15° and front relief angle of 8°. Calculate the actual cutting angles if the cutting tool is set (a) 1.5 mm above the centre line of the work piece, and (b) 1.5 mm below the centre line of the work piece.

[16.72°, 6.28°, 13.28°, 9.72°]

4.11 A work piece with 125 mm diameter is turned in a centre lathe with a back rake angle of –10° and front relief angle of 8°. Calculate the actual cutting angles if the cutting tool is set (a) 1.2 mm above the centre line of the work piece, and (b) 1.2 mm below the centre line of the work piece.

[–8.9°, 6.9°, –11.1°, 9.1°]

4.12 Calculate the tailstock offset required for a taper of 8° on a job that is 120 mm long. With the same setting, what would be the taper produced if the length of the shaft change by ± 3 mm.

[7.8°, 8.21°]

4.13 Calculate the power required for turning a mild steel bar of 50 mm diameter with a spindle speed of 140 RPM. Assume a depth of cut of 3 mm and a feed rate of 1 mm/rev. [1.76 kW]

4.14 The following component (Fig. 4.48) is to be manufactured at the rate of 500 components per month. Specify the machine tools and cutting tools to be used for the component. The machining faces are identified with the help of the letter '*f*'. Justify your choice. If an alternative, less economical choice is possible just mention it with explanation.

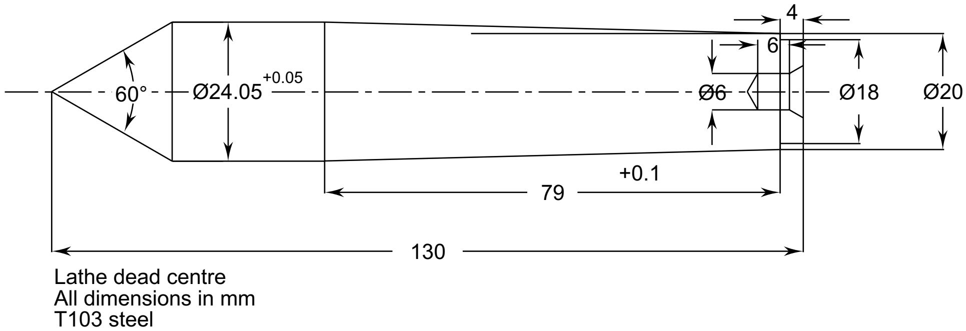


Fig. 4.48

4.15 Component shown in Fig. 4.49 is to be manufactured from a C45 steel bar stock of 75 mm diameter. A batch of 10 such components need to be manufactured using the general purpose machines available in the shop. The tolerances on all dimensions are ± 0.100 mm. Mention the machine tool used along with any accessories and tools needed. Calculate the actual machining time (ignoring the idle times) for a single component. Make any valid assumptions. [7.44 minutes]

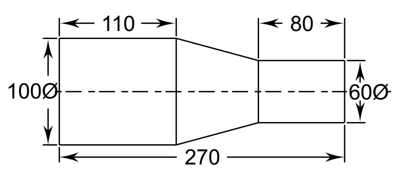


Fig. 4.49

Multiple Choice Questions

4.1 Lathe specification of swing over bed specifies
- (a) Maximum diameter of job that can be turned in the lathe
- (b) Minimum diameter of job that can be turned in the lathe
- (c) Maximum radius of job that can be turned in the lathe
- (d) Minimum radius of job that can be turned in the lathe

4.2 Lathe specification of distance between centres specifies
- (a) Minimum length of the job that can be turned
- (b) Maximum length of the job that can be turned
- (c) Minimum diameter of the job that can be turned
- (d) Maximum diameter of the job that can be turned

4.3 Spindle of a lathe is housed in
- (a) Tailstock
- (b) Headstock
- (c) Carriage
- (d) Apron

4.4 3-jaw chuck is used in a lathe to clamp
- (a) Cylindrical work piece to locate the axis of rotation
- (b) Eccentric work piece to locate the axis of rotation
- (c) Square bar to locate the axis of rotation
- (d) Any type of work piece to locate the axis of rotation

4.5 4-jaw chuck is used in a lathe to clamp
- (a) Only for cylindrical work piece to locate the axis of rotation
- (b) Only for eccentric work piece to locate the axis of rotation
- (c) Only for square bar to locate the axis of rotation
- (d) Any type of work piece to locate the axis of rotation

4.6 A face plate is used in a lathe to clamp
- (a) Only for cylindrical work piece to locate the axis of rotation
- (b) Only for eccentric work piece to locate the axis of rotation
- (c) Only for square bar to locate the axis of rotation
- (d) Any odd shaped work piece to locate the axis of rotation

4.7 The most common included angle for the centre hole is
- (a) 45º (b) 60º
- (c) 90º (d) 75º

4.8 A collet chuck is used for holding
- (a) Cylindrical work piece to locate the axis of rotation quickly
- (b) Eccentric work piece to locate the axis of rotation quickly
- (c) Square bar to locate the axis of rotation quickly
- (d) Any type of work piece to locate the axis of rotation quickly

4.9 For proper cutting, the cutting tool in a lathe should be set
- (a) Slightly below the axis of the work piece so that no rubbing takes place
- (b) Exactly at the centre of the axis of the work piece
- (c) Slightly above the axis of the work piece so that no rubbing takes place
- (d) Anywhere, since the work piece is actually rotating

4.10 Facing operation in a lathe is used for producing
- (a) A cylindrical surface
- (b) A plane surface
- (c) A tapered surface
- (d) A hole

4.11 Knurling operation in a lathe is used for producing
   (a) A plane surface
   (b) A cylindrical surface
   (c) A serrated surface
   (d) A tapered surface

4.12 Boring operation in a lathe is used for
   (a) Generating a plane surface
   (b) Enlarging a hole
   (c) Generating a serrated surface
   (d) Generating threads

4.13 Taper turning using the compound slide is used for
   (a) Only for small tapers over long length
   (b) Only for large and steep tapers
   (c) Only for small and steep tapers
   (d) For all types of tapers

4.14 Taper turning using the tailstock offset method is used for
   (a) Only for small tapers over long length
   (b) Only for large and steep tapers
   (c) Only for small and steep tapers
   (d) For all types of tapers

4.15 Taper turning using the taper turning attachment is used for
   (a) Only for small tapers over long length
   (b) Only for large and steep tapers
   (c) Only for small and steep tapers
   (d) For all types of tapers

4.16 Thread chasing is required in cutting screw threads because
   (a) It improves surface finish
   (b) It is necessary to follow the thread
   (c) It increases the threading tool life
   (d) It reduces the cutting forces

4.17 Giving the depth of cut using the plunge cutting method in thread cutting in a lathe is
   (a) Preferred because it improves the surface finish
   (b) Preferred because it increases the threading tool life
   (c) Not preferred because the surface finish is poor
   (d) Preferred because it produces accurate thread

4.18 Giving the depth of cut using the compound cutting method in thread cutting in a lathe is
   (a) Preferred because it improves the surface finish
   (b) Not preferred because it decreases the threading tool life
   (c) Not preferred because the surface finish is poor
   (d) Preferred because it produces accurate thread

4.19 Centre lathe is used
   (a) For high production rates
   (b) In normal workshops and repair shops
   (c) For large production volumes
   (d) For automated production

4.20 The lead screw of a centre lathe is 6 mm. If a thread of 1.5 mm pitch is to be cut on a work piece, which is rotating at 120 rpm, what should be the rpm of the lead screw?
   (a) 20 rpm (b) 30 rpm
   (c) 120 rpm (d) 180 rpm

Answers to MCQs

4.1 (a)	4.2 (b)	4.3 (b)	4.4 (a)	4.5 (d)
4.6 (d)	4.7 (b)	4.8 (a)	4.9 (b)	4.10 (b)
4.11 (c)	4.12 (b)	4.13 (c)	4.14 (a)	4.15 (d)
4.16 (b)	4.17 (c)	4.18 (a)	4.19 (b)	4.20 (b)

CASE STUDY

PROCESS PLANNING

Analyze and interpret the following components from process planning point of view for mass production. All dimensions are in mm. Stock material is 40 mm diameter × 92 mm length AISI 4340 CRS steel. Using the planning tables given, provide the detailed analysis of the part drawing, giving the following information: The Brinell hardness of 4340 steel is about 250.

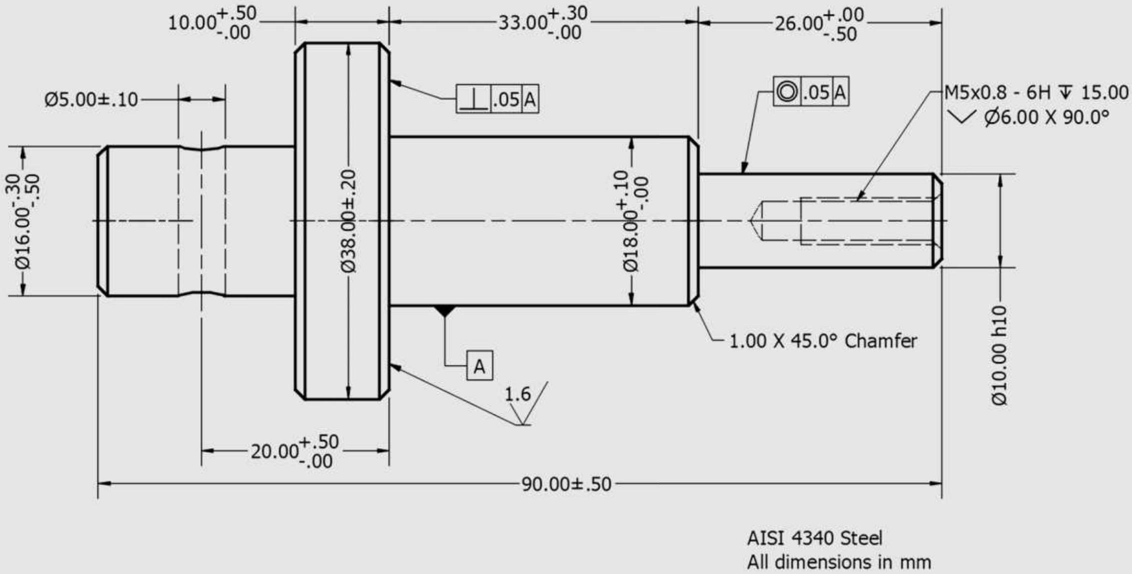


Fig. 1 *A part that needs to be mass produced using automated machining*

Manufacturing Considerations

- The overall length of the part is 90.00 mm diameter and this dimension should be within ±0.50 mm.
- Datum A is located on the surface of the Ø18.00 mm with a tolerance of $18.00^{+0.10}_{-0.00}$ mm with a length of $33.00^{+0.30}_{-0.00}$ mm.
- The left end is of the shaft is diameter $16.00^{+0.30}_{-0.50}$ mm.
- The left end is of the shaft has a concentricity tolerance of 0.05 mm with datum A.
- The right (small) end of the part is having a diameter of 10.00 mm with h10 tolerance and a length of $26.00^{+0.00}_{-0.50}$ mm.
- The largest diameter of the shaft is 38.00 ±0.20 mm with a width of $10.00^{+0.50}_{-0.00}$ mm.
- The right end of the large diameter has a perpendicularity tolerance of 0.05 mm from datum A.
- Ignore the chamfers, radial hole and the tapping to be done on the right end.

Machining operations required to achieve the specified dimensions and tolerances

The dimension 10h10 is $10.00^{+0.000}_{-0.058}$ mm.

Given stock size of 40 mm diameter, chucking on one end, Facing, turning 10, and 18 mm, and chucking on the other end turn 38 and 16 mm.

Process Planning Sheet

Part Name:	**Part no: Q1**	**Drg. No.:**	
Quantity:	**Material: 4340 Steel**	**Planner:**	
Date: Dec. 10, 2017	Revision no.:	Page 1 of 1	Order no.:

Op. no.:	Description	Dimension and Tolerance, mm	Allowance, mm	Machine Tool
10	Facing	91.0	1.0	CNC Turning center
20	Rough turning	Ø 19 ±0.10	2 × 5	CNC Turning center
30	Finish turning	$Ø18^{+0.10}_{-0.00}$	0.5	CNC Turning center
40	Facing			CNC Turning center
50	Turning	Ø11.0±0.10 mm	3.5	CNC Turning center
60	Grinding	$10^{+0.0}_{-0.058}$	0.5	Cylindrical grinder
70	Facing	90.0	1.0	CNC Turning center
80	Turning	Ø 38	1.0	CNC Turning center
90	Turning	Ø 16	11	CNC Turning center
100	Facing			CNC Turning center

For the process plan that is given above

- Identify the cutting tools to be used for each of the operation.
- Identify the cutting process parameters to be used based on the manufacturing requirements.
- Calculate the machining time for each of the operation.

CHAPTER 5

Special Purpose Lathes

Objectives

Special purpose lathes are developed to facilitate the mass manufacturing industry. After completing this chapter, the reader will be able to

- Understand the limitations of centre lathe for mass production
- Utilise capstan and turret lathes for different parts
- Understand different types of automatic lathes and their application methods
- Develop tool layouts for an automatic lathe for a given part

5.1 LIMITATIONS OF A CENTRE LATHE

As discussed in the previous chapter, the centre lathe though a general purpose machine tool, has a number of limitations that preclude it to become a production machine tool. The main limitations of centre lathes are as follows:

- The setting time for the job in terms of holding the job is large.
- Only one tool can be used in the normal course. Sometimes the conventional tool post can be replaced by a square tool post with four tools.
- The idle times involved in the setting and movement of tools between the cuts is large.
- Precise movement of the tools to destined places is difficult to achieve, unless proper care is exercised by the operator.

All these difficulties mean that the centre lathe cannot be used for production work in view of the low production rate. Thus the centre lathe is modified to improve the production rate. The various modified lathes are:

- Turret and capstan lathes
- semi-automatics
- automatics

The improvements are achieved basically in the following areas:

- work holding methods
- multiple tool availability
- automatic feeding of the tools
- automatic stopping of tools at precise locations
- automatic control of the proper sequence of operations

5.2 CAPSTAN AND TURRET LATHES

The main characteristic feature of the capstan and turret lathes is the six sided (hexagonal) block mounted on one end of the bed replacing the normal tailstock as shown in Fig. 5.1. This allows for mounting six tool blocks each of which can contain one or more tools depending upon the requirement. Further on the cross slide, two tool posts are mounted, one in the front and the other in the rear. Each one of them can hold up to four tools each. Thus the total carrying capacity is a maximum of 14 tools when only one tool is mounted in each of the locations.

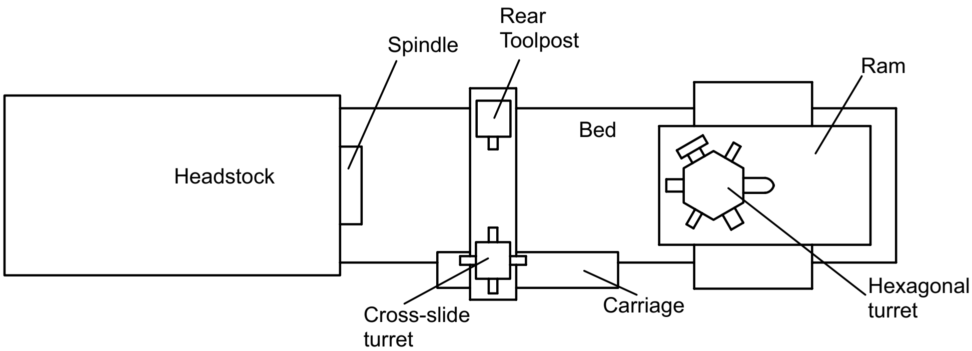


Fig. 5.1 *A view of the turret lathe*

As shown in Fig. 5.1 the turret lathe consists of an all gear, heavy duty headstock with a greater range of spindle speeds. The turret is mounted on a saddle, which in turn is sliding on the bed. When the saddle moves on the bed during the return stroke it would automatically be indexed to the next tool position, thus reducing the idle time of the machine.

The tools in the turret lathe are provided with a system of stops and trips on the feed rod which can precisely control the actual distance moved by the tool. Thus it is possible to set and control the individual movements of the tools as required by the component.

The type of work holding devices that can be used with turret lathes is similar to the conventional lathes, but in view of the higher productivity demanded and greater repeatability required, generally automatic fixtures such as collets, self centring chucks or pneumatic chucks are used.

The collet chucks come in a variety of designs as shown in Fig. 5.2. The actual clamping is done by the movement of the collet tube along the axis of the spindle by either pushing Fig. 5.2(a) or pulling Fig. 5.2(b). Sometimes it is possible that the bar material will be either pushed or pulled back during the closing of the collet. This can be prevented by having an external tubular locking stop so that the axial movement is prevented as shown in Fig. 5.2(c).

Often a large variety of components on a turret lathe are machined from raw material which is in a bar form. For the purpose of continuous feeding of the bar special bar feeding arrangements are available which pushes the bar by a precise amount against a stop provided on the face of the hexagonal turret at the beginning of the cycle. The last operation in such cases is the parting off operation from the cross slide tool which separates the machined component from the bar stock.

Most of the tools used in the cross slide tool post are very similar to those used in the centre lathe. Form tools are generally used in the cross slide. A large variety of special tool holders are available for use in the

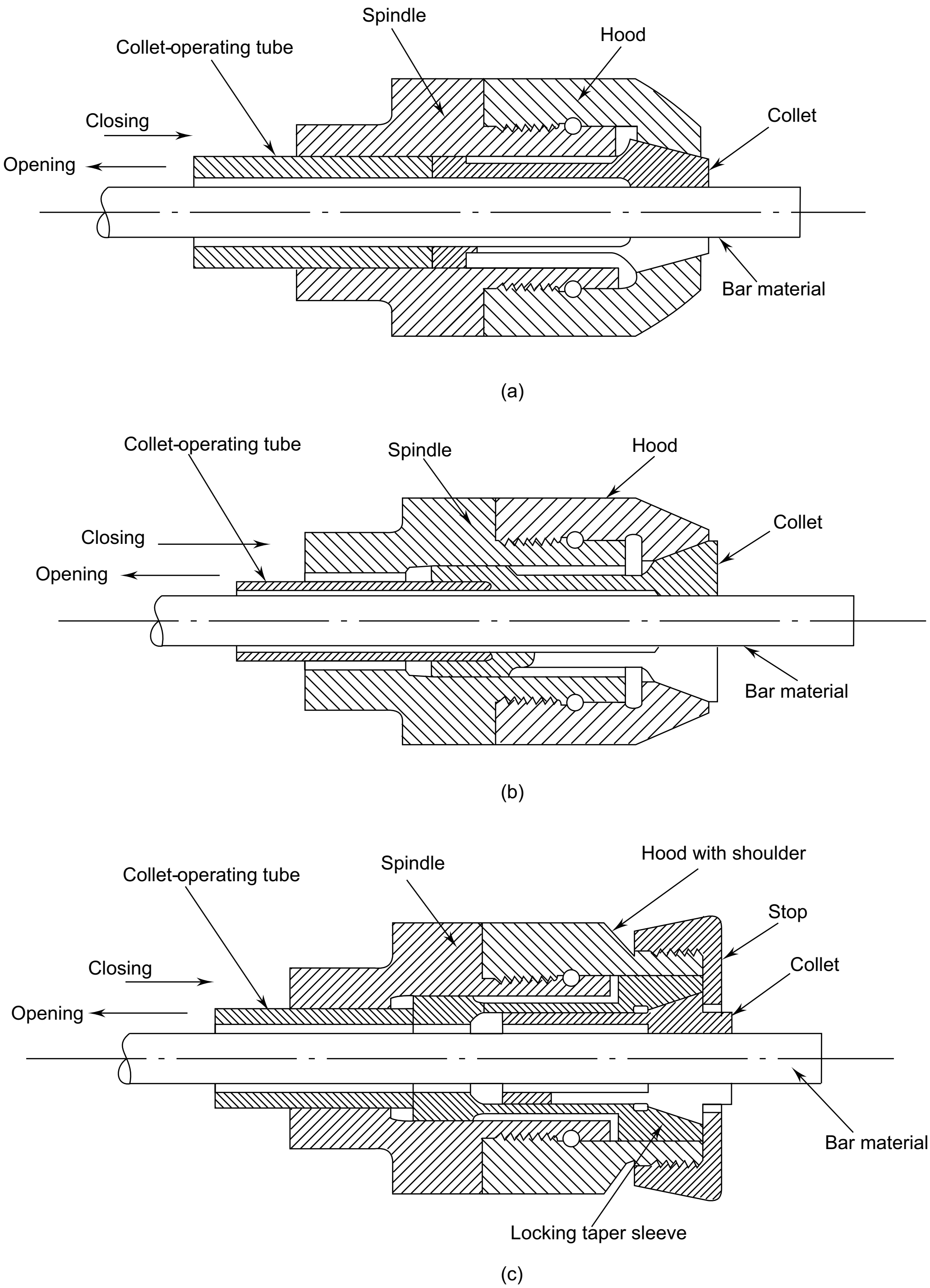


Fig. 5.2 *Different forms of collet chucks used in turret lathes*

turret for providing greater productivity. A few of such specialised tooling are shown in Fig. 5.3 and 5.4. A box tool is generally used for long turning jobs since the tool while cutting also supports the job. They have a cutting tool and also support rollers for providing the necessary support to the work piece. This helps in machining of bars which are not well supported and is generally used for bar work. There are a number of ways in which the bar can be supported during the machining operation as shown in Fig. 5.3. It is also possible to have more than one cutting tool held in a box tool such that there is an overlap of the cuts while also providing support for the work piece.

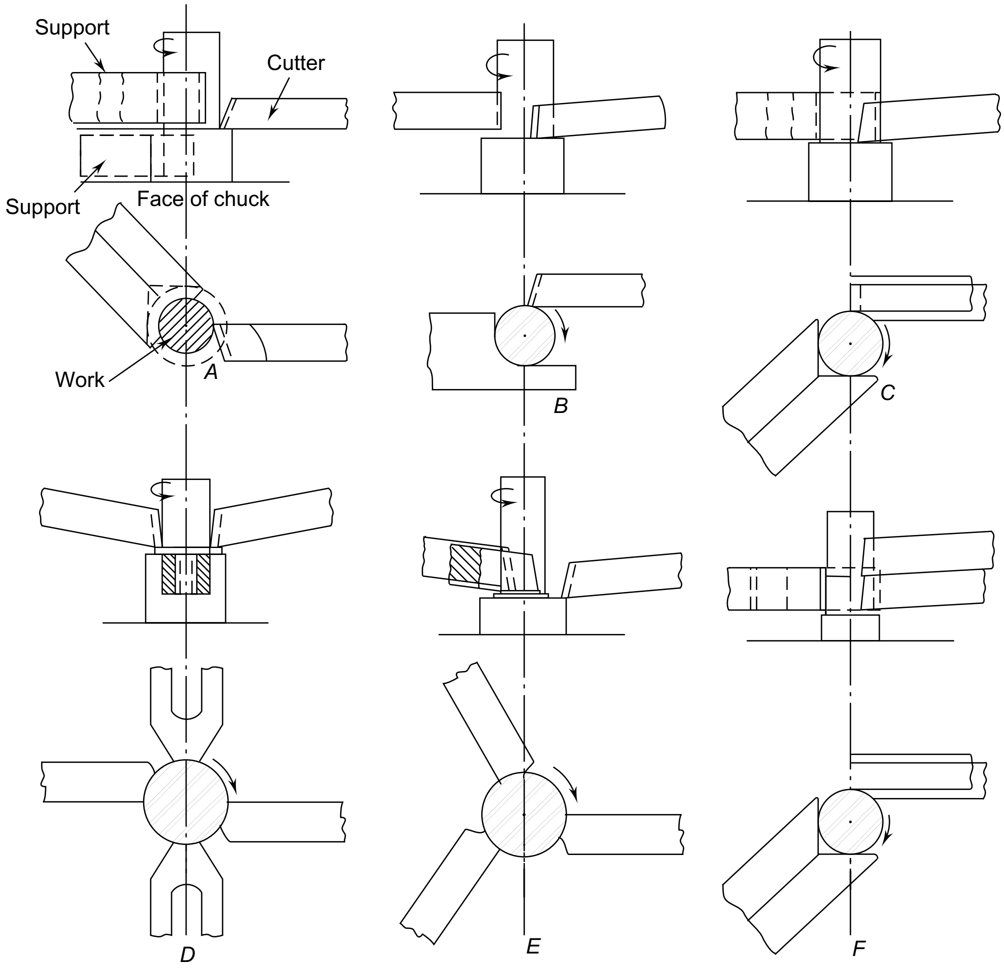


Fig. 5.3 *Different types of box tools used in turret lathes*

Combination tool holders allow for mounting multiple cutting tools with the provision for their adjustment to suit the machining situation. These have the ability to perform more than one cutting operation at the same time, thereby reducing the actual machining time required for the operation. Some typical tools are shown in Fig. 5.4. They can have both the internal and external cutting tools in a single tool holder such that the work piece support can be taken care of, thereby allowing higher accuracy to be achieved.

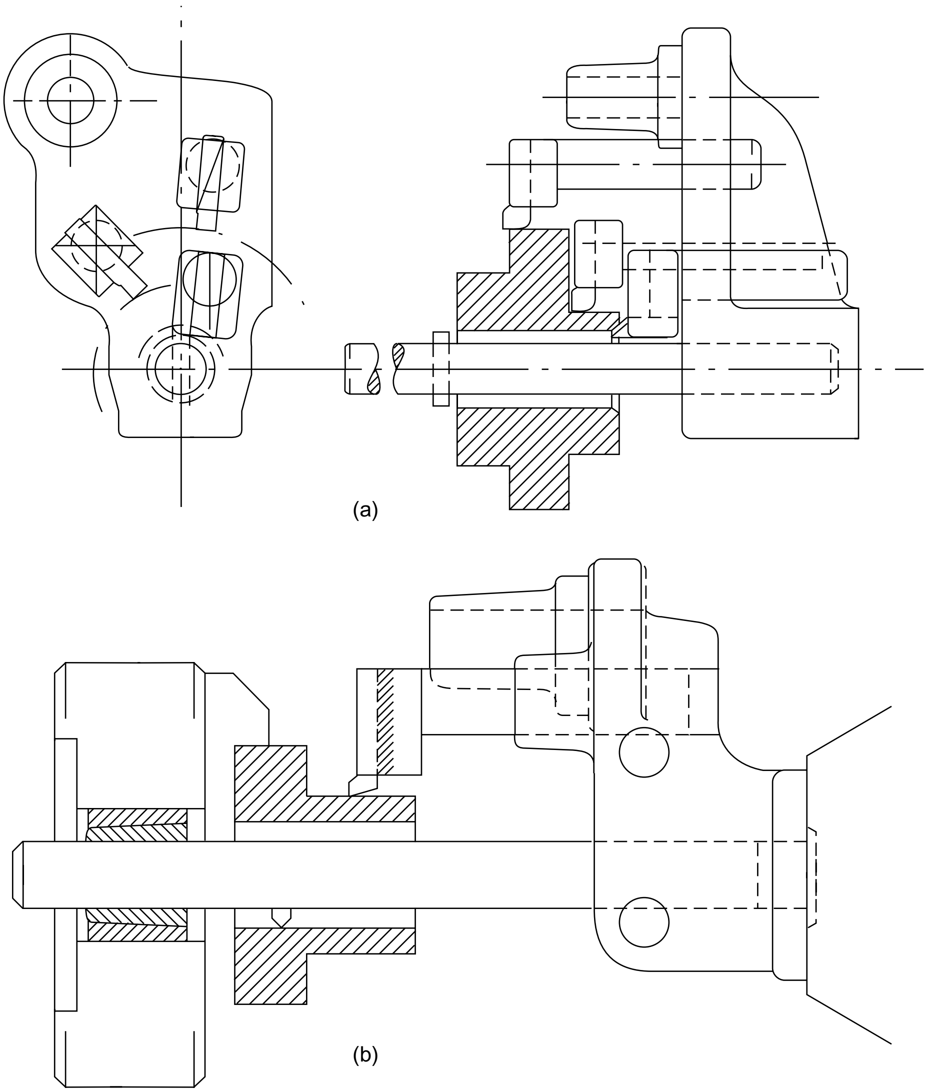


Fig. 5.4 *Special tooling used in turret lathes*

Many turret lathes would be fitted with taper turning attachment very similar to that used in centre lathes, for machining tapers. Small tapers can be produced by form tools from the cross slide, while internal tapers are produced by taper reamers.

Thus the various differences between capstan and turret lathes, and a general purpose centre lathe are:

1. Headstock has more range of speeds and is heavier to allow for higher rate of production.
2. Tool post is indexable (four tools). Any one tool can be brought into cutting position.
3. Tail stock is replaced by a tool turret with six tool positions.
4. Feed of each tool can be regulated by means of feed stops.
5. Two or more tools mounted on a single tool face can cut simultaneously.

6. Semi-skilled operators are required.
7. Used for production operations involving better repeatability.

A variation of the turret lathe is the capstan lathe, in which the turret moves on the saddle while the saddle can itself be fixed at any position on the bed, depending upon the length of the job as shown in Fig. 5.5. Thus, the tool travel length is limited to the length of the saddle. This type of arrangement is normally used for small size machines.

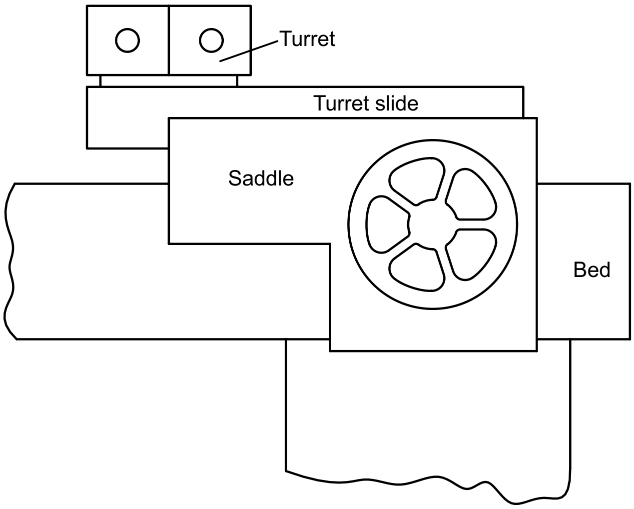


Fig. 5.5 *Intermediate slide arrangement in capstan lathe*

The various differences between capstan and turret lathes are given in Table 5.1.

Table 5.1 **Differences between capstan and turret lathes**

Capstan Lathe	Turret Lathe
Short slide since the saddle is clamped on the bed in position	Saddle moves along the bed, thus allowing the turret to be of large size.
Light duty machine, generally for components whose diameter is less than 50 mm	Heavy duty machine, generally for components with large diameters such as 200 mm
Too much overhang of the turret when it is nearing cut	Since the turret slides on the bed, there is no such overhang

5.2.1 Typical Tool Layouts

The tool layout for a given job is the predetermined order of machining operations to be performed to produce it. An efficient tool layout produces accurate parts as per the requirements, at the most economical cost. The tool layout is generally influenced by the nature of the job, the condition of the raw materials whether casting, forging or bar stock, the amount of stock to be removed and the number of pieces to be produced.

It is very important that the planner should fully understand the capability of the turret lathe and the available tooling, to reduce the total machining time. Very careful consideration is to be given for planning the best tool layout. A few rules that one should consider while planning the operations on turret lathes is given below:

1. For small batches use the standard tooling as far as possible and make the layout simple.
2. Cuts should be combined as far as possible. For example a tool from the hexagonal turret along with another from the square tool post on the cross slide can cut simultaneously. It would also be desirable to increase the number of tools operating simultaneously.
3. Similarly it is also necessary the handling operations be combined with the cutting operations such that total cycle time is reduced.
4. The planning for the finishing operations must be done till the end of the cycle. In between, there is a possibility of spoiling the finished surface. Also the combination of rough and finish operations in the same cycle should be done only when there is no detrimental effect on the quality of surface produced.
5. When multiple cutting tools are cutting at the same time, they should be so arranged that the cutting forces by the different tools get balanced.
6. If a given surface is achieved in a number of cuts, a finishing cut with a single tool is desirable in the interest of quality.
7. When concentricity is desired between two or more surfaces, all such surfaces should be machined in single setting only.
8. Contoured form surfaces are better obtained in two cuts rather than in single cut as far as possible.
9. While doing any heavy operation such as threading, care has to be taken to consider the rigidity of the work piece. Do not carry out any operation in the early stages, which reduces the rigidity of the component. Examples are deep grooves or large bores.
10. It is desirable to use centre drill before final drilling in case of small size drills. This would give rise to better drill axis location and smooth drilling.
11. Cored holes should normally be expanded and finished by boring and not by drilling.
12. In the case of stepped holes, make the large size hole first and follow with the small hole later. This would help in reducing the total drill travel and also reduce the machining time. The small drill would also not have to travel a long distance, which is always difficult.
13. To drill very long holes (e.g. length > 3 diameter) special care has to be taken. For example frequent withdrawal of the tool from the hole for flushing chips lodged in the flutes with cutting fluid is necessary in deep hole drilling, which is termed as peck drilling.

Some typical tool layouts are shown in Fig. 5.6 to 5.8 to get an idea about the range of capability and productivity that can be obtained with the turret lathes.

Fig. 5.6 shows the tool layout and machining plan for a cast iron V-belt pulley. The casting is held in the capstan lathe by means of a standard 3-jaw chuck. A total of 10 operations are required to complete the job. The operational sequence and the type of tools required are shown in Fig. 5.6 which is self-explanatory. Generally the roughing operations are completed before doing the finishing to size for accuracy considerations.

Fig. 5.7 shows the tool layout for the closing sleeve of a collet chuck. The component is made of alloy steel. The sequence of operations is as follows:

1. Drill 45 mm diameter hole
   (a) Rough form 16 mm groove from the cross slide tool
2. (a) Rough bore 58.74 mm diameter
   (b) Rough turn 90.5 mm outside diameter
3. Rough bore 74.6 mm diameter

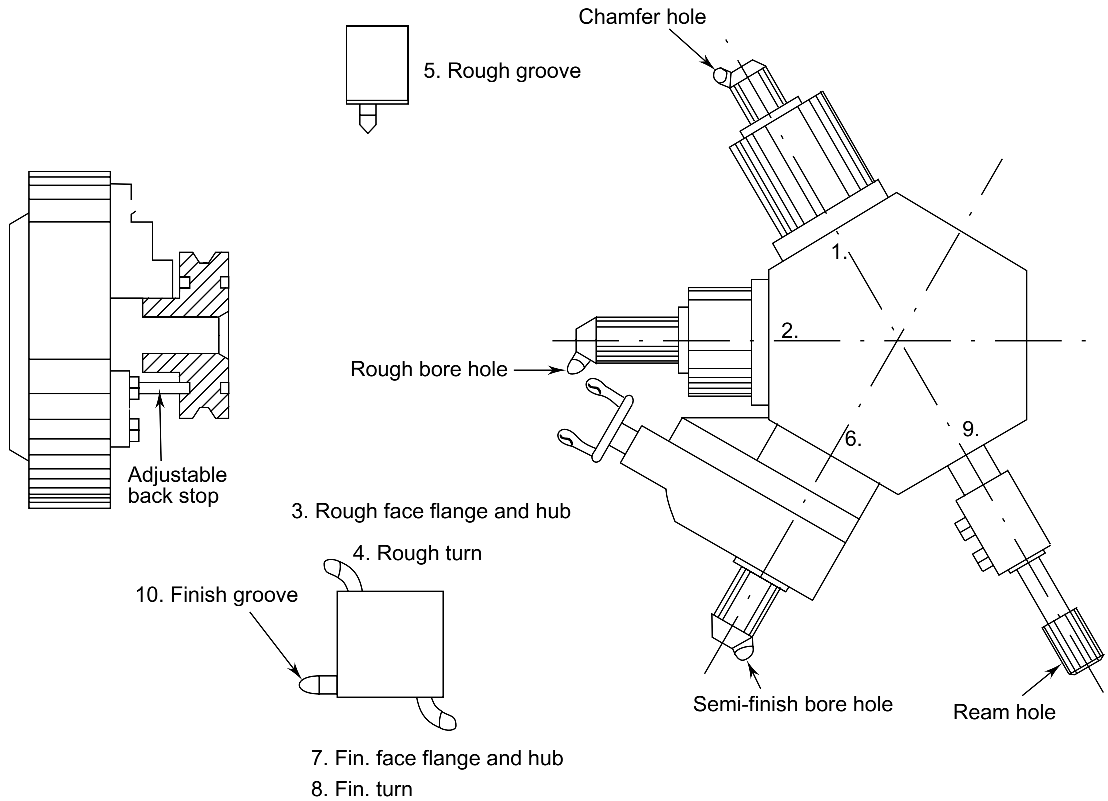
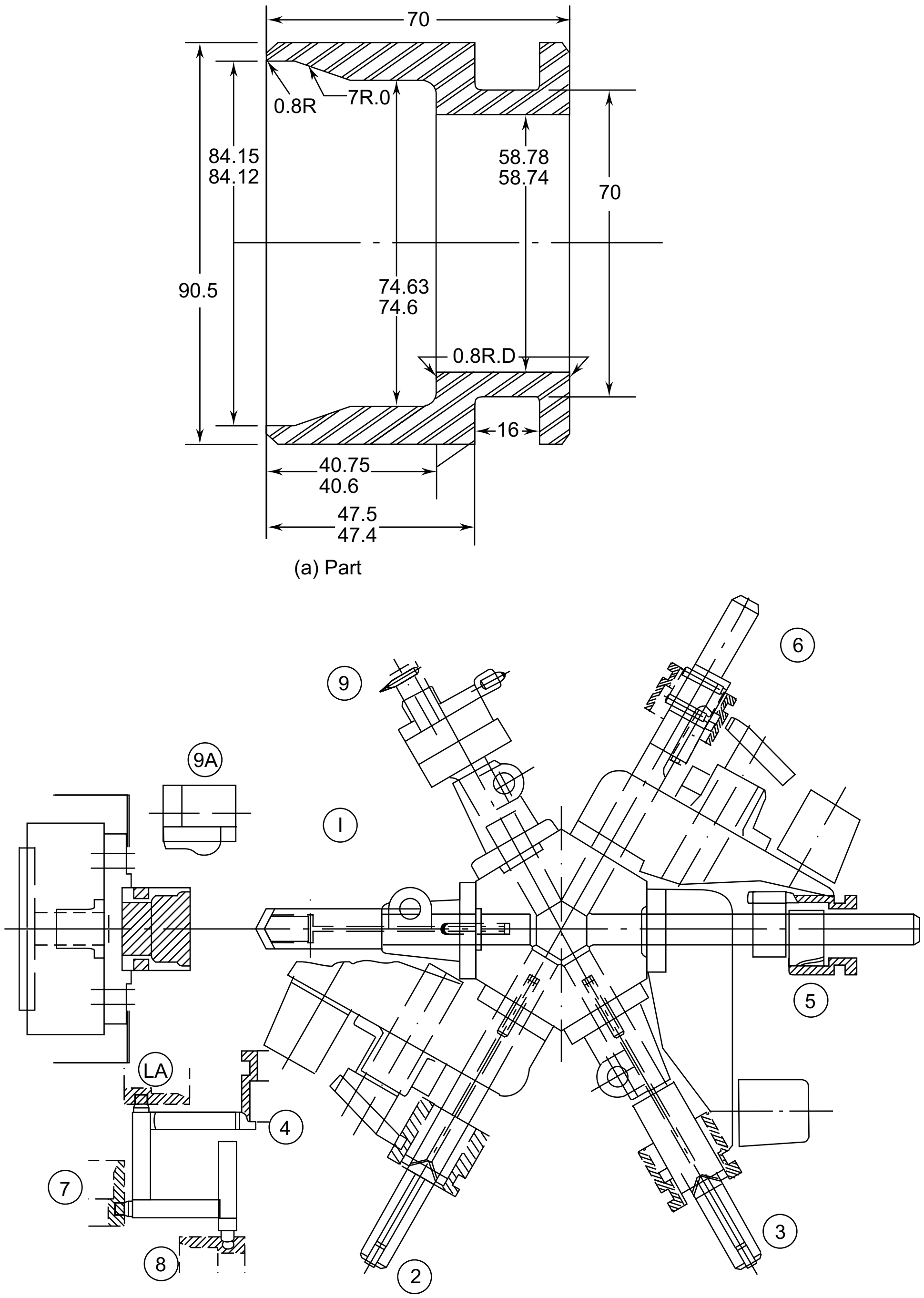


Fig. 5.6 *Tooling layout of cast iron V-belt pulley casting*

4. Finish and chamfer the end face
5. Rough form 16 mm and 7 mm radii and also form 0.8 mm radius
6. (a) Finish bore 58.74 mm and 74.60 mm diameter
   (b) Finish turn 90.5 mm outside diameter
   (c) Form 0.8 mm radius on 58.74 mm diameter
7. Finish form 16 mm groove
8. Chamfer corners of 16 mm groove
9. Finish profile to form 16 mm and 7 mm radii

Figure 5.8 shows a brass pipe bend casting being machined in a turret lathe. The part is held in a pneumatic chuck fitted with a setting block and balance weight in view of the odd shape of the part. The operational sequence is as follows:

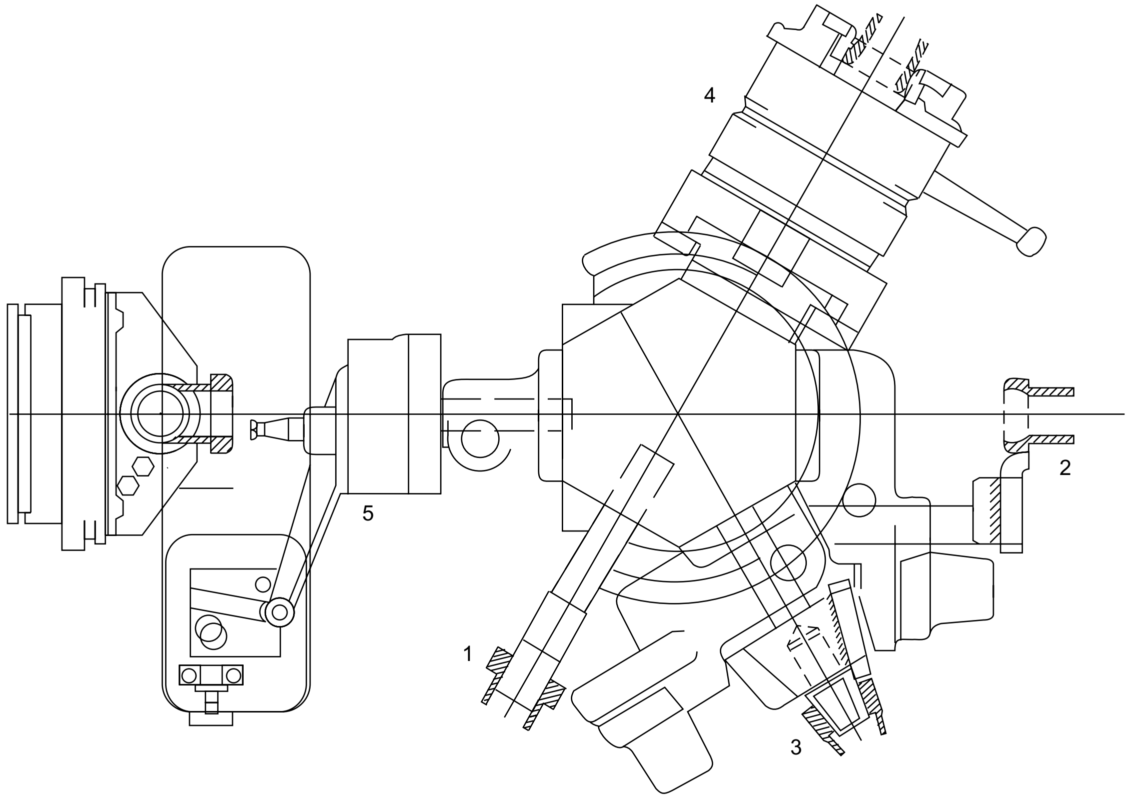
1. Load the casting using a loading bar
2. Turn thread diameter 'Á' using a tool in the knee turning tool
3. (a) Taper bore 'B' using taper boring reamer
   (b) Face end 'C' and chamfer 'D' using a facing and chamfering form tool in the holder
4. Cut the external threads on 'Á' using a die head and automatic cam operated elevating holder
5. Finish turn taper 'B' using a single point turning tool



(a) Part

(b) Tool layout

Fig. 5.7 *Tooling layout of closing sleeve of a collet chuck made of steel*



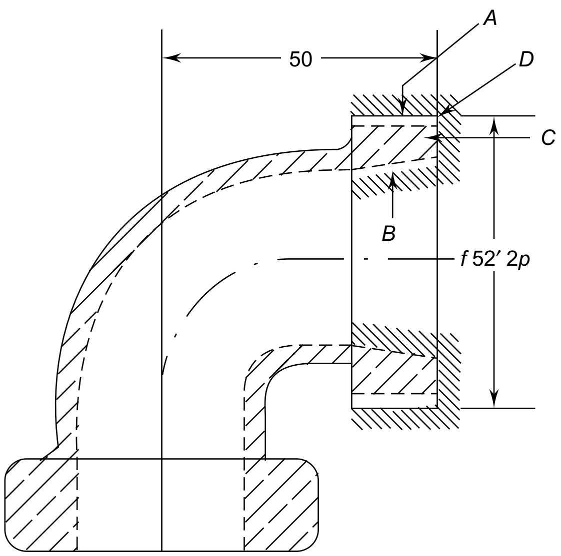


Fig. 5.8 *Tooling layout of brass pipe bend casting*

5.3 AUTOMATIC LATHES

The term automatic is somewhat loosely applied, but is normally restricted to those machine tools capable of producing identical pieces without the attention of an operator after each piece is completed. Thus after setting up and providing an initial supply of material, further attention beyond replenishing the material supply is not required, until the dimensions of the work pieces change owing to tool wear.

All operations in machining a work piece on a metal cutting machine tool are classified as processing and handling operations. Processing operations are those in which the actual cutting process or chip removal takes place. The rest are handling operations and include loading and clamping the work, advancing and withdrawing the cutting tools, releasing and unloading the work, checking the size of the work, etc.

In up-to-date machine tools, the processing operations are performed by the operative mechanisms of the machine tool. Handling operations are performed in various ways in different machine tools. A part or even all of the handling operations are performed in certain cases by corresponding mechanisms. The operators of other types of machine tools perform the handling operations themselves.

The faster the working and handling operations are performed in a machine tool, the less time will be required to produce a work piece and more work pieces can be produced in the same period of time by a given machine tool, i.e. higher is the productivity of the machine tool. If more handling operations are performed by the machine tool without the participation of the operator, he will spend less time on attending the given machine tool, and can handle other machine tools at the same time.

Highly automated machine tools especially of the lathe family are ordinarily classified as automatics and semi-automatics.

Automatics as their name implies are machine tools with a fully automatic work cycle.

Semi-automatics are machine tools in which the actual machining operations are performed automatically in the same manner as on automatics. In this case however, the operator loads the blank into the machine, starts the machine, checks the work size and removes the completed piece by hand.

5.3.1 Classification

The automatic lathes may be classified based on their

- size
- type of blank machined
- processing capacity (operations performed)
- machining accuracy obtained
- principle of operation design features
- number of spindles and work positions
- type

A typical classification of the automatic lathes is given in Fig. 5.9. In accordance with the arrangement of the spindles, they are called horizontal or vertical.

The vertical machines are more rigid and more powerful than the horizontal models and are designed for machining large diameter work of comparatively short length. Vertical machines occupy less floor space in the shop but require higher bays than horizontal machines.

Automatic bar machines are designed for producing work pieces of bar or pipe stock while magazine loaded automatic lathes process work from accurate separate blanks.

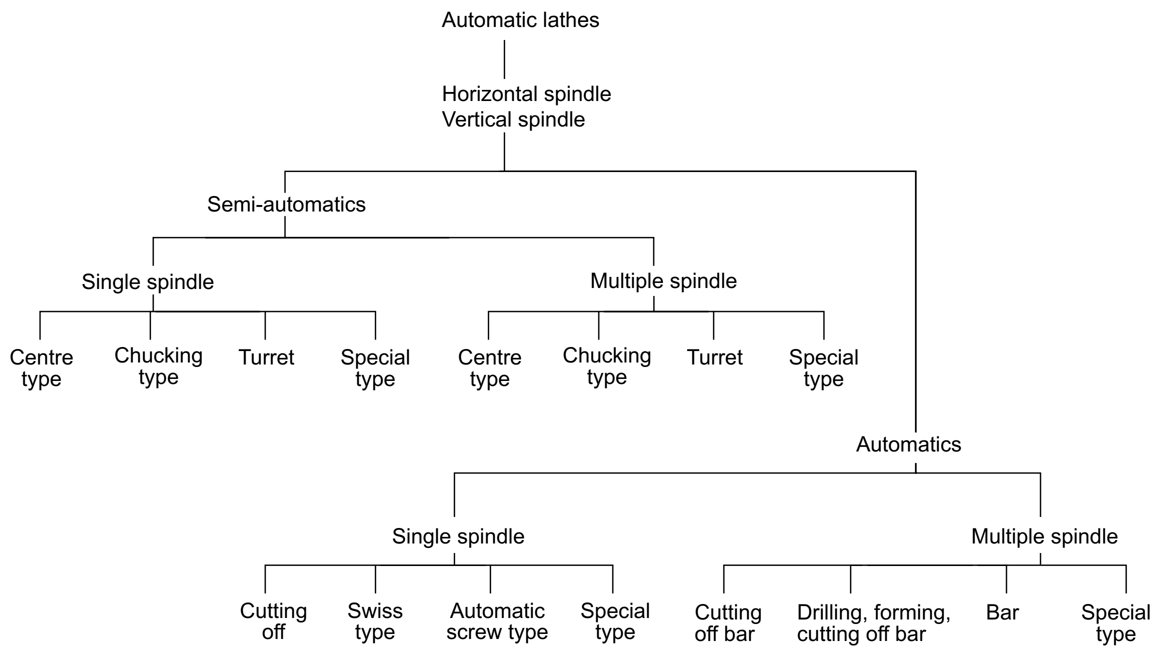


Fig. 5.9 *Classification of automatic lathes*

Chucking machines are employed for machining separate blanks (hammer or die forgings, castings or pieces of previously cut-off bar or pipe stock).

Automatic bar machines are employed for the manufacture of high quality fastenings (screws, nuts and studs), bushings, shafts, rings, rollers, handles, and other parts usually made of bar or pipe stock. The machining accuracy obtained by these automated machines depends on the type of machine and cutting tool employed.

Multiple spindle machines may have two to eight spindles. Their production capacity is higher than that of single spindle machines but their machining accuracy is somewhat lower. The rate of production of a multiple spindle automatic is less than that of the corresponding number of single spindle automatic machines. The production capacity of a four spindle machine, for example may be about 2 ½ to 3 times more than that of a single spindle machine.

The typical operations carried out on automatic lathes are:

- centring
- turning cylindrical, tapered and formed surfaces
- drilling
- boring
- reaming
- spot facing
- knurling
- thread cutting
- facing
- cutting off

5.3.2 Cutting off Machines

They can produce short work pieces of simple form by means of cross feeding tools. These automatics are simple in design. The headstock with the spindle is mounted on the bed. Two cross slides are located on the bed at the front end of the spindle as shown in Fig. 5.10. Cams on a cam shaft actuate the working movements of the cross slides through a system of levers.

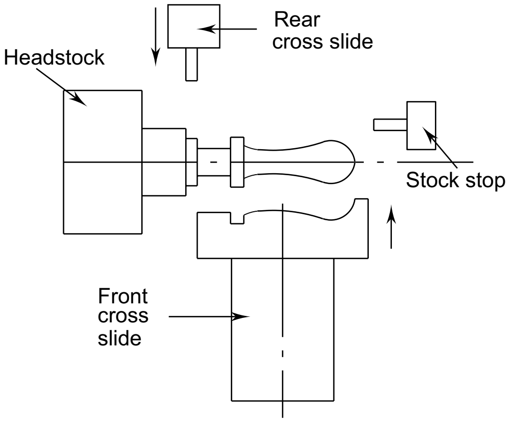


Fig. 5.10 *Typical arrangement of tool slide in a cutting-off type automat*

Typical machining sequence of a cutting-off machine comprises of the following sequence:

1. Stock stop advances to the working position
2. Stock is fed till it meets the stock stop
3. Stock is chucked
4. Stock stop is withdrawn
5. Rapid approach of the tool slides
6. Working feed of the tool slides
7. Rapid return of the tool slides
8. Release of the stock

Depending upon the requirement some of these operations are carried out simultaneously. Setting up of these machines is a simple procedure with very little adjustment and resetting.

5.3.3 Swiss Type Automatics or Sliding Headstock Automatics

These are designed for machining long accurate parts of small diameter (4 to 25 mm). An exclusive feature of these machines is the longitudinal travel of the headstock or of a quill carrying the rotating work spindle. The end of the bar, projecting from the chuck, passes through a guide bushing (steady rest) beyond which the cross feeding tool slides are arranged. A wide variety of formed surfaces may be obtained on the work piece by co-ordinated alternating or simultaneous travel of the headstock (longitudinal feed) and the cross slide (approach to the depth of cut). Holes and threads are machined by attachments.

The bar stock used in these machines has to be highly accurate and is first ground on centre less grinding machines to ensure high accuracy. These consist of two rocker arm tool slides (front and rear) on which the turning tools are normally clamped. In addition three radial slides are arranged for additional tools.

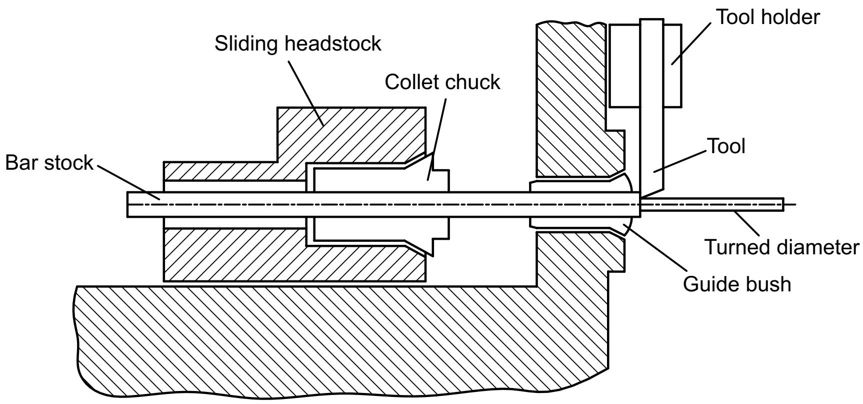


Fig. 5.11 *Schematic of a Swiss automatic lathe*

5.3.4 Automatic Screw Machines

These are essentially wholly automatic bar type turret lathes. These are very similar to capstan and turret lathes with reference to tool layout, but all the tool movements are cam controlled such that full automation in manufacture is achieved. They are designed for machining complex external and internal surfaces on parts made of bar stock or of separate blanks. Up to ten different cutting tools may be employed at one time in the tooling of such a screw machine. The tools are clamped in the holes at the positions of the periodically indexing turret and in the cross slide. The stationary headstock, mounted on the left end of the bed, houses the spindle, which rotates in either direction. The turret slide is arranged at the right end of the bed and carries the turret having six tool holes. Two cross slides (front and rear) are provided for cross feeding tools. A vertical slide attachment that provides an additional (third) cross slide may be employed. It is installed above the work spindle. All movements of the machine units are actuated by cams mounted on the cam shaft. Since these are fully automated, an operator can take care of more than two single spindle screw cutting or also called as bar automatic lathes, unlike a turret lathe for which an operator is required for each machine tool.

5.3.5 Multiple Spindle Automatics

In these machines there is more than one spindle where the work piece can be mounted. As a result, more than one work piece can be machined simultaneously in these machines. The number of spindles present could be four, five, six or eight. Each of the spindles is provided with its own set of tools for operation. The possible types of multi spindle machines are:

- Parallel action
- Progressive action

In parallel action machines the same operation on each spindle and a work piece is finished in one working cycle. This means that as many work pieces can be simultaneously machined as there are spindles. Such machines have a very high rate of production but may be applied for very simple work only, since the whole machining process takes place at one position.

These are usually automatic cutting off bar type machines as shown in Fig. 5.12 and are used to produce the same work as single spindle automatic cut off machine.

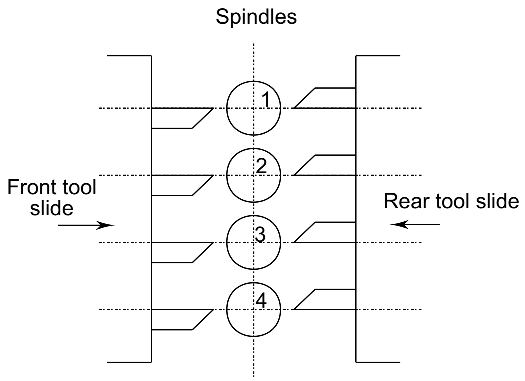


Fig. 5.12 *Parallel action multi spindle machine tool*

In progressive action machines, the blanks clamped in each spindle are machined progressively in station after station as shown in Fig. 5.13.

As a rule, these machines have a headstock mounted at the left end of the base. It contains the spindle carrier. Independent side tool slides are provided on both sides of the spindle carrier. The end tool slide, which accommodates tooling for all of the spindles, travels on the spindle carrier stem.

The spindles of these machines are mounted on a carrier which periodically indexes through a definite angle. In one revolution of the spindle carrier the number of components produced is equal to the number of spindles. It is evident then that many complex shapes can be produced by progressive type machines.

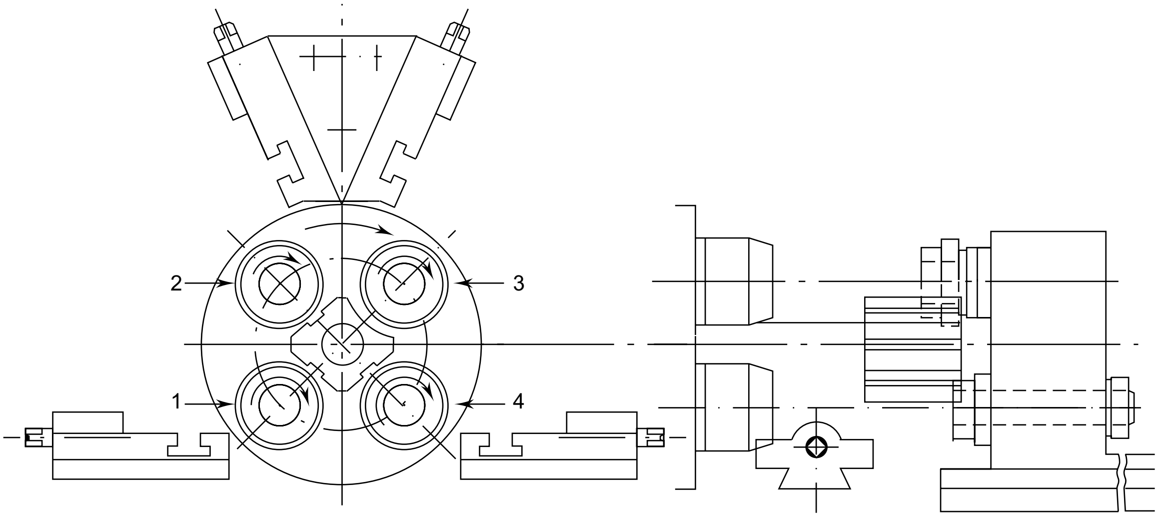


Fig. 5.13 *Progressive action multi-spindle machine tool*

There can also be a parallel progressive machine.

The accuracies that can be achieved using the automatics are given in the following Table 5.2.

TABLE 5.2 Accuracies achievable in Multi spindle machines

Types and Bar Capacities or Chucking Capacities, mm	Maximum Out of Roundness	Maximum Taper mm per Length mm	Maximum Diameter Variation in a Single Lot, mm
Swiss type automatic			
3 to 6.5	0.01	0.01 per 50	0.02
6.5 to 10	0.01	0.01 per 50	0.03
16 to 25	0.01	0.02 per 100	0.04
Automatic screw machines			
10 to 16	0.015	0.02 per 50	0.04
25 to 40	0.015	0.03 per 100	0.05
Single spindle semi-automatic			
80 to 100	0.015	0.03 per 100	0.08
120 to 200	0.020	0.03 per 150	0.10
250 to 300	0.030	0.03 per 200	0.15
400 to 500	0.040	0.03 per 250	0.20
Multiple spindle automatic bar machine			
25 to 40	0.015	0.03 per 100	0.08
65 to 100	0.020	0.03 per 150	0.10
Multiple spindle semi-automatic chucking machine	0.015	0.03 per 100	0.10
80 to 100	0.020	0.03 per 150	0.12
120 to 200	0.030	0.03 per 200	0.20
250 to 300	0.040	0.03 per 250	0.25
400 to 500			

Work Holding and Feeding

A chucking device must clamp the stock or blank reliably and rotate it with an accuracy that guarantees finished work of the specified form and size. Most of the time spring collet chucks are generally used with automats. They are very accurate besides the quick action, which can be achieved by the use of a simple lever. Typical accuracy of spring collet is

12 mm dia 0.02 to 0.03 per 30 to 35 mm
40 mm dia 0.02 to 0.05 per 100 mm

Movement of the tool slides in case of automatic lathes is controlled by means of cams operated by a cam shaft which is linked to the main spindle drive. Mostly plate cams are used for this purpose while drum cams are used in multi-spindle automats. Cam controls the time, length of the tool stroke, as well as feed rate as illustrated in Fig. 5.14. The cam design has to be individually done for each of the component and is one of the most crucial parts in the use of automated lathes.

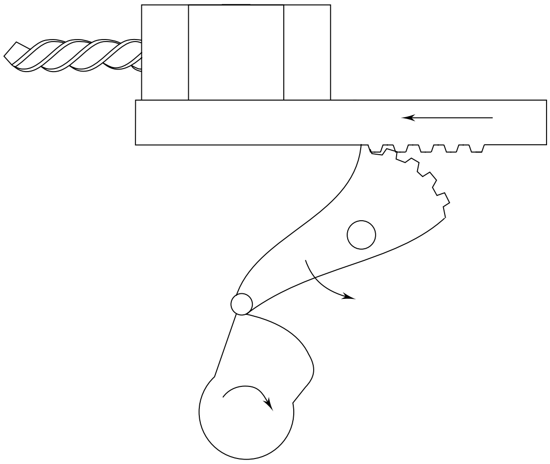


Fig. 5.14 *Cam controlling the tool stroke in automatic machine tool*

5.4 TOOLING LAYOUT AND CAM DESIGN FOR AUTOMATIC LATHES

The tool layout and cam design for a job constitutes the predetermined plan for the order and method of the machining operations necessary to produce it. Accuracy and cost of manufacture are largely dependent on an efficient layout. The following steps are recommended while planning the layout for an automatic lathe.

1. Choose the best available machine taking into consideration the availability as well as price of the component.
2. Determine the sequence of operations.
3. Choose the available standard tooling as far as possible.
4. Decide on any possible design for special tooling if absolutely necessary.
5. Based on the machine capability and surface finish desired, decide the cutting process parameters for each of the tool to be used. In case of heavy jobs, the available spindle power of the machine may be verified.
6. Check the movement of each of the tool in conjunction with the work piece for machining.
7. Arrange for any overlap of operations to reduce the total cycle time.
8. Compute the processing time including the idle time and from that the number of revolutions needed for each operation.
9. Calculate the spacing required on the cam periphery.
10. Draw the tool layout and cam details while verifying all the tool movements and clearances.

5.4.1 Tool Layouts

The making of a knurled screw using a single spindle automatic machine tool is shown in Fig. 5.15. It requires operations like external turning, drilling, threading (tapping), knurling and parting off. The following is the sequence in which they are carried out.

1. The bar is fed against a stop located in the first turret position.
2. Index the turret and rough turn the large diameter using an overhang turning attachment. At the same time the drilling of the large hole is done to the full depth using a twist drill located in the second turret position.

3. Index the turret and knurl the external diameter using a knurling tool located in the third turret position.
4. Index the turret and drill the small hole to the full depth using a twist drill located in the fourth turret position. At the same time the overhang form tool is used to do the chamfering. In the same operation (1a) turn the external profile using circular form tool located in the front cross slide.
5. Index the turret and cut the external thread using a die located in the fifth turret position.

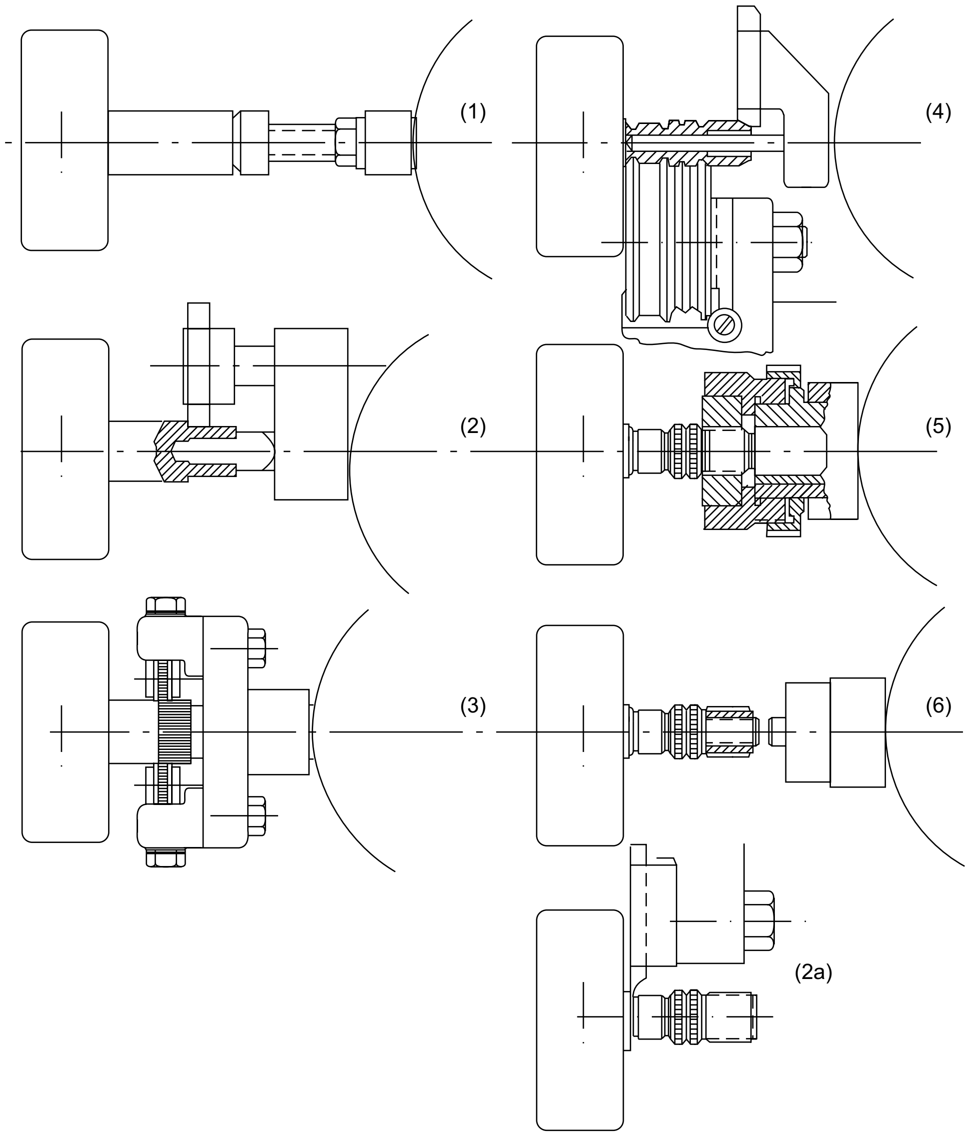


Fig. 5.15 *Tooling layout for a knurled screw machining in a single spindle automatic machine tool*

6. Index the turret and cut the internal thread using a tap located in the fifth turret position. Simultaneously the component is parted off using the parting tool located in the rear cross slide which is generally used for parting off.

The cam profiles as used for carrying out the above machining operations are shown schematically in Fig. 5.16.

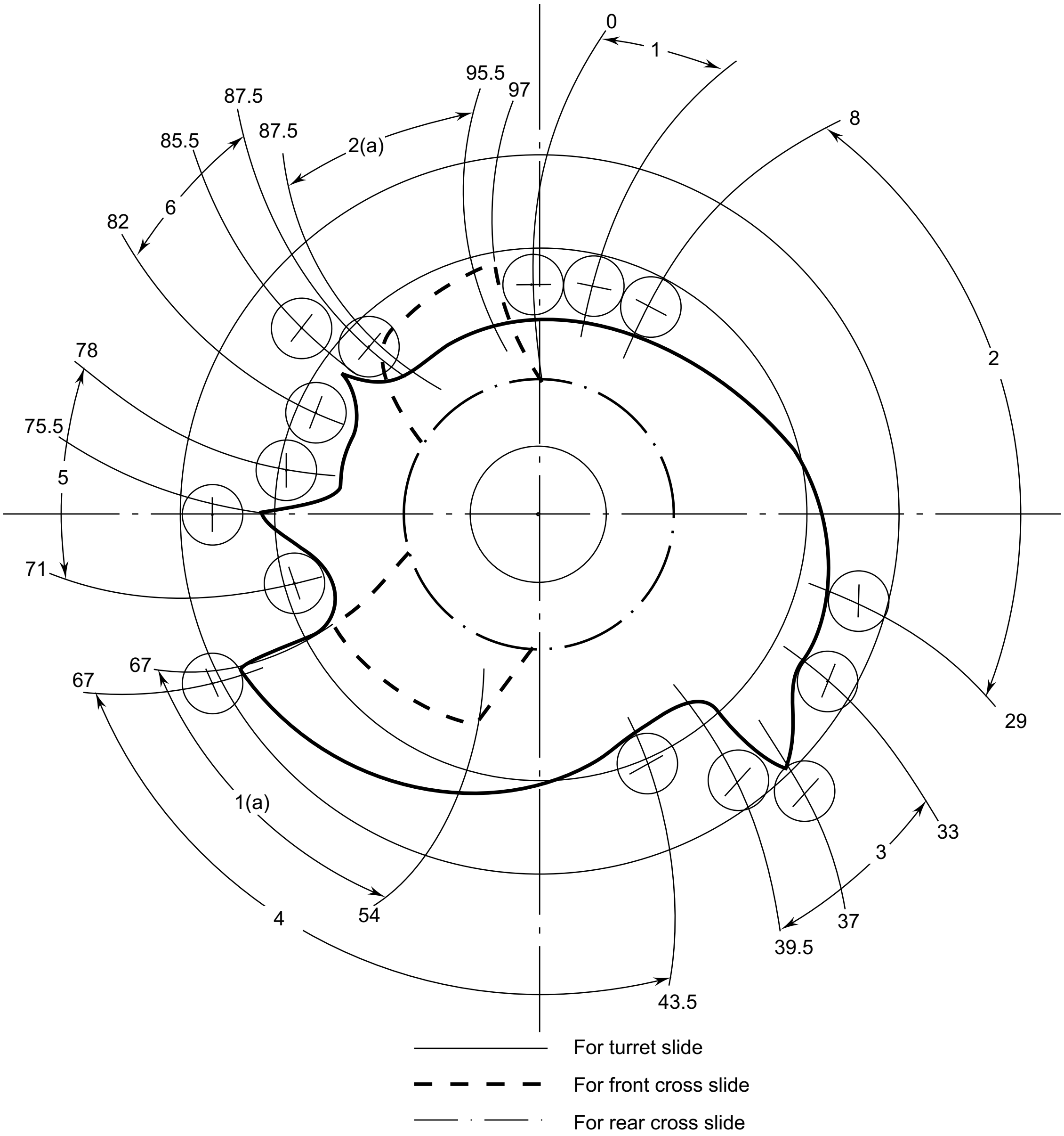


Fig. 5.16 *Cams used for carrying out the operations shown in Fig. 5.15*

5.4.2 Cam Design

To carry out the cam design, it is necessary to consider the operation sequence in great details, to determine the amount of movement to be provided to each of the tool together with their relative timing to achieve the necessary part geometry. For this purpose, a few details need to be looked into before the actual cam design.

Tool Travel or Throw

In each instance the distance the tool must travel is finally determined by carefully considering the component drawing; often a large scale drawing of the component will assist in this direction. Normally an amount varying between 0.125 to 0.180 mm, sometimes more, is added to the computed figures to provide an "approach" so that the tool when under control of the cam makes contact free from shock.

Dwell

With a number of operations, the tool is given a dwell so that any spring caused by the cutting forces may be eliminated and the correct size obtained. Normally stated in revolutions of the machine spindle, the dwell varies from 3 to 15, depending upon the actual operation.

Overlapping Operations

A problem encountered with the majority of automatic lathe layouts arises when planning is to be done for the simultaneous use of tools placed in both the cross slide and turret. Much depends upon the characteristics of the component, but the aim, when considering the overlapping of one or more stages in the production cycle, is to achieve an increase in the hourly output.

Example Knurled thumb screw shown in Fig. 5.17 is to be produced from a brass rod of 16 mm in diameter. The analysis of the component gives the following sequence of operations:

	Operation	*Tool*	*Tool position*
1.	Turn the external dia of M8	Hollow mill	Turret
2.	Knurl	Knurling tool	Turret
3.	Cut the external threads	Threading die	Turret
4.	Form the head shape	Form tool	Rear cross slide
5.	Parting off	Parting off tool	front cross slide

For the work material (free cutting brass), a feed rate of 0.15 mm/rev and spindle speed of 2100 rpm has been selected in view of the small diameter of the component.

Step 1: Calculate the tool travel required for each of the operation.

Hollow milling tool:

Approach	0.4 mm
Cutting (20 – 0.4)	18.6 mm
Total travel	19.0 mm

Knurling tool:

Approach	1.6 mm
Length to knurl	6.0 mm
Over run	1.6 mm
Total travel	9.2 mm

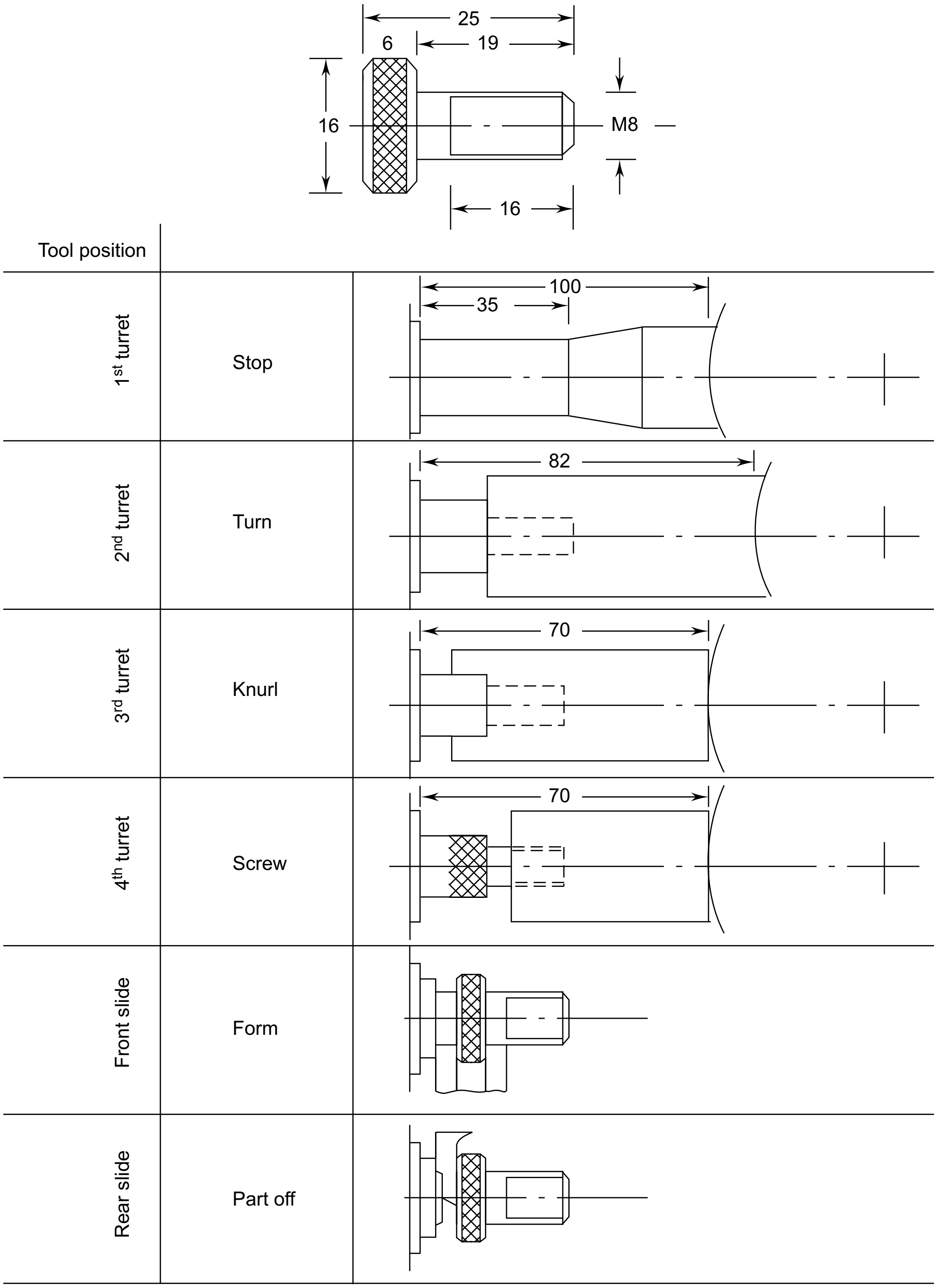


Fig. 5.17 *Machining plan for the knurled thumb screw in a single spindle automatic machine tool*

Threading tool:

Approach	7.0 mm
Length of thread	16.0 mm
Total travel	23.0 mm

Forming tool:

Approach	0.25 mm
Cutting 0.5(16 - 8)	4.0 mm
Total travel	4.25 mm

Parting tool:

Approach to cut position	4.25 mm
Cutting 0.5(16 - 8)	4.0 mm
Allowance for trailing angle	1.40 mm
Over run	0.4
Total travel	10.05 mm

Step 2: Convert the tool travel in terms of the number of revolutions of the work piece by deciding on the feed rate.

Hollow milling tool:
Revolutions = 19 ÷ 0.15 ≈ 127

Knurling tool:
Assume a feed rate of 0.50 mm/rev for running on while it is 1.0 mm/rev for running off.

Revolutions running on = 9.2 ÷ 0.50 ≈ 19
Revolutions running off = 9.2 ÷ 1.00 ≈ 10

Threading (Die) tool:

For M8 thread, pitch = 1.25 mm
Number of threads = 16 ÷ 1.25 = 12.8 ≈ 13
Revolutions = Number of threads + approach ≈ 13 + 5 = 18

The threading operation needs to be done at a lower cutting speed compared to the other operations. Hence, taking a correction ratio of 5, the actual revolutions required is 90

Forming tool:

A feed rate of 0.06 mm/rev is assumed for this operation.
Revolutions of approach = 4.25 ÷ 0.06 ≈ 71
Revolutions for dwell = 10
Total revolutions 81

Parting tool:

Assume a feed rate of 0.13 mm/rev for approach while 0.06 mm/rev for cutting.
Revolutions for advancing = 4.25 ÷ 0.13 ≈ 33
Revolutions for cutting = 5.8 ÷ 0.06 ≈ 97
Total Revolutions 130

Step 3: Check for any possibility of overlapping operations.
This component allows very limited possibilities and hence no overlapping is possible.

Step 4: Tabulate the above calculations with reference to machine tool requirements. The total number of revolutions required is 505. A check with the machine manual says that a 21 second cycle has 612 revolutions. The number of revolutions is therefore corrected to reflect this value in the last column of the table on next page.

Tool Position	Operation	Cam Rise mm	Feed mm	Revolutions of Work Spindle			
				Pre Operation	Non Cutting	Est. Work Cycle	Corrected Cycle
1	Feed Index			24 24	24 24		29 24
2	Turn Index	19	0.15	127 24	 24	127	127 24
3	Knurl on Knurl off Clearance Change speed and index	9.2 9.2	0.50 1.00	19 10 24	 24	19 10	21 12 24
4	Screw Change speed and index	24	1.25	18 24	 24	90	92 24
5	Form head and dwell	4.25	0.06	81		81	81
6	Part off advance Cutting Clearance and index	4.25 5.8	0.13 0.06	33 97 24	 24	33 97	33 97 24
	Total			529	144	457	612

Step 5: The corrected cycle having calculated along with the cam rises required are located on the type of cams to be used for each of the slides (turret, front slide and rear slide) to get the actual cam profiles on standard cam blanks.

SUMMARY

Centre lathe is good for general machining applications in a job shop but is not suitable for production application.

- Capstan and turret lathes improve productivity by utilising a number of tools simultaneously.
- By controlling the tool travel in sequence, capstan and turret lathes allow for controlling the precise geometry of the part.
- A number of types of automatic lathes are developed that can be used for large volume manufacture applications such as single spindle automatics, Swiss type automatics and multi spindle automatics.
- In automatic lathes, the tool motion is controlled by cams for precise geometry control.
- Tool layouts signify the proper sequence of the action of the tools, which can then be converted into a cam profile that is used for the control of the tool.

Questions

5.1 What are the various types of automatic lathes you are familiar with in the multi-spindle category? Explain their differences and applications. Is the productivity of multi spindle machines higher compared to single spindle automatic lathe? Explain your answer.

5.2 Describe with a simple sketch the operation of a Swiss type (sliding headstock) automatic lathe in terms of operations possible, tools used, method of setting and types of components made on the lathe.

5.3 What are the differences between automatic lathe and capstan lathe? Give an example component suited for capstan lathe with dimensions.

5.4 What is the classification method that could be used for the different type of automatic lathes used in the industry? Explain briefly about each of the types in two sentences.

5.5 What is the importance of tool layout in automats? Explain with an example any one type with a component sketch.

5.6 Briefly explain with neat sketches the types of work holding devices that are commonly employed in automatic lathes. Specify the limitations of them.

5.7 Describe the method of operation of the Swiss type automatic lathe, with applications and tools used. What is its speciality?

5.8 List three most commonly employed types of single spindle automatics.

5.9 Sketch four parts suitable for production on a Swiss-type automatic.

5.10 Sketch four parts not suited for production on a single spindle automatic.

5.11 What are the steps of procedure involved in changing over to a new part production on a Swiss-type auto?

5.12 List the items of specification of a Swiss-type of automatic lathe.

5.13 Name the industry in which the Swiss type of automatic lathes are used.

5.14 How does an automatic cutting- off machine differ from a Swiss type machine?

5.15 List the number of tool locations on the Swiss-type auto.

5.16 State the features of machine construction, which lead to the attainment of high accuracy of the parts.

5.17 What is the accuracy requirement of the bar stock for this machine?

5.18 Give a sketch illustrating the principle of operation of the Swiss-type automatic.

Multiple Choice Questions

5.1 Automatic lathes are used
- (a) For automated production
- (b) In repair shops
- (c) For single piece production
- (d) For small batch production

5.2 Capstan lathes can be used
- (a) For fully automated production
- (b) In repair shops
- (c) For single piece production
- (d) For small batch production

5.3 In a capstan or turret lathe the tool motion is stopped
- (a) By cams mounted on the lead screw
- (b) By trip dogs mounted on the feed rod
- (c) By special mechanism
- (d) When it reaches the end of the work piece

5.4 A box tool is normally used in turret lathes
- (a) For turning jobs with multiple tools
- (b) For short turning jobs
- (c) Long turning jobs since it supports the work piece to reduce its deflection under cutting force
- (d) For facing operation

5.5 In an automatic lathe the motion of the tool is controlled
- (a) By cams
- (b) By trip dogs
- (c) By special mechanism
- (d) By motors

5.6 Automatic lathes of cutting-off type are used only for
- (a) Any type of parts
- (b) Simple parts produced by cross feeding tools
- (c) Small parts with different contours
- (d) Long parts

5.7 Swiss type automatic lathes are used only for
- (a) Small axi-symmetric parts
- (b) Long parts
- (c) Simple parts produced by cross feeding tools
- (d) Parts with cross holes

5.8 Special feature of the Swiss type automatic lathe is
- (a) Any type of tool motion is possible
- (b) Headstock slides for feeding the tools
- (c) Simple parts can be produced by cross feeding tools
- (d) Parts with cross holes can be produced

5.9 In a progressive action type multi-spindle automatic lathe, the spindles are arranged in such a way that
- (a) Each spindle produces identical parts
- (b) Each spindle has different types of tools and produce different surfaces on the same part
- (c) Some spindles carry only the parting-off tool
- (d) Some spindles carry only the drill

Answers to MCQs

5.1 (a)	5.2 (d)	5.3 (b)	5.4 (c)	5.5 (a)
5.6 (b)	5.7 (a)	5.8 (b)	5.9 (b)	

CHAPTER 6

Reciprocating Machine Tools

Objectives

The reciprocating machine tools utilise single point cutting tools to generate plane surfaces. After completing this chapter, the reader will be able to

- Understand the operation of a shaper for producing plane surfaces
- Learn the operation of a planer for producing large plane surfaces
- Understand the operation of a slotter for producing slots in vertical axis

6.1 INTRODUCTION

In the previous chapters, discussion was centred on the lathes and the various varieties that are used for general purpose work as well as mass manufacture. In the lathes the component is rotated while the cutting tool is axially moved to generate cylindrical surfaces. In the present chapter, the machines which use only reciprocating motion are discussed. The major machine tools that fall in this class are:

- Shaper
- Planer
- Slotter

The main characteristics of this class of machine tools are that they are simple in construction and as a result are very economical in initial cost as well as cost of operations.

6.2 SHAPER

The shaper is a relatively slow machine tool with very low metal removal capability. Hence it is replaced by more versatile milling machine in many shops. This is a low cost machine tool and hence is used for initial rough machining of the blanks. It is rarely used in production operations.

It uses a single point tool similar to a lathe which is clamped to a tool post mounted to a clapper box which in turn is mounted to a reciprocating ram as shown in Fig. 6.1. The ram while undertaking the cutting stroke pushes the cutting tool through the work piece to remove the material. When the ram returns, no cutting takes place. In between the return and cutting strokes, the table moves in the horizontal direction perpendicular to the cutting direction, which is termed as the feed direction.

6.2.1 Shaper Construction

The main constructional pieces of a mechanical shaper are:

- The Base
- Column
- Ram
- Tool head (clapper box)
- Cross rail
- Table

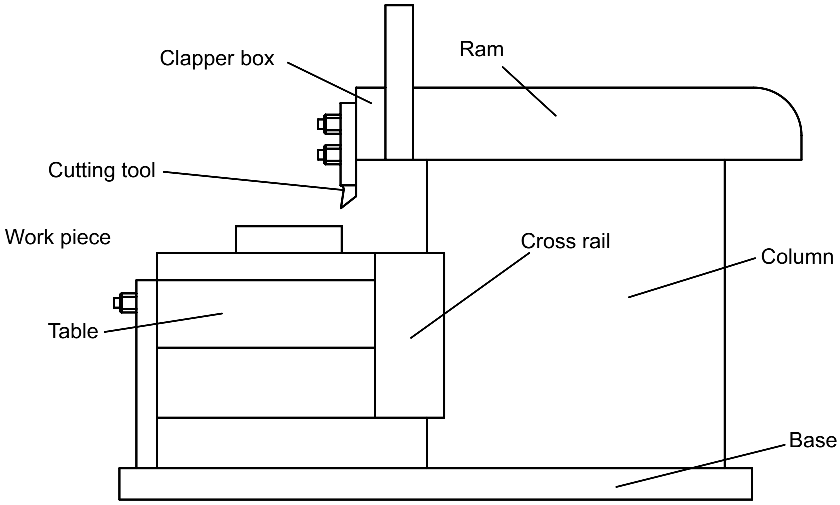


Fig. 6.1 *Schematic block diagram of a shaper with main parts shown*

The Base The base provides the stability for the shaper as it supports all other equipment present as well as absorbs the forces coming due to the cutting. Generally it is made of grey cast iron and has the necessary arrangement of bolts so that it can be bolted to the factory floor with proper levelling.

Housing (column) The housing is a box like structure to provide the necessary rigidity and also houses all the motors and power transmission equipment. On the top of the housing the necessary guideways are provided for the linear motion of the ram for the cutting stroke.

Ram It is the part of shaper that provides the reciprocating motion for the cutting tool. Ram gets the motion directly from the quick return mechanism (described later) present in the housing.

Tool head (clapper box) The single point cutting tool is clamped in the tool head as shown in Fig. 6.2. The tool head has the ability to swivel the cutting tool at any angle while clamping the tool with any overhang depending upon the requirement. The swivelling ability is important for the tool to machine surfaces that are not in horizontal plane. Further the tool should be firmly supported during the forward motion to carry out the material removal. During the return stroke, the cutting tool will not do the cutting and hence will be an idle stroke. If the tool is held firmly as in the cutting stroke, the tool will rub the already machined work piece and also the flank surface of the tool will wear out quickly. To reduce this, the tool is lifted during the return stroke by the clapper box arrangement as shown in Fig. 6.2.

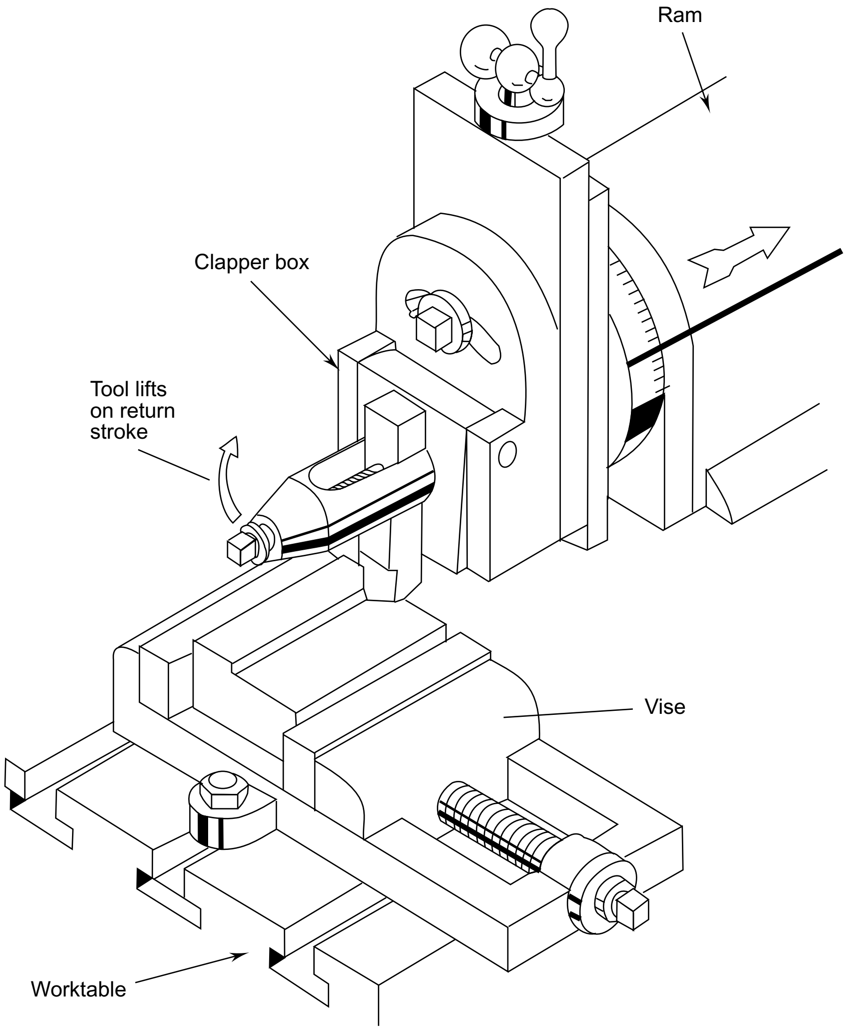


Fig. 6.2 *Typical arrangement of work piece and the tool in shaper*

Table A heavy table is present at the front end of shaper. Table is provided with T-slots for mounting the work pieces or work holding fixtures. The table can be moved up and down along the guide ways provided on the cross rail attached to the housing.

A shaper is generally used for machining flat surfaces in horizontal, vertical and angular directions. It can also be used for machining convex and concave curved surfaces. The actual surface generated is by means of the linear motions of the cutting tool. The feed rate and the depth of cut are so arranged that the resultant surface is a flat surface as shown in Fig. 6.3 schematically.

Because of the nature of the tool geometry, the actual surface generated consists of triangular hills with the angles that are based on the tool geometry. The feed rate has to be reduced to lower the hill to valley height and thereby improve the surface finish. This is schematically shown in Fig. 6.4 for different feed rates. It is also possible to use a broad nosed cutting tool, which has a flat cutting edge parallel to the work piece surface, to provide a surface that is smoother. This allows for larger feed rates thereby reducing the machining times. However, these are prone to chatter.

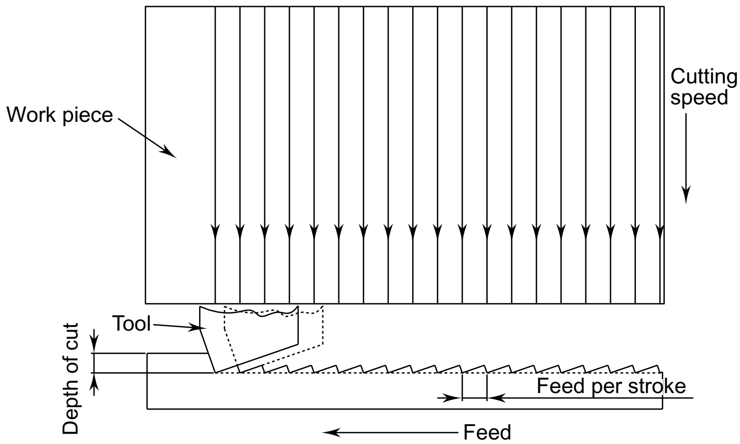


Fig. 6.3 *Generation of a flat surface with a single point tool*

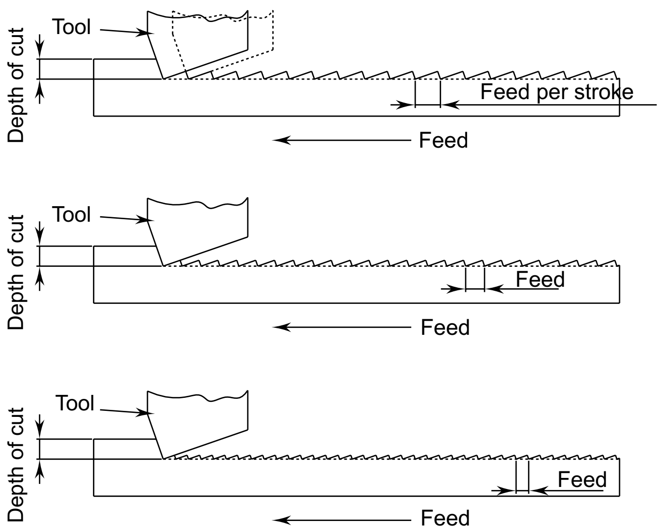


Fig. 6.4 *Effect of feed per stroke on the flat surface generated in shaping*

6.2.2 Quick Return Motion

The major motions required in a shaper are the reciprocating motion of the cutting tool and the auxiliary motion. During the forward motion of the tool cutting is performed while during the return stroke the tool will simply be sliding. Hence it is necessary to reduce the idle time of the machine to make the return stroke faster compared to the forward motion. This is generally accomplished by means of a quick return mechanism. One typical mechanism used in shapers is the quick return mechanism as shown in Fig. 6.5(a).

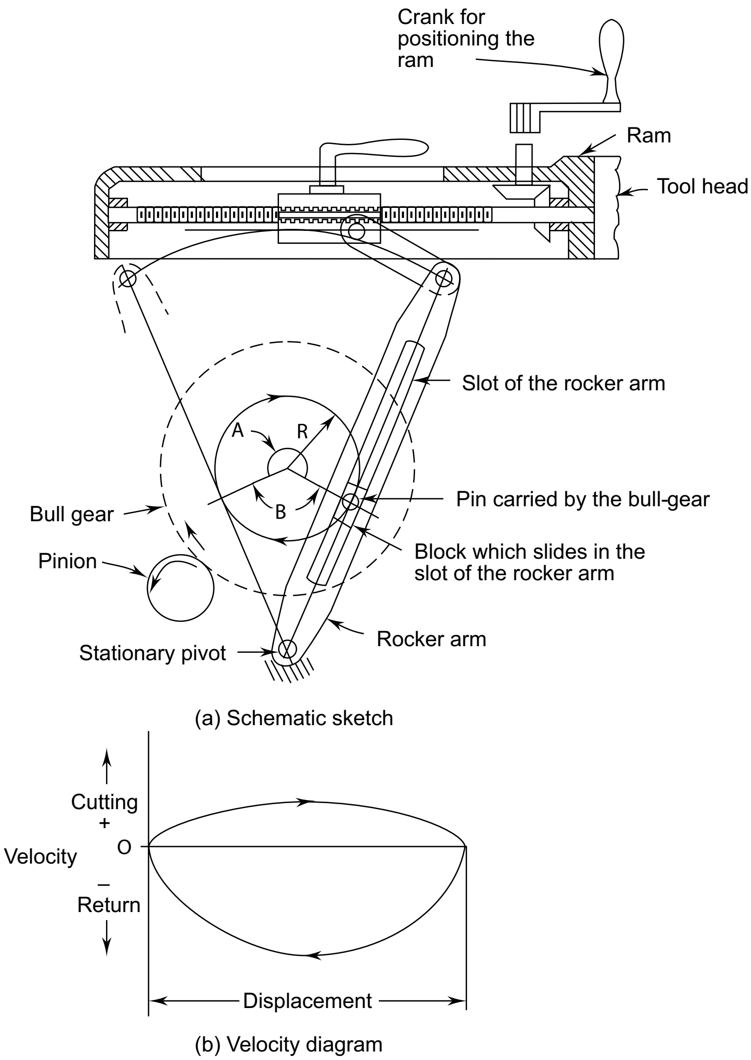


Fig. 6.5 *The quick return motion in a crank shaper*

The motor drives the bull gear, which carries a pin in a circular motion. The RPM of the bull gear is controlled by the motor. This pin fits into the slot of the rocker and is free to slide in a straight line path. As the bull gear rotates, the rocker arm oscillates about its pivot point. The end of the rocker arm is connected to the ram of the shaper through a link arm. The length of the stroke is changed by changing the radius of the circle in which the pin on the bull gear rotates. The length of travel should be a little longer than the actual length of the work piece. This allows sufficient time for the tool block of the clapper box to swing back to its position for cutting.

The typical velocity profile of the crank mechanism is shown in Fig. 6.5(b). As can be noticed the cutting speed continuously varies through the entire rotation of the bull gear. Also, the return stroke has higher velocity while the forward stroke has lower velocity.

An alternative to the crank mechanism as described above is a geared shaper, where one or two racks are fixed on the underside of the ram. The rotary motion from the motor is converted by the rack-pinion arrangement through appropriate belting and idling pulleys. Adjustable dogs on the ram, engage the belt shifting linkage to move the forward belt onto the fixed pulley for the cutting stroke, replacing it with the reverse belt for the return stroke.

6.2.3 Mechanical Feed Drive

Automatic feeding of the cutting tool at the end of the cutting stroke is obtained by moving the feed screw as shown in Fig. 6.6 through part of a revolution. The feed screw is engaged by a pawl that sits in a notched wheel attached to the feed screw. The pawl operates once for each of the rotation of the bull gear. During one revolution of the bull gear, the oscillating motion of the pawl carrier moves the pawl forward and then back by one or more teeth, depending upon the feed rate that was set. The oscillating motion is generally obtained by means of a crank pin arranged to make one cycle per revolution of the bull gear. The feed is normally given during the return stroke. The amount of feed is controlled by the number of teeth in the notched wheel that are moved during the return stroke.

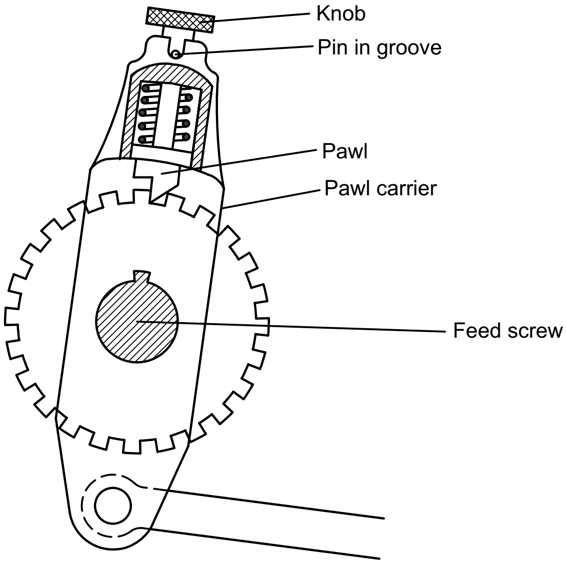


Fig. 6.6 *Intermittent feeding arrangement in mechanical shapers*

6.2.4 Shaper Specifications

Shapers can be specified by means of a number of parameters as follows:

- Maximum length of stroke, mm
- Maximum table size, length, mm × width, mm × height, mm
- Maximum table travel, length, mm × width, mm
- Maximum power of the drive motor used in the machine, kW
- Range of cutting speeds, strokes/min
- Range of feeds, mm/stroke
- Maximum weight of the machine
- Maximum dimensions of the machine for installation (Floor space)

6.2.5 Types of Shapers

The standard mechanical shaper is described above. In addition to that, there are a few more variations of shapers that are used in the industry. They are:

Universal shaper It is similar to the mechanical shaper, but has a table that can be tilted about an axis parallel to the ram ways as well as an axis that is perpendicular to the ram ways.

Draw cut shaper Similar to the mechanical shaper, but built much heavier. The heavier construction helps it to take much deeper cuts without any chatter thereby increasing the material removal rate. Another difference is that the cutting stroke is when the ram moves towards the body rather than the away motion, as is most common in the mechanical shapers.

Vertical shaper It is similar to a mechanical shaper except the reciprocating axis here is vertical. This is similar to a slotter (described later) in action and often used interchangeably. The vertical shaper is a much smaller version of the slotter, and was developed for tool room work. The stroke of the ram is generally limited below 300 mm. The main difference between the two is that while the slotter only allows for vertical motion, the vertical shaper has the ability to adjust the ram in the vertical position about 15° from the vertical. This will help in cutting of proper clearances in tools and dies.

6.2.6 Hydraulic Shaper

The mechanical shaper has the problem of inertia of the main drive components, which require some time for reversal for every stroke and as a result, a large proportion of time is spent with the tool cutting air. An alternative drive system can be provided by means of a simple hydraulic circuit to provide the reciprocating motion shown schematically in Fig. 6.7.

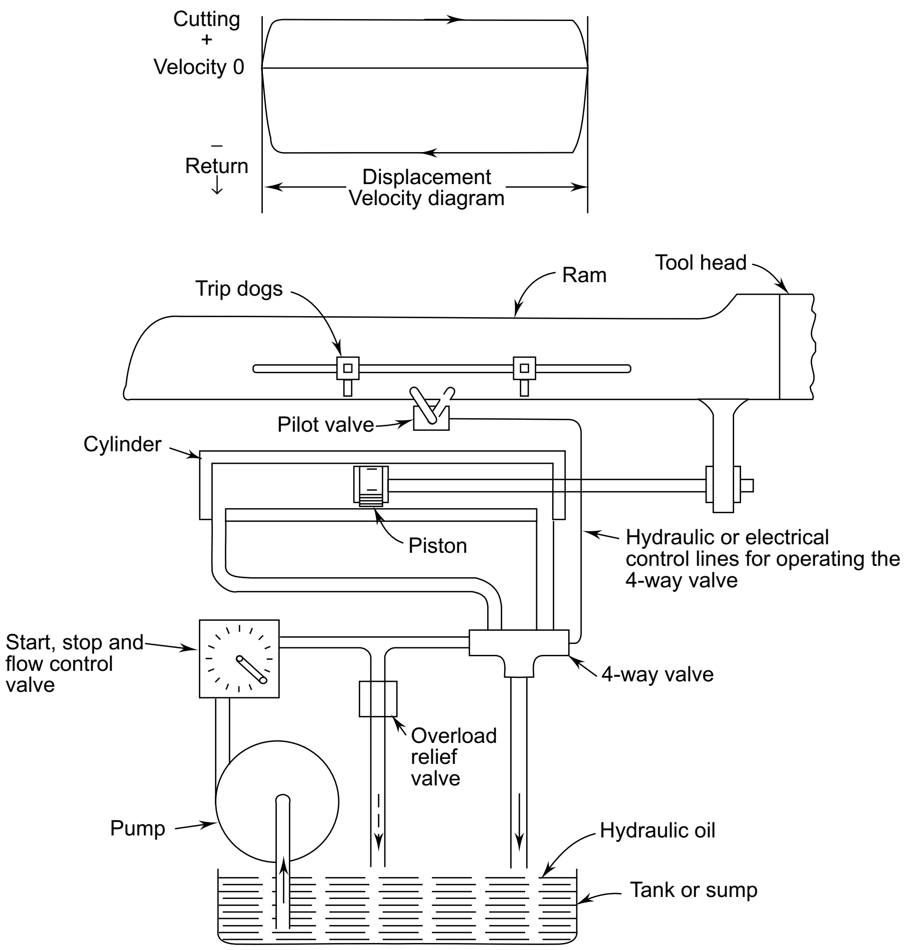


Fig. 6.7 *Schematic representation of the functioning of a hydraulic shaper*

In a hydraulic shaper, the ram is connected to a hydraulic cylinder, which is controlled by means of a 4-way valve. The hydraulic fluid is pumped to the hydraulic cylinder through the 4-way valve. The 4-way valve is also connected to the sump. 4-way valve controls the direction of high pressure fluid into the cylinder thereby controlling the direction of motion, either the cutting stroke or return stroke. The flow control valve controls the flow rate of the hydraulic fluid thereby controlling the speed with which the ram moves.

A finger operated lever serves the purpose of starting and stopping the machine. An adjustable trip dog operated lever controls the operation of the 4-way valve to control the reversal of the ram.

Advantages of hydraulic shaping

1. Cutting speed remains constant throughout most of the cutting stroke unlike the crank shaper where the speed changes continuously.
2. Since the power available remains constant throughout, it is possible to utilise the full capacity of the cutting tool during the cutting stroke.
3. The ram reverses quickly without any shock due to the hydraulic cylinder utilised. The inertia of the moving parts is relatively small.
4. The range and number of cutting speeds possible are relatively large in hydraulic shaper.
5. More strokes per minute can be achieved by consuming less time for reversal and return strokes.

Disadvantages

1. It is more expensive compared to the mechanical shaper.
2. The stopping point of the cutting stroke in hydraulic shaper can vary depending upon the resistance offered to cutting by the work material.

6.2.7 Work Holding in Shaping

The work pieces which are small are normally held in a vice fixed to the shaper table. Some other work holding arrangements such as angle plates, strap clamps and support elements can also be used. Their use depends upon the work piece shape and the geometry to be generated.

6.2.8 Cutting Tools Used

A large variety of single point cutting tools with various approach and edge geometries can be used in the shaper. Single point cutting tools similar to the types used in lathe are used in shapers too. The main difference is that the clearance angle required will have to be properly ground into the tool as there is no adjustment possible in a shaper's clapper box arrangement. Both the right hand left hand tools are used in shapers. Since the cutting speeds used are relatively low, high speed steel is the most widely used cutting tool material for shaping tools.

The cutting tool in the tool head needs to be rotated to a suitable angle to make it at right angles to the work piece surface being generated as shown in Fig. 6.8.

6.2.9 Shaping Time and Power Estimation

The stroke of the mechanical shaper should be adjusted in such a way that the tool starts the cutting stroke a small distance before the work piece and completes the stroke after a further small distance. This distance is necessary to give sufficient time for the reversal of motion and also the cutting does not take place at a very low speed, which is the case at the beginning and end of the stroke. The approach distance and over travel each can be taken as 15 to 25 mm.

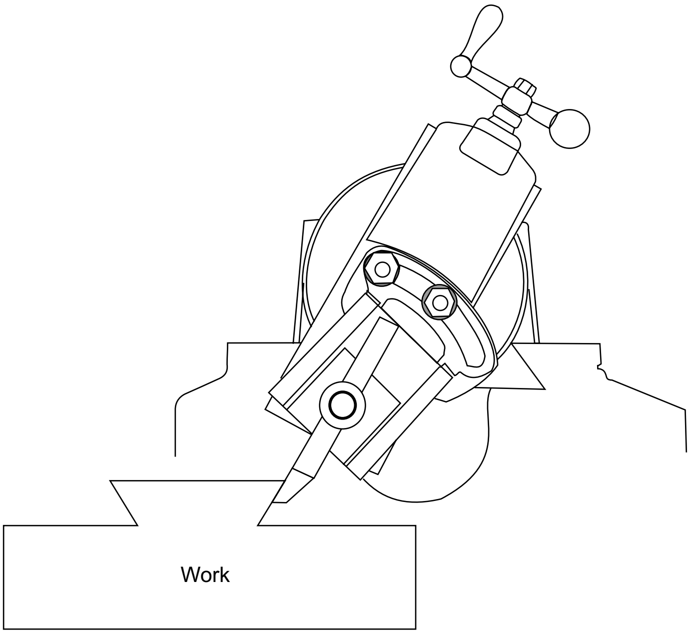


Fig. 6.8 *Swivelling of the tool head in shaper to suit the work piece surface*

The cutting speed in the case of shaping is the speed of the cutting tool in the forward direction during actual cutting. In the case of mechanical shaper it is the average speed. Let N be the rotational speed of the bull gear and L be the length of the stroke. The speed ratio indicates the proportion of time actual cutting is taking place and is defined as

$$\text{The speed ratio, } r = \frac{\text{Time for forward stroke}}{\text{Time for return stroke}} = \frac{N_f}{N_r} = \frac{3}{2} \text{ (normally)}$$

The value of r for typical mechanisms is about 1.5. This means that the time for completing the stroke is

$$\text{Time for completing the cutting stroke} = \frac{N_f}{N \times (N_f + N_r)}$$

The cutting speed is the speed of the ram in the cutting direction.

$$\text{Thus the cutting speed, } V = \frac{L \times N\,(N_r + N_f)}{N_f}$$

The time for completing one stroke, T is

$$T = \frac{L}{N} \text{ minutes}$$

Feed in shaping is the small lateral movement given to the tool in a direction perpendicular to the cutting speed direction. It is given before the beginning of each cutting stroke. This feed is specified as mm/stroke.

Number of strokes, S_N required for removing one layer of material from the surface of the work piece depends upon the width of the work piece, W and the feed rate, f employed.

Number of strokes required, $S_N = \frac{W}{f}$

Total machining time = $T \times S_N$

Example 6.1

A shaper is operated at 120 cutting strokes per minute and is used to machine a work piece of 250 mm in length and 120 mm wide. Use a feed of 0.6 mm per stroke and a depth of cut of 6 mm. Calculate the total machining time to for machining the component. If the forward stroke is completed in 230°, calculate the percentage of the time when the tool is not contacting the work piece.

Solution Let the approach distance = 25 mm

Length of stroke, $L = 250 + 25 = 275$ mm

Number of strokes required, $S_N = \frac{120}{0.6} = 200$

The time for completing one stroke, T is

$$T = \frac{275}{120} = 2.292 \text{ minutes}$$

Total machining time = 2.292 × 200 = 458.33 minutes

The forward stroke is during 230°.

Percentage of time when tool is not cutting = $\frac{360 - 230}{360} = 36.11\%$

The cutting speed, $V = \frac{275 \times 120 \times 360}{1000 \times 230} = 51.65$ m/min

Example 6.2

A part measuring 250 mm × 100 mm × 40 mm is to be machined using a hydraulic shaper along its wide face (250 mm × 100 mm). Calculate the machining time taking approach as well as over travel as 20 mm each. Take cutting speed as 5 m/min, and a machining allowance on either side of plate width is 3 mm and feed is 1 mm/stroke.

Solution Given the approach and over travel distance = 20 mm

Length of stroke, $L = 20 + 250 + 20 = 290$ mm

Width of the plate to be completed = 3 + 100 + 3 = 106 mm

Given Feed = 1 mm/stroke

Number of strokes required, $S_N = \frac{106}{1} = 106$

Cutting speed, $v = 5$ m/min = 5 000 mm/min = 83.3333 mm/s

For calculating time for one stroke, the total distance travelled is twice the stroke length (taking the return distance). The time for completing one stroke, T is

$$T = \frac{2 \times 290}{83.3333} = 6.96 \text{ seconds}$$

Total machining time = 6.96 × 106 = 737.76 s = 12.296 minutes

Power consumed

In a very simple way the power consumed can be calculated using the following formula.

$$\text{Power} = K \times \text{MRR HP}$$

Where MRR = material removal rate in mm^3

Example 6.3

Calculate the power in shaping steel with a hardness of 375 BHN with a depth of cut of 2.8 mm, the cutting speed 65 m/min and the work diameter is 40 mm. The feed rate used is 0.4 mm/rev.

Solution Material removal rate = $V f d = 1000 \times 65 \times 2.8 \times 0.4 = 72.8 \times 10^3$ mm^3

Power required = $79 \times 72.8 \times 10^3 \times 10^{-6} = 5.3$ HP

Table 6.1 The constant *K* for calculating the horse power consumed.

Material	Hardness BHN	Constant $K \times 10^6$ Machining Operation		
		Turning/Shaping	Drilling	Milling
Aluminium alloys	30–150	12	12	18
Copper alloys	20–80 R_B 80–100 R_B	37 61	31 49	37 61
Cast iron	110–190 190–320	49 85	49 79	49 98
High-temperature alloys	200–360	98	98	122
Nickel alloys	80–360	110	110	128
Steels	85–200 300–375 375–500 500–600	61 79 98 110	49 67 85 98	61 73 98 110
Stainless steel	135–275 300–450	73 73	61 67	67 79

6.3 PLANING MACHINE (PLANER)

Planing machine is very similar to the shaper in terms of the surfaces that can be generated. Generally planer is used for machining large work pieces, which cannot be held in the shaper. In the shaper, the cutting tool reciprocates during the cutting motion, while in the case of planer the work table reciprocates. Feeding motion in the planer is given to the cutting tool, which remains stationary during the cutting motion.

A typical planer is shown in Fig. 6.9. The main constructional pieces of a planer are:

- Bed
- Table
- Column
- Cross Rail
- Tool head

Bed The bed of a planer, generally made of cast iron, is large in size and consequently is heavier to support the heavy machine. Levelling jacks, or pads, are included on the bottom of the bed to provide a means of levelling it during installation. The bed is a little more than twice the length of the table to provide the full reciprocating motion for the entire part mounted on the table. Guide ways both *V* and flat as described in chapter 3 are used depending upon the size of the planer are present along the entire length of the bed.

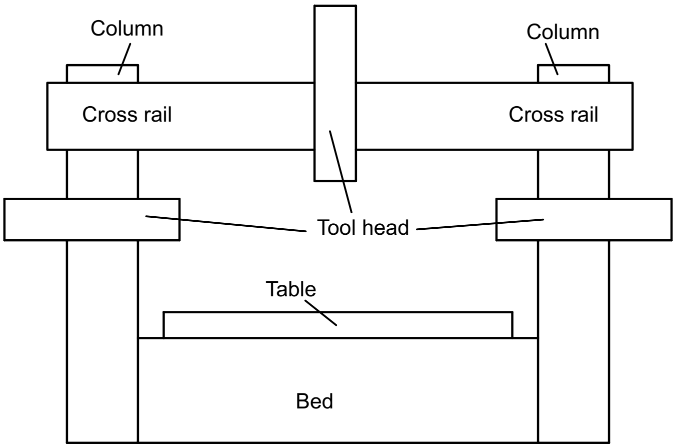


Fig. 6.9 *Schematic of a planing machine*

Table The table is the platform where the work will be setup for machining. The table supports and reciprocates the heavy work pieces during the cutting action. The table moves back and forth on the bed, carrying the work past the stationary tool. T-slots are provided on the entire length of the table for securely fastening the work pieces to the table. In addition to the T-slots, a row of holes are provided on the table to accept stop pins that prevent the movement of the work under the heavy cuts common to the planer. To reverse the table automatically at the end of each stroke adjustable dogs are provided on the side of the table. Tool feed is normally adjusted to occur during the return stroke of the work.

Similar to a shaper, the planer also has a quick return mechanism to save on idle time during the return stroke of the table. However the type of mechanism used is different. The customary return stroke of a planer is three or four times faster than the cutting stroke. This is normally achieved by the use of DC reversible motor and an associated pinion and herringbone rack under the table. The herringbone gears equalize the side thrust which is present when cutting tools are used.

Column On either side of the bed two heavily ribbed columns are located. Two tool heads can be mounted, one on each of these columns. Within the columns are the various mechanisms that transmit power to the upper parts of the machine from the main drive motor.

Cross Rail A cross rail is connected to the two columns to help in accommodating a third tool head. If necessary an additional tool head can be mounted on the cross rail. The rail can be clamped at any position on the columns by means of hand or power clamps. The tool heads can be fed manually or more generally by automatic feeding arrangement with the power coming from the motor and feed gearbox that is housed within the table.

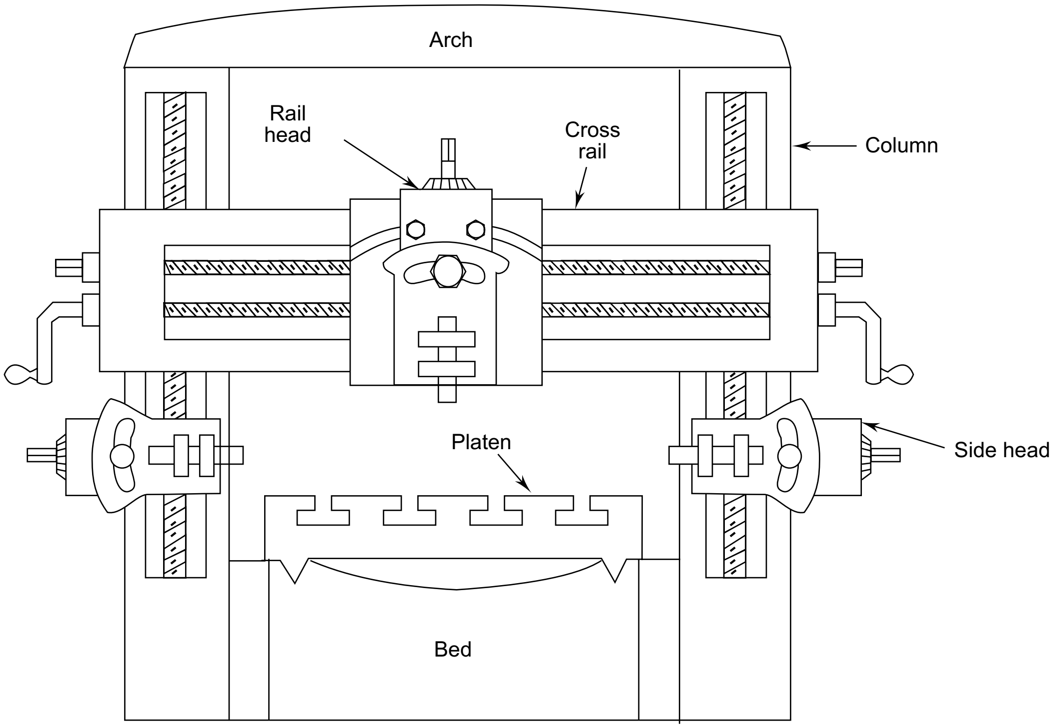


Fig. 6.10 *The construction of a planing machine*

Tool head The tool head in planer is similar in construction to the clapper box of a shaper and is mounted on the cross rail through a saddle. The tool head can be moved along the cross rail for the feeding action while the depth of cut can be controlled by moving the tool downwards. The tool head can be tilted for any given angle cut on the work piece. The tool heads will be provided with counterweights for force equalization and smooth movement. As shown in Fig. 6.10, it is possible to mount more than one tool head on the cross rail as well as on the columns on both sides, so that multiple surfaces can be completed simultaneously. This helps in reducing the total machining time since planing is a relatively slow operation like shaping.

The types of surfaces that are generally produced in planers are:

- Flat surfaces either in the horizontal plane or vertical plane
- Curved surfaces
- Any inclined plane
- Slots, grooves and dovetails

Typical examples of parts machined on a planer include the following: large castings, bases and tables for different types of machine tools, lathe beds, frames for printing presses, textile machines, forging-hammer die blocks, large fixtures and moulds, rolling-mill parts, and parts for large hydraulic presses.

The work piece during the machining operation will have to remain fixed to the table. Since heavy cuts are made in planing, the cutting forces are also large and the work piece should not be shifted due to these forces. Most of the time complex work pieces will be machined, so the setting up process requires heavy duty T-bolts, clamps, angle plates, planer jacks, step blocks and stops to cater to the wide variety of geometries that can be handled.

The cutting tools used in planers in general are single point cutting tools similar to shapers. However in view of the heavy cuts taken these normally have large cross section and are more rigid. If the entire tool is made from high speed steel, it becomes expensive. Tool holders made from less expensive heat-treated steel may be used to hold the high speed steel bits that are used for actual cutting. Cemented carbides tools can also be used with planers to get higher material removal rates. The tool geometries are similar to that of the shaper.

Similar to shapers, planers also can be mechanically or hydraulically driven.

6.3.1 Types of Planers

The type of planer that is common and described above is the double housing planer. Some other types of planers are as follows:

Open Side Planer Open side planer consists of only one vertical column or housing with the cross rail mounted as a cantilever. The column and the cross rail carry tool heads. This type of machine is suited for oversize work pieces.

Planer-miller This is a hybrid of a planer and a milling machine (Chapter 7), but is generally considered as a planer because of the size and construction. The construction is similar to a double housing planer, with a tool head which is different from the conventional planer tool head with a single point tool. The miller head holds a tool similar to a shell end mill (described in Chapter 7) driven by its own motor. The main advantage of this type of tool head is that more power can be provided to the tool so that higher material removal rates can be achieved.

Pit Planer A pit planer is used for work pieces that are larger than the ones taken by normal planers. In this the table and the work piece remain stationary while the cutting tool reciprocates across the work surface. The work piece is either mounted on a stationary table under the rail, or on the factory floor. No clamping of the work piece is required because of the massive sizes involved. One or two tool heads can be mounted on the cross rail as required. The tool heads will be travelling along the cross rails driving the cutting tool past the work surface during operations.

Divided Table Planer (Duplex Table Planner) The planer work in general calls for a lot of time in setting up the work piece because of the massive sizes involved. This type of planer has two work tables, such that when machining is taking place on one table the other work piece can be set on the other table, thereby increasing the productivity of the planer. It is also possible to join the two tables together for really large work pieces.

This would also be useful in situations in gang planing (a situation where a number of parts are to be machined at the same time) where setup time is generally large because of arranging all these parts in proper alignment.

Plate Planer (Edge Planer) It is a single purpose planer to machine the edges of heavy work pieces such as armour plates. The work piece is clamped on the bed and remains stationary while the side mounted carriage supporting the cutting tool is reciprocated along the edge of the work piece. Cutting can take place during both directions of carriage travel.

6.3.2 Planer Specifications

The geometric dimensions of the planer are identified by the width, height and length in that order. The width in this case refers to the maximum width of the work that can be planed on the machine, taking some margins around the housings. The height refers to the maximum height of the work that can be planed while the length of a planer refers to the maximum table stroke or to the maximum length of a piece of work that can be planed on the machine. In addition to these geometric specifications, the rest of the specifications are similar to that explained for shapers.

6.3.3 Planing Time Estimation

The suggested starting parameters for planning for common work materials are shown in Table 6.2. The time required to take one cut over a given surface in planing can be calculated based on the length of stroke, width of the surface to be cut, feed per stroke, cutting speed and table return speed. The approximate time can be calculated from the formula:

Table 6.2 Suggested Starting Speeds and Feeds for Planers

Work Material	HSS Tool	
	Speed, m/min	Feed, mm/Stroke
Aluminium	60–90	3.00
Brass (soft)	45–75	6.30
Bronze (medium)	20–40	1.90
Bronze (hard)	10–20	1.25
Cast iron (soft)	15–25	3.00
Cast iron (hard)	10–15	1.50
Malleable iron	15–30	2.25
Cast steel	10–20	1.25
Steel (soft)	20–30	1.25
Steel (medium)	15–20	1.50
Steel (hard)	5–10	0.90

$$\text{Planing time, } t = \frac{W}{f}\left(\frac{l}{V_c} + \frac{l}{V_r} + a\right)$$

Where l = length of the planer stroke, m
W = width of surface to be cut, mm
f = feed rate, mm/stroke
V_c = cutting speed, m/ min
V_r = table return speed, m/ min
a = time for reversal of table, min which is about 0.015 to 0.040 minutes

Due allowance should be made for the approach and over travel distance and side allowance for the width of machining while calculating the time, similar to those taken while calculating the shaping time. However their effect on machining time is very small and hence can be neglected.

Example 6.4

The flat surface of a large cast iron part measuring 2 m × 1 m × 300 mm is to be machined using a planer along its face (2 m × 1 m). Estimate the machining time taking approach as well as over travel as 20 mm each. Take cutting speed as 20 m/min, return speed is 40 m/min and a machining allowance on either side of plate width is 5 mm and feed is 1 mm/stroke.

Solution Given the approach and over travel distance = 20 mm

Length of stroke, $l = 20 + 2000 + 20 = 2040$ mm = 2.04 m

Width of the plate to be completed = 5 + 1000 + 5 = 1010 mm

Given Feed, $f = 1$ mm/stroke

Cutting speed, $V_c = 20$ m/min; $V_r = 40$ m/min

Take $a = 0.02$ minutes

$$\text{Planing time, } t = \frac{W}{f}\left(\frac{l}{V_c} + \frac{l}{V_r} + a\right) = \frac{1010}{1}\left(\frac{2.04}{20} + \frac{2.04}{40} + 0.02\right) = 174.73 \text{ minutes}$$

$$\text{Neglect the allowances, Planing time, } t = \frac{1000}{1}\left(\frac{2}{20} + \frac{2}{40} + 0.02\right) = 170 \text{ minutes}$$

Example 6.5

Estimate the time required to machine a grey cast iron part measuring 6 m long × 1.25 m wide on a double housing planer. Cutting tools used are made of carbide which can take a cutting speed of 80 m/min and return speed is 160 m/min. To finish the part it requires two rough cuts with a feed of 1.5 mm/stroke and two finish cuts with a feed of 0.5 mm/stroke. Assume the time for reversing the table is 0.02 minutes.

Solution Length of stroke, $L = 6$ m

Width of the plate to be completed = 1.25 m = 1250 mm

Roughing Passes:

Given Feed = 1.5 mm/stroke

Cutting speed, $v = 80$ m/min; Return speed = 160 m/min

Take $a = 0.02$ minutes

$$\text{Planing time, } t = \frac{W}{f}\left(\frac{l}{V_c} + \frac{l}{V_r} + a\right) = \frac{1250}{1.5}\left(\frac{6}{80} + \frac{6}{160} + 0.02\right) = 110.417 \text{ minutes}$$

Finishing Passes:

Given Feed = 0.5 mm/stroke

Cutting speed, $v = 80$ m/min; Return speed = 160 m/min

Take $a = 0.02$ minutes

$$\text{Planing time, } t = \frac{W}{f}\left(\frac{l}{V_c} + \frac{l}{V_r} + a\right) = \frac{1250}{0.5}\left(\frac{6}{80} + \frac{6}{160} + 0.02\right) = 331.25 \text{ minutes}$$

Total machining time = 2 × 110.417 + 2 × 331.25 = 883.333 minutes = 14.72 hours

6.4 SLOTTING MACHINE (SLOTTER)

Slotting machine is basically a vertical axis shaper. This is a larger version of the vertical shaper with ram strokes up to 1800 mm long. Thus the work pieces, which cannot be conveniently held in a shaper, can be machined in a slotter. Generally keyways, splines, serrations, rectangular grooves and similar shapes are machined in a slotting machine. The stroke of the ram is smaller in slotting machines than in shapers to account for the type of the work that is handled in them.

A typical slotter is shown in Fig. 6.11. The main constructional pieces of a slotter are:

- Base
- Table
- Column
- Ram

Base Similar to a shaper and planer the base of the slotter is a heavy structure to support all the weight of the machine tool and the accompanying cutting forces. Since the cutting force in slotting is directed against the table, the base of the machine is rigidly built. Precision guide ways are provided on top of the base for the cross-slide to move.

Table Table is generally a circular one similar to the rotary table of a milling machine. T-slots are cut on the table to facilitate the fixing of work pieces utilizing various fixturing elements such as T-bolts, clamps, etc. Tables on slotters can be rotated as well as moved longitudinally or transversely. With such flexibility in the feed direction, a slotter can cut any type of groove, slot, or keyway.

Column The column of a slotter is a support structure to the cutting tool and its reciprocating motion. It is also massive and houses the power and drive mechanism used for the reciprocation of the cutting tool.

Ram The ram holds and supports the tool head during the cutting action. Since gravity acts on the ram during its upward travel, a counterweight is added to equalize the power requirements on the upward and downward strokes. This will provide a smooth action to the machine. The actual cutting takes place during the downward motion of the tool. The stroke length can be adjusted suitably depending upon the part.

The types of tools used in slotter are very similar to that of a shaper except that the cutting actually takes place in the direction of cutting. However in view of the type of surfaces that are possible in the case of slotter, a large variety of boring bars or single point tools with long shanks are used. A typical component that is being machined in a slotter is shown in Fig. 6.12.

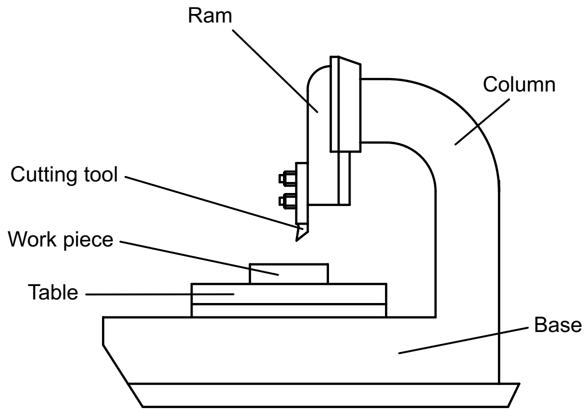


Fig. 6.11 *Schematic of a typical slotting machine*

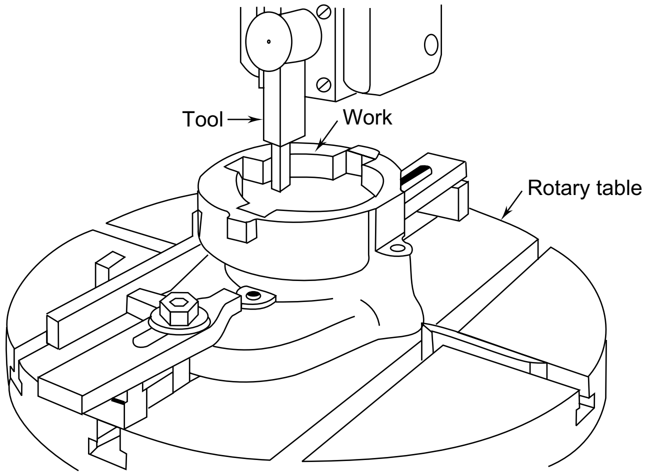


Fig. 6.12 *Typical component machined in a slotting machine*

6.5 COMPARISON OF RECIPROCATING MACHINE TOOLS

As can be seen there are a large number of similarities between the different reciprocating machine tools in the operation, while there are a number of significant differences which make them suitable for the type of applications for which they are used. Table 6.3 summarizes these differences.

Table 6.3 Comparison of shaper, planer and slotter

		Shaper	Planer	Slotter
1	Work-Tool motion	Tool reciprocates in horizontal axis and work feeds in intermittently	Work reciprocates in horizontal axis and tool feeds in intermittently	Tool reciprocates in vertical axis and work feeds in intermittently
2	Construction and rigidity	Lighter in construction and less rigid	Heavier in construction and more rigid	Lighter in construction and less rigid
3	Motor Power required	Relatively less power	Higher power compared to shaper	Relatively less power
4	Typical work size and setup time	Relatively small parts. Typical work envelope is: 450 × 450 × 600 mm. Quick setup time.	Bigger parts require lengthy setup time. Typical work envelope is: 3 × 3 × 15 m	Relatively small parts. Typical work envelope is: 450 × 450 × 300 mm. Quick setup time.
5	Number of surfaces that can be machined at a time	Only one surface at a time	Three surfaces can be machined at a time	Only one surface at a time
6	Material removal rate (MRR)	Low MRR	High since multiple tools can work at a time	Low MRR
7	Tool size	Regular size similar to lathe	Bigger size tools that can take higher depth of cut and feed	Regular size similar to lathe
8	Range of speeds and feeds	Smaller range and smaller number of speeds and feeds	Wide range and more number of speeds and feeds available	Smaller range and smaller number of speeds and feeds

SUMMARY

Unlike other machines tools, this class of machine tools do not have any rotating spindle, and the material gets removed by the reciprocating motion of a single point cutting tool. There are three types of machine tools that are normally found in this category.

- Shaper is by far the simplest type of machine tool in this category.
- Mechanical shapers are the simplest and low cost machine tools used for small size parts for generating flat surfaces.
- Hydraulic shapers offer better performance.
- Planers are with reciprocating table rather than the tool. As a result large size parts can be machined.
- Slotters are used to cut small grooves generally along the vertical axis similar to the keyways in shafts.

Questions

6.1 Explain with a neat sketch the operation and need for a clapper box in a mechanical shaper.
6.2 Give a schematic sketch of a shaper labelling important parts and their functions.
6.3 What are the applications of shaping machines in a typical machine shop?
6.4 Explain the following principal parts of a mechanical shaper.
  (a) Ram
  (b) Tool post
  (c) Quick return motion
6.5 Give the details of different types of shapers and their applications.
6.6 Give the various details that need to be specified for a shaping machine.
6.7 Describe the operation of the quick return motion in a mechanical shaper.
6.8 How are the tools held in a shaper?
6.9 Describe the methods of holding the work pieces in shapers. Give simple sketches of the same.
6.10 Give a neat sketch of the mechanical feed drive of a horizontal shaper and explain its function.
6.11 Describe with a schematic sketch the operation of a hydraulic shaper.
6.12 Give the advantages of Hydraulic shaper compared to mechanical (crank) shaper.
6.13 Compare the shaper and planer in terms of the operation, and type of work pieces.
6.14 What are the different planing machines used in industrial shops? Explain double housing planer and plate planer (edge-planing machine).
6.15 What is a planing machine? Explain the principal parts of a planing machine with neat sketches.
6.16 What type of work holding is normally used in planing machines?
6.17 Give the various details that need to be specified for a planing machine.
6.18 Give similarities and differences among shaping and planing machines with respect to constructed features, applications and working.
6.19 Write a small note on slotting machines.
6.20 Give a neat sketch of a slotter and describe its main parts.

6.21 Give similarities and differences among shaping, planning and slotting machines with respect to constructed features, applications and working.

6.22 Give similarities and differences among shaping, and slotting machines with respect to constructed features, applications and working.

6.23 What is the main difference between a vertical shaper and slotter?

Problems

6.1 Calculate the RPM of the bull gear of a mechanical shaper if the cutting speed is 35 m/min with the stroke length adjusted to 250 mm. Assume the ratio of cutting stroke to idle stroke as 1.5.

[233.333 RPM]

6.2 Calculate the machining time required for shaping a flat surface of 250 mm long and 200 mm wide on a hydraulic shaper using a cutting speed of 40 m/min and feed of 0.5 mm per stroke. The depth of cut is 4 mm. Calculate the material removal rate and the power required.

[80 000 mm^3/stroke, 4.88 HP]

6.3 A hydraulic shaper is used for shaping a plane surface of 30 × 250 mm with a cutting speed of 60 m/min and feed of 0.6 mm per stroke. Calculate the machining time and material removal rate if the depth of cut is 3 mm. Also calculate the power consumed in the process.

[0.306 minutes, 108 000 mm^3/stroke, 6.588 HP]

6.4 The two faces of a 90° V-block with dimensions of 50 mm each and width 150 mm is to be cut on a mechanical shaper. Calculate the actual machining time required. Make judicious assumptions if required and justify them. [145.833 minutes]

6.5 A 60° male dovetail with a dimension of 100 mm and width 50 mm is to be cut on a mechanical shaper. Calculate the machining time required. Make judicious assumptions if required.

[100 minutes]

6.6 A part measuring 300 mm × 100 mm × 40 mm is to be machined using a hydraulic shaper along its wide face (300 mm × 100 mm). Calculate the machining time taking approach as well as over travel as 25 mm each. Take cutting speed as 15 m/min, and a machining allowance on either side of plate width is 5 mm and feed is 0.4 mm/stroke. [12.83 minutes]

6.7 The flat surface of a large cast iron part measuring 1 m × 0.5 m × 300 mm is to be machined using a planer along its face (1 m × 0.5 m). Take cutting speed as 20 m/min, return speed as 40 m/min and feed is 1 mm/stroke. Neglect the over travel and approach distances in calculating the planing time. Assume the time for reversing the table is 0.02 minutes. [47.5 minutes]

6.8 Estimate the time required to machine a grey cast iron part measuring 5 m long × 1.50 m wide on a double housing planer. Cutting tools used are made of carbide which can take a cutting speed of 85 m/min and return speed is 170 m/min. To finish the part it requires two rough cuts with a feed of 1.5 mm/stroke and two finish cuts with a feed of 0.5 mm/stroke. Assume the time for reversing the table is 0.02 minutes. [14.43 hours]

6.9 A casting made from a class 30 gray iron is to be machined on a planer with a high speed cutting tool. The feed rate to be used is 0.4 mm/stroke for roughing and 0.15 mm/stroke for finishing. Cutting speed to be used is 20 m/min and while return speed is 40 m/min. Estimate the machining time if the surface area to be machined is 2 m long × 1.25 m wide. For finishing two roughing cuts and one finishing cut is required. Assume the time for reversing the table is 0.02 minutes. [64.93 hours]

6.10 A shaper is operated at 2 cutting strokes per second and is used to machine a work piece of 150 mm length at a cutting speed of 0.5 m/s using a feed of 0.4 mm per stroke and a depth of cut of 6 mm.
(a) Calculate the total machining time to produce 800 components each 100 mm in width.
(b) If the forward stroke is over 230°, calculate the percentage of the time when the tool is not contacting the work piece. State any assumptions made.

[27.78 hours, 36.11%]

Multiple Choice Questions

6.1 Which of the following machine tools has only linear motion for the cutting tool and the work piece for generating any type of surface?
(a) Lathe machine
(b) Shaper
(c) Milling machine
(d) Grinding machine

6.2 In which of the following machines, the work piece reciprocates while the cutting tool remains stationary during cutting?
(a) Planer
(b) Shaper
(c) Milling machine
(d) Grinding machine

6.3 For generating flat surfaces with large material removal rates, the following process is the most economical
(a) Facing
(b) Shaping
(c) End milling
(d) Surface Grinding

6.4 One of the advantages of a hydraulic shaper compared to the mechanical shaper is
(a) Variable cutting speed during the cutting stroke
(b) Cutting speed remains constant throughout the cuttings stroke
(c) Higher cutting speed at the start
(d) Higher cutting speed in the end

6.5 One of the disadvantages of a hydraulic shaper compared to the mechanical shaper is
(a) Stopping point of the cutting stroke can vary depending upon the resistance offered to cutting
(b) Less strokes per minute
(c) Power available varies during the cutting stroke
(d) Cutting speed remains constant throughout the cuttings stroke

6.6 The cutting tool moves in a vertical reciprocating motion in this machine tool
(a) Shaper
(b) Slotter
(c) Planer
(d) Vertical Lathe

6.7 In a planer the reciprocating motion of the table is obtained by
(a) Slider crank and connecting rod mechanism
(b) Cams and trip dogs
(c) Rack and pinion with associated belt drives and trip dogs
(d) None of the above

6.8 In a shaper the reciprocating motion of the table is obtained by
(a) Slider crank and connecting rod mechanism
(b) Cams and trip dogs
(c) Rack and pinion with associated belt drives and trip dogs
(d) None of the above

6.9 The best machine tool to cut an internal spine in steel is
(a) Milling machine
(b) Slotting machine
(c) Lathe
(d) None of the above

6.10 In which of the following machines the cutting tool reciprocates while the work piece remains stationary during cutting?

(a) Lathe machine
(b) Shaper
(c) Milling machine
(d) Grinding machine

6.11 The best machine tool to cut T-slots on a large milling machine table is
(a) Slotter
(b) Shaper
(c) Planer
(d) Milling machine

6.12 More than one tool head can be used in which of the following machine tools?
(a) Slotter
(b) Shaper
(c) Planer
(d) Milling machine

6.13 The standard work table in the following machine tool can have rotary motion in addition to the translatory motion
(a) Slotter
(b) Shaper
(c) Planer
(d) Milling machine

6.14 The machine tool that has the cutting stroke (removes material) when the ram moves towards the body is
(a) Universal shaper
(b) Draw cut Shaper
(c) Mechanical shaper
(d) Vertical shaper

Answers to MCQs

6.1 (b)	6.2 (a)	6.3 (b)	6.4 (b)	6.5 (a)
6.6 (b)	6.7 (c)	6.8 (a)	6.9 (b)	6.10 (b)
6.11 (c)	6.12 (c)	6.13 (a)	6.14 (b)	

CHAPTER 7

Milling

Objectives

Milling is a versatile machine tool that uses a rotating milling cutter while the work piece reciprocates in contact with the cutting tool. After completing this chapter, the reader will be able to

- Understand the characteristic features of the milling process
- Know the various types of milling machines and their applications
- Select the various types of milling cutters based on the application
- Understand the different types of milling operations that can be done in the milling machines
- Select the type of work holding device to be used for a given part surface
- Understand different types of indexing possibilities for parts
- Understand the mechanics of milling operations to appreciate the effect of different process variables on the performance
- Calculate the machining time and power based on the material properties
- Setup special operations in the milling machine for specific types of surfaces

7.1 INTRODUCTION

After the class of lathes, milling machines are the most widely used machine tools for manufacturing applications. In milling, the work piece is fed into a rotating milling cutter, which is a multi-point tool as shown schematically in Fig. 7.1. It is unlike a lathe, which uses a single point cutting tool. The tool used in milling is called milling cutter.

The milling process is characterised by:

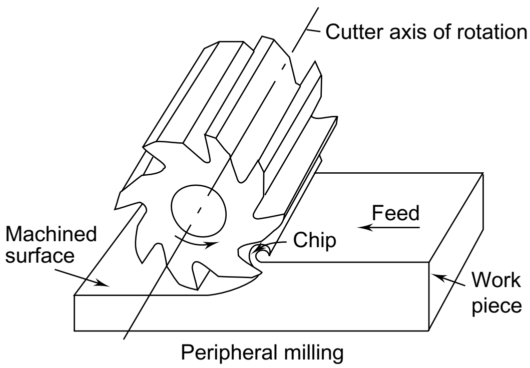


Fig. 7.1 *Schematic of milling operation*

Interrupted cutting Each of the cutting edges removes material for only part of the rotation of the milling cutter. As a result, the cutting edge has time to cool before it removes material again. Thus the milling operation is much cooler compared to the turning operations seen earlier. This allows for much larger material rates.

Small size of chips Though the size of the chips is small, in view of the multiple cutting edges in contact, a large amount of material is removed and as a result the component is generally completed in a single pass unlike the turning process which requires a large number of cuts for finishing.

Variation in chip thickness This contributes to the non-steady state cyclic conditions of varying cutting forces during the contact of the cutting edge with the chip thickness varying from zero to maximum size or vice versa. This cyclic variation of the force can excite any of the natural frequencies of the machine tool system and would be harmful to the tool life and surface finish generated.

This is one of the most versatile machine tools. It is adaptable for quantity production as well as in job shops and tool rooms. Versatility of milling is because of the large variety of accessories and tools available with the milling machines. Typical tolerance expected from the process is about ± 0.050 mm.

7.2 TYPES OF MILLING MACHINES

To satisfy the variety of requirements as mentioned above, milling machines come in a number of combinations , sizes and varieties. In view of the large material removal rates the milling machines come with a very rigid spindle and large power. The varieties of milling machines available are:

(a) Knee and Column type
- Horizontal
- Vertical
- Universal
- Turret type

These are the general purpose milling machines, which have a high degree of flexibility and employed for all types of works including batch manufacturing. A large variety of attachments to improve the flexibility are available for this class of milling machines.

(b) Production (Bed) type
- Simplex
- Duplex
- Triplex

These machines are generally meant for regular production involving large batch sizes. The flexibility is relatively less in these machines that suits productivity enhancement.

(c) Plano millers

These machines are used only for very large work pieces involving table travels in meters.

(d) Special type
- Rotary table
- Drum type
- Copy milling (Die sinking machines)
- Key way milling machines
- Spline shaft milling machines

These machines are special class to provide special facilities to suit specific applications that are not catered by the other classes of milling machines.

7.2.1 Knee and Column Milling Machine

The knee and column type is the most commonly used machine in view of its flexibility and easier setup. The typical machine construction is shown schematically in Fig. 7.2 for the horizontal axis. The knee houses the

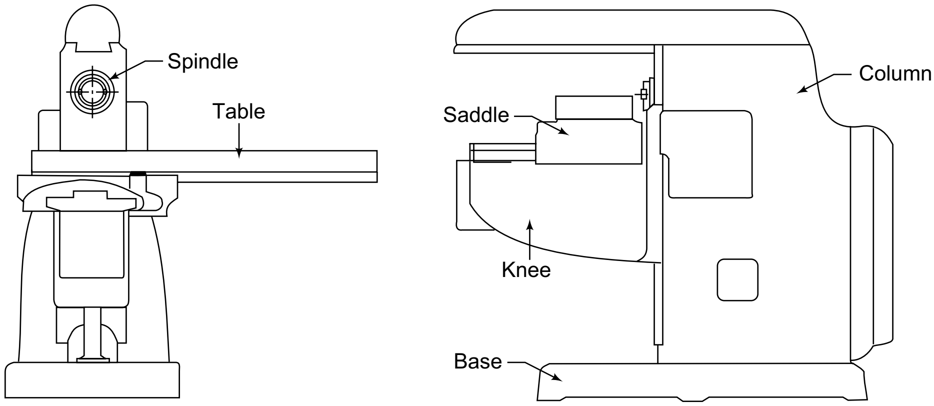


Fig. 7.2 *Horizontal Knee and column type milling machine*

feed mechanism and mounts the saddle and table. The table basically has the T-slots running along the X-axis for the purpose of work holding. Table moves along the X-axis on the saddle while the saddle moves along the Y-axis on the guide ways provided on the knee. The feed is provided either manually with a hand wheel or connected for automatic feed by the lead screw, which in turn is coupled to the main spindle drive. The knee can move up and down (Z-axis) on a dovetail provided on the column.

The massive column at the back of the machine houses all the power train, including the motor and the spindle gearbox. The power for feeding the table lead screw is taken from the main motor through a separate feed gear box. Sometimes it is possible that a separate feed motor is provided for the feed gearbox as well.

While the longitudinal and traverse motions are provided automatically, the raising of the knee is generally made manually.

The spindle is located at the top end of the column. Arbour used to mount the milling cutters is mounted in the spindle and is provided with a support on the other end to take care of the heavy cutting forces by means of an over-arm with bearing. As shown in Fig. 7.2 the over arm extends from the column with a rigid design. The spindle nose has the standard Morse taper, the size of which depends upon the machine size.

The milling cutters are mounted on the arbour at any desired position, the rest of the length being filled by standard hardened collars of varying widths to fix the position of the cutter. The arbour is clamped in the spindle with the help of a draw bar and then fixed with nuts.

Milling machines are generally specified based on the following:

- Size of the table, which specifies the actual working area on the table and relates to the maximum size of the work piece that can be accommodated.
- Amount of table travel, which gives the maximum axis movement that is possible.
- Horse power of the spindle, which actually specifies the power of the spindle motor used. Smaller machines may come with 1 to 3 hp while the production machines may vary from 10 to 50 hp.

Another type of knee and column milling machine is the vertical axis type (Fig. 7.3). Most of the construction is very similar to the horizontal axis type except the spindle type and location. The spindle is located in the vertical direction and is suitable for using the shank mounted milling cutters such as end mills. In view of the location of the tool, the setting up of the work piece and observing the machining operation is more convenient.

The vertical axis milling machine is more flexible (Fig. 7.4) and suitable for machining complex cavities such as in die cavities in tool rooms. Also the vertical head is provided with swivelling facility in horizontal direction whereby the cutter axis can be swivelled. This is useful for tool rooms where more complex milling operations are carried out.

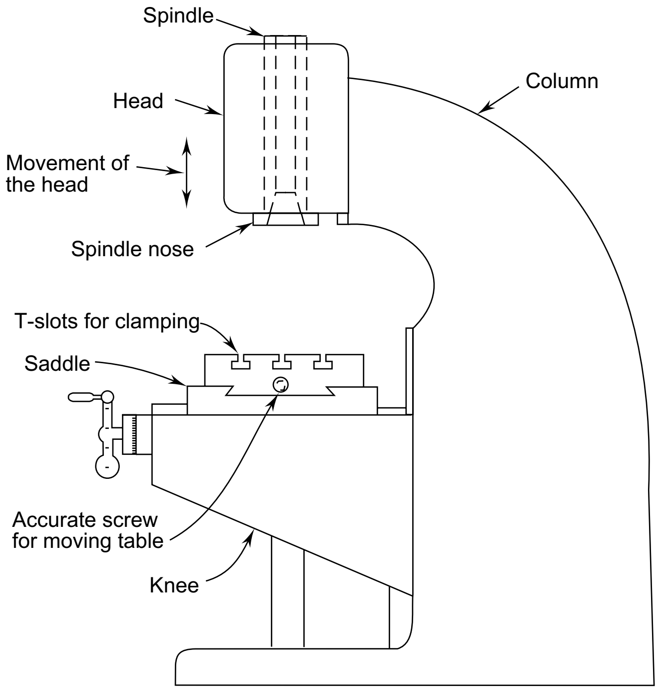


Fig. 7.3 *Vertical Knee and column type milling machine*

The universal machine has the table, which can be swivelled in horizontal plane about 45° to either the left or right. This makes the universal machine suitable for milling spur and helical gears, as well as worm gears and cams.

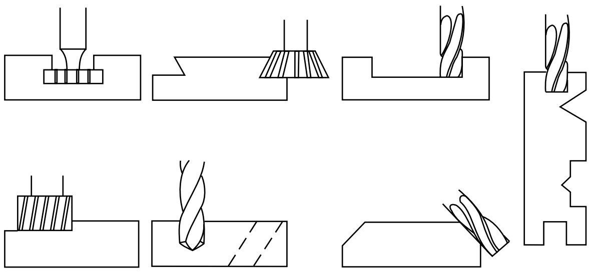


Fig. 7.4 *Some of the Milling Operations normally carried out on vertical axis machines*

7.2.2 Bed Type Milling Machine

In production milling machines it is desirable to increase the metal removal rates. If it is done on conventional machines by increasing the depth of cut, chatter is likely to result. Hence another variety of milling machines named as the bed type machines, which are made more rugged and consequently are capable of removing more material. The ruggedness is obtained as a consequence of the reduction in the versatility. The table in the case of bed type machines is directly mounted on the bed and is provided with only the longitudinal motion.

The spindle will be moving along with the column to provide the cutting action. Simplex machines (Fig. 7.5) are the ones with only one spindle head while the duplex machines have two spindles (Fig. 7.6). The two spindles are located on either side of a heavy work piece and normally remove material from both sides simultaneously.

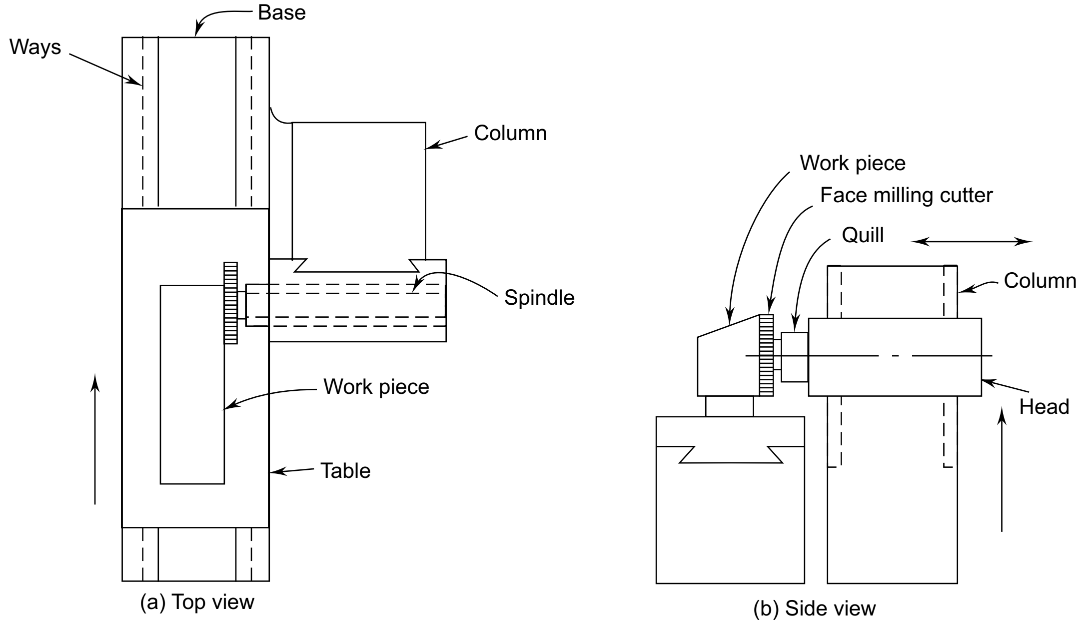


Fig. 7.5 *Simplex bed type milling machine*

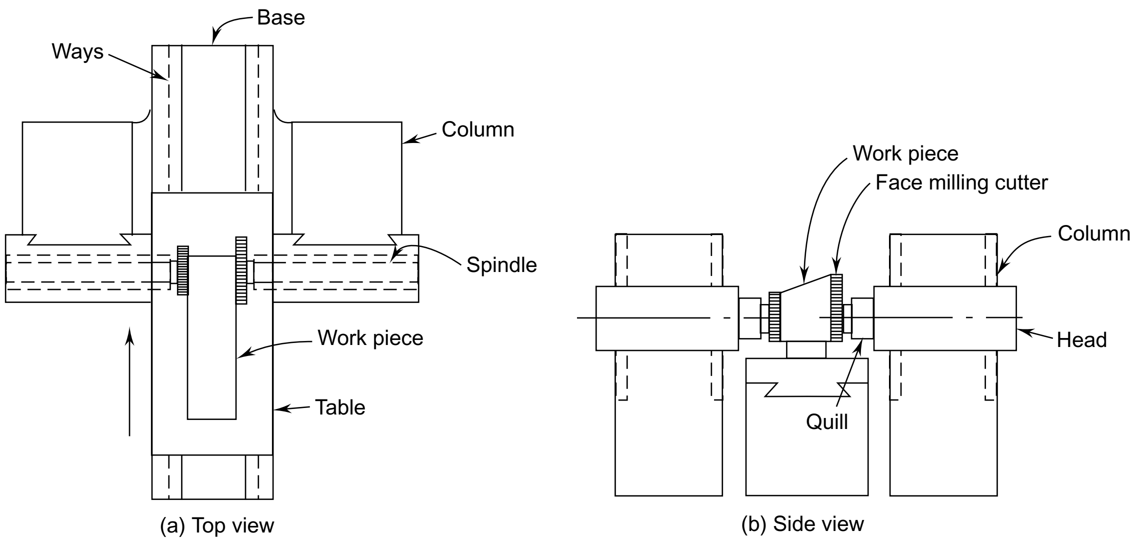


Fig. 7.6 *Duplex bed type milling machine*

7.3 MILLING CUTTERS

There are a large variety of milling cutters available to suit specific requirements. The versatility of the milling machine is contributed to a great extent by the variety of milling cutters that are available.

7.3.1 Types

Milling cutters are classified into various types based on a variety of methods.

Based on construction

- Solid
- Inserted tooth type

Based on mounting

- Arbour mounted
- Shank mounted
- Nose mounted

Base on rotation

- Right hand rotation (Counter clockwise)
- Left hand rotation (Clockwise)

Based on helix

- Right hand helix
- Left hand helix

Milling cutters are generally made of high speed steel or cemented carbides. The cemented carbide cutters can be with brazed tip variety or more commonly with indexable tips. The indexable variety is more common since it is normally less expensive to replace the worn out cutting edges than to regrind them.

Plain Milling Cutters

These are also called slab milling cutters and are basically cylindrical with the cutting teeth on the periphery as shown in Fig. 7.7(a). These are generally used for machining flat surfaces.

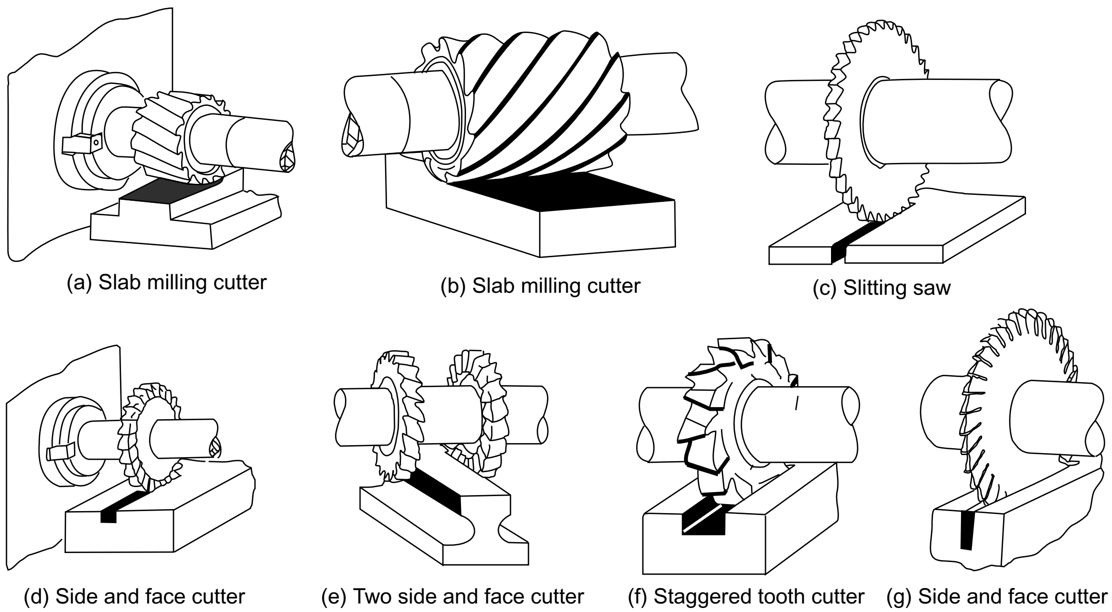


Fig. 7.7 *Arbor mounted milling cutters general purpose*

Light duty slab milling cutters generally have a face width, which is small of the order of 25 mm. They generally have straight teeth and large number of teeth.

The heavy duty slab milling cutters come with smaller number of teeth to allow for more chip space. This allows taking deeper cuts and consequently high material removal rates.

Helical milling cutters have a very small number of teeth but a large helix angle. This type of cutter cuts with a shearing action, which can produce a very fine finish. The large helix angle allows the cutter to absorb most of the end load and therefore the cutter enters and leaves the work piece very smoothly.

Side and Face Milling Cutters

These have the cutting edges not only on the face like the slab milling cutters, but also on both the sides. As a result, these cutters become more versatile since they can be used for side milling as well as for slot milling.

Staggered tooth side milling cutters are a variation where the teeth are arranged in alternate helix pattern. This type is generally used for milling deep slots, since the staggering of teeth provides greater chip space.

Another variation of the side and face cutter is the half side milling cutter, which has the cutting edge only on one side. This arrangement will allow a positive rake angle and will be useful for machining on only one side. These have much smoother cutting action and long tool life. The power consumed is also less for these cutters.

Slitting Saws

The other common form of milling cutters in the arbour mounted category is the slitting saw. This is very similar to a saw blade in appearance as well as function. Most of these have teeth around the circumference while some have side teeth as well. The thickness of these cutters is generally very small and is used for cutting off operation or for deep slots.

Special Form Cutters

In addition to the general type of milling cutters described above, there are a large number of special form milling cutters available, which are used for machining specific profiles.

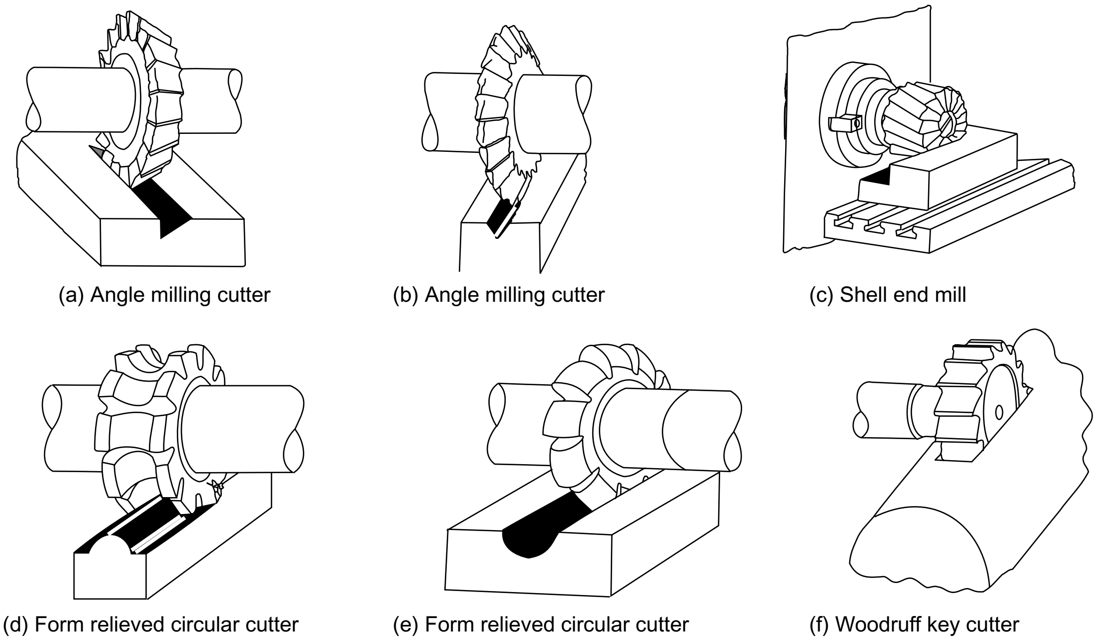


Fig. 7.8 *Arbor mounted milling cutters special forms*

Angular milling cutters are made in single or double angle cutters for milling any angle such as 30, 45 or 60°.

Form relieved cutters are made of various shapes such as circular, corner rounding, convex or concave shapes.

T-slot milling cutters are used for milling T-slots such as those in the milling machine table. The central slot is to be milled first using an end mill before using the T-slot milling cutter.

Woodruff key seat milling cutters as the name suggests are used for milling Woodruff key seats.

Some other special form cutters are:

- Dovetail milling cutters
- Gear milling cutters

End Mills

These are shank mounted as shown in Fig. 7.9 and are generally used in vertical axis milling machines. They are used for milling slots, key ways and pockets where other type of milling cutters cannot be used. A depth of cut of almost half the diameter can be taken with the end mills.

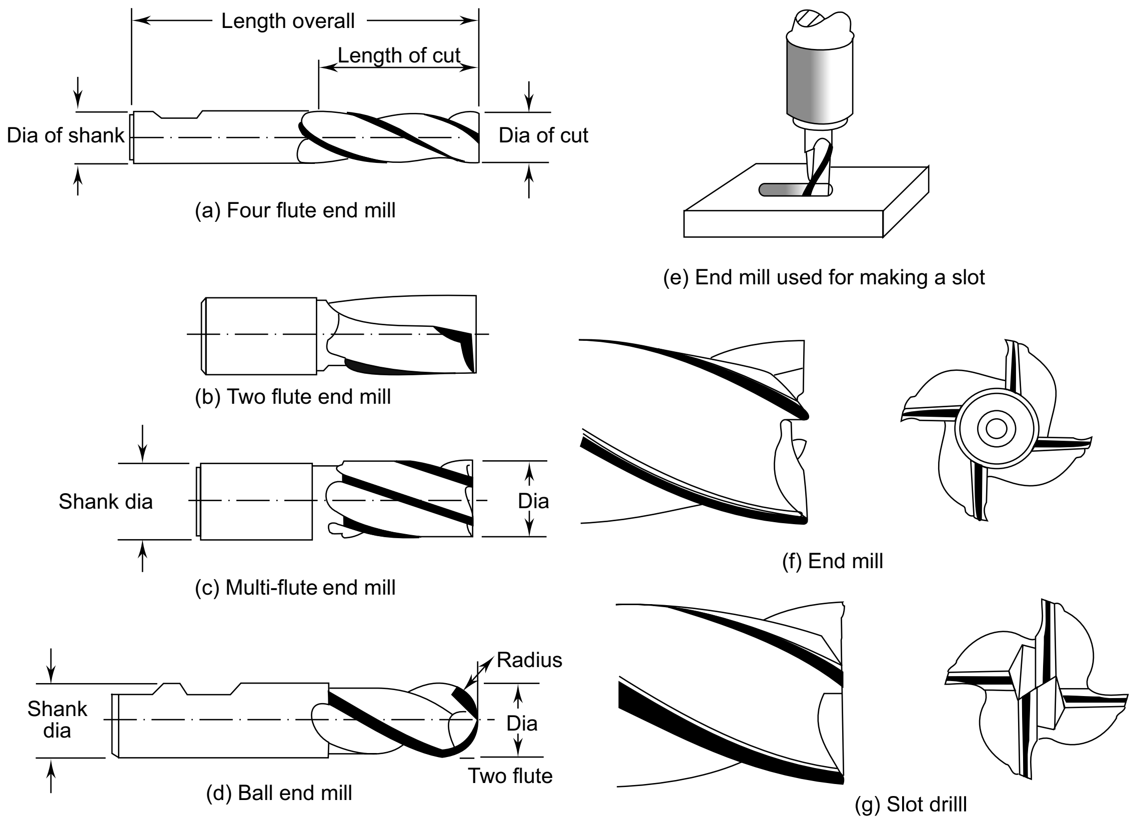


Fig. 7.9 *Shank mounted milling cutters, various types of end mills*

The end mills have the cutting edge running through the length of the cutting portion as well as on the face radially upto a certain length. The helix angle of the cutting edge promotes smooth and efficient cutting even at high cutting speeds and feed rates. High cutting speeds are generally recommended for this type of milling cutters.

There are a large variety of end mills. One of the distinctions is based on the method of holding, i.e. the end mill shank can be straight or tapered. The straight shank is used on end mills of small size and held in the milling machine spindle with the help of a suitable collet. The tapered shank can be directly mounted in the spindle with the help of the self holding taper. If the taper is small compared to the spindle taper, then an adopter accommodating both the tapers is used.

The end teeth of the end mills may be terminated at a distance from the cutter centre or may proceed till the centre. Those with the cutting edge upto the centre are called slot drills or end cutting end mills since they have the ability to cut into the solid material. The other type of end mills, which have a larger number of teeth, cannot cut into solid material and hence require a pilot hole drilled before a pocket is machined.

The cutting edge along the side of an end mill is generally straight and some times can be tapered by grinding on a tool and cutter grinder such that the draft required for mould and die cavities can be automatically generated. Further the end face can be square with the side as in the normal case or a ball end shape to be used for milling three dimensional contours such as in die cavities. It can also have a rounded corner for milling special round edged pockets as shown in Fig. 7.10.

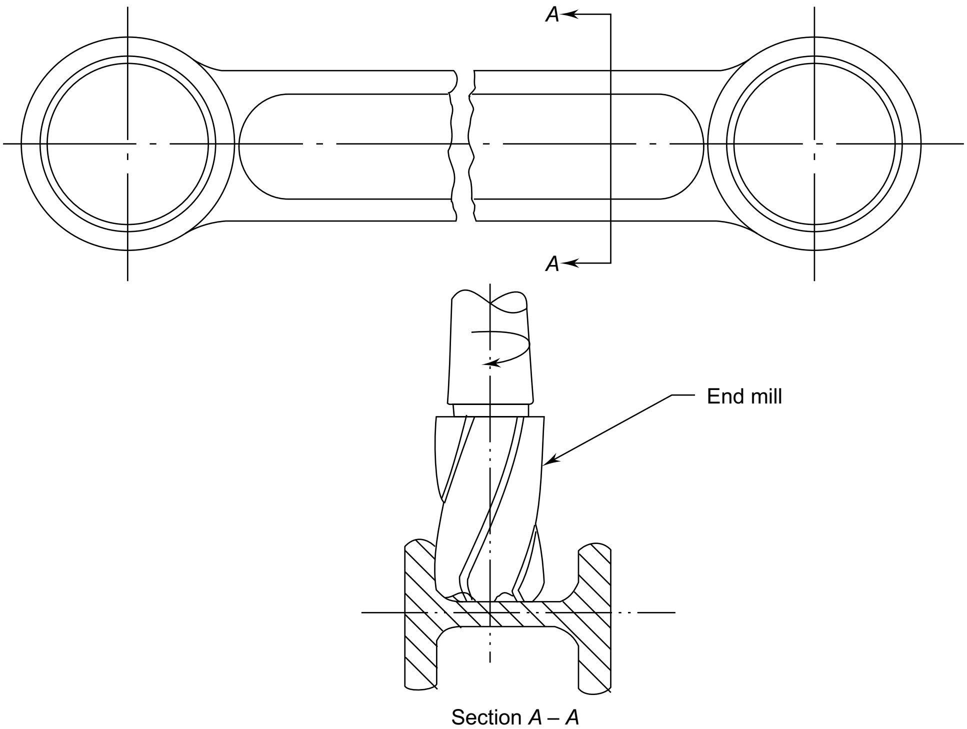


Fig. 7.10 *End milling using a corner radius*

Large size end mills are called shell end mills, which do not have any shank and can be mounted with the help of a central hole. Consequently these can be used in horizontal axis as well as vertical axis milling machines. These will be mounted with the help of a stub arbor on to the spindle as shown in Fig. 7.9.

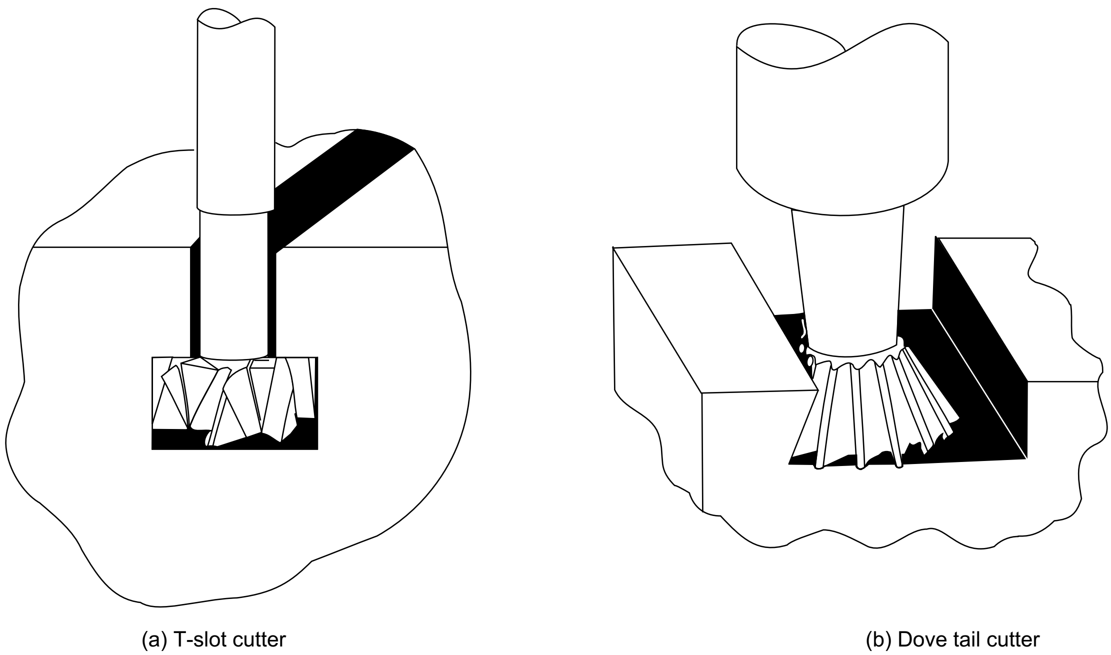


Fig. 7.11 *Special milling cutters for specific applications*

Face milling cutters (Fig. 7.12) are used for machining large, flat surfaces. They have the cutting edges on the face and periphery. It is generally mounted directly on the nose of the spindle with the entire face free for machining. The teeth on the face do most of the machining while those on the side are used for cleaning the surface. These are generally made of carbide insert variety in view of the large material removal involved, though high speed steel types are also used.

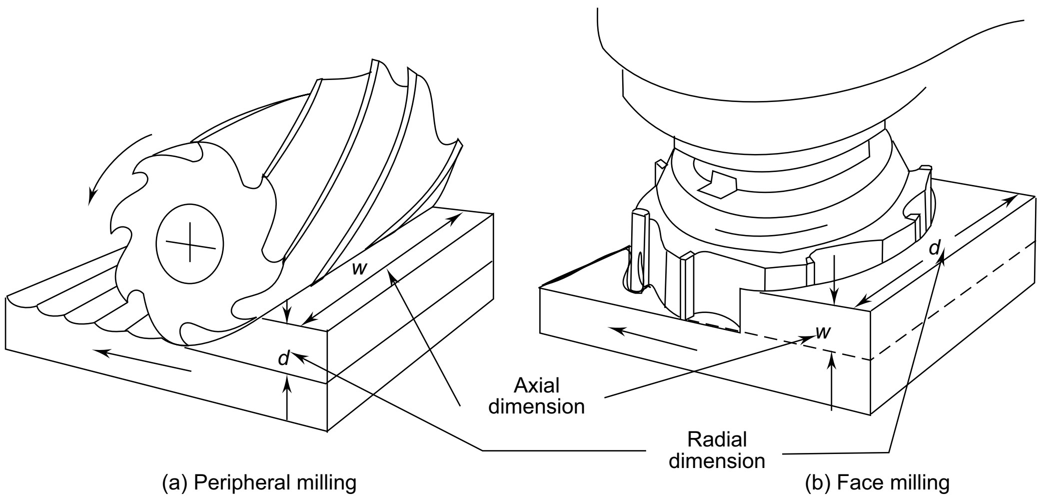


Fig. 7.12 *Generating plane surfaces in Milling using face milling*

In connection with the milling cutter designation the following terms are often used.

Hand of cut This refers to the direction in which the cutter is rotated. When viewed towards the spindle, if the cutter moves counter clockwise it is called right hand rotation while the opposite is called the left hand rotation.

Hand of helix In case of helical milling cutters, when viewed from the end if the flutes move in a clockwise direction it is called the right hand helix while the opposite is called the left hand helix. The axial cutting force direction depends upon the hand of the helix. If two milling cutters of different helices are arranged side by side in a gang milling operation, the net axial force can be reduced to zero depending upon the cut taken by each of the milling cutters.

While selecting a milling cutter for a given application the following points should be considered.

- Use standard tools whenever possible.
- Use a short overhang from the spindle and largest possible mounting type.
- Select a cutter diameter 30% larger than the width of cut (face milling).
- Use a close pitch cutter as first choice.
- Use a coarse pitch cutter for long overhangs and unstable conditions.
- Use an extra close pitch cutter for short chipping materials and small radial depths of cut.

The slab milling cutter can have cutting teeth, which can be straight and parallel to the axis of rotation. Alternatively they can be at an angle to the axis along a helix as shown in Fig. 7.13.

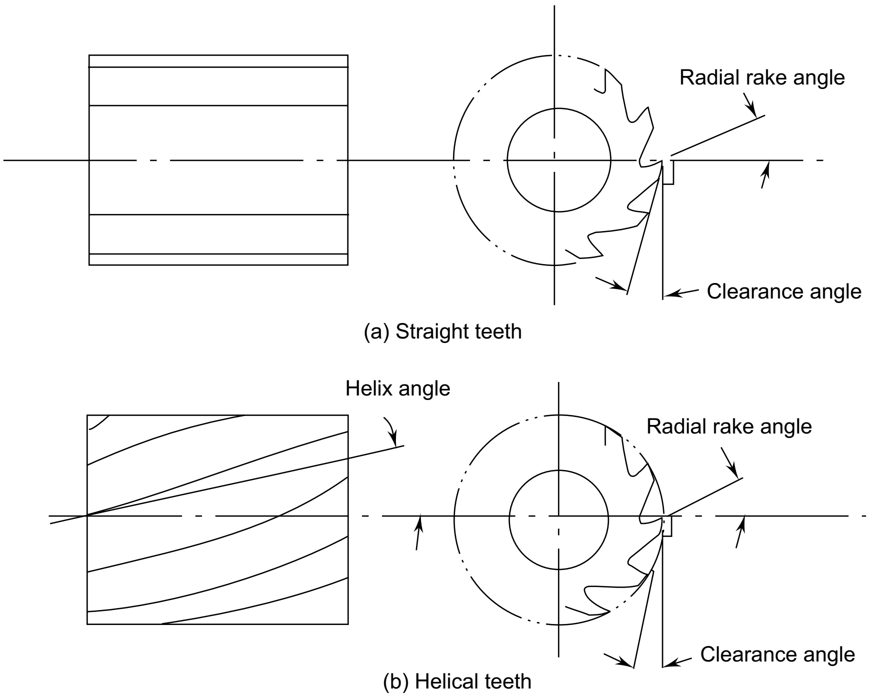


Fig. 7.13 *Slab milling cutter, straight and helical*

The straight teeth as shown in Fig. 7.13(a) will always have one tooth in contact with the work piece. When the cutting starts, the chip thickness is maximum, which gradually reduces to zero, before the next tooth comes into contact with the work piece. As a result, the cutting force rises to maximum value and then rapidly drops to zero before rising again as shown in Fig. 7.14(a) schematically. This force variation gives rise to impact loads on the milling cutter and may induce vibrations. In the case of the helical milling cutter each tooth is longer than the straight tooth. As a result, at any given time more than one tooth will be in contact with the work piece each having a different chip thickness. As a result, the cutting force variation will be steadier as shown in Fig. 7.14(b).

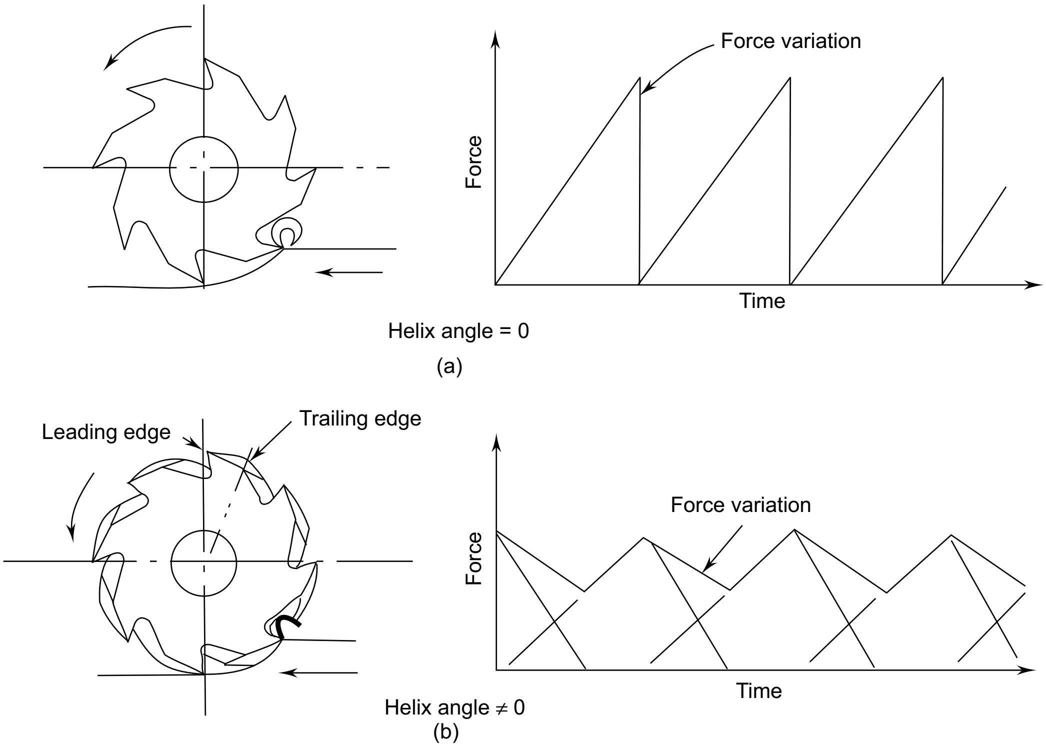


Fig. 7.14 *Slab milling cutter, effect of helix angle*

7.3.2 Up and Down milling

Based on the directions of movement of the milling cutter and the feeding direction of the work piece, there are two possible types of milling:

- Up milling (conventional milling)
- Down milling (Climb milling)

Up Milling

In up milling the cutting tool rotates in the opposite direction to the table movement. In the conventional or up milling, the chip starts at zero thickness and gradually increases to the maximum size as shown in Fig. 7.15(a). This tends to lift the work piece from the table. There is a possibility that the cutting tool will rub the work piece before starting the removal. However, this process is inherently safe.

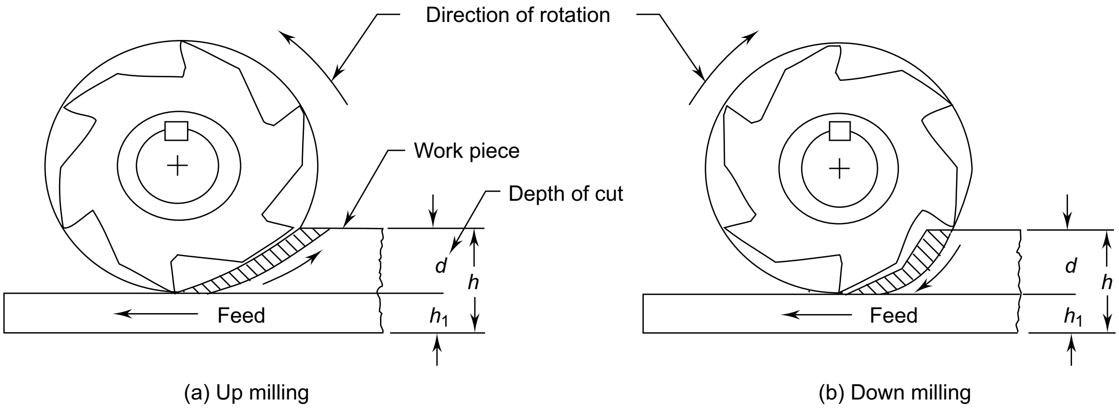


Fig. 7.15 *Up Milling and Down milling*

The initial rubbing of the cutting edge during the start of the cut in up milling tends to dull the cutting edge and consequently have lower tool life. Also since the cutter tends to cut and slide alternatively, the surface generated is left with the machining marks.

Down Milling (Climb Milling)

In down milling the cutting tool rotates in the same direction as that of the table movement. In the climb or down milling, the chip starts at maximum thickness and goes to zero thickness gradually as shown in Fig. 7.15(b). This is suitable for obtaining fine finish on the work piece. The cutting force will act downwards and as such would keep the work piece firmly in the work holding device. This is good for thin and frail work pieces.

In this case the cutting force direction as well as the lead screw motion being in the same direction, there is a possibility that the backlash present in the table lead screw will interfere with the actual motion of the table by making it jerky. Sometimes it is possible that the work may be pulled into the cutter, which may result in a broken milling cutter or damaged work piece. This may some times be dangerous to the machine tool as well. Also the chip starts with maximum thickness and this gives a large force, which will have to be taken care by rigid lead screw for table feeding.

In down milling, though the cut starts with a full chip thickness, the cut gradually reduces to zero. This helps in eliminating the feed marks present in the case of up milling and consequently better surface finish. Climb milling also allows greater feeds per tooth and longer cutting life between regrinds than the conventional milling.

Advantages

1. Suited for machine thin and hard-to-hold parts since the work piece is forced against the table or holding device by the cutter.
2. Work need not be clamped as tightly.
3. Consistent parallelism and size may be maintained, particularly on thin parts.
4. It may be used where breakout at the edge of the work piece could not be tolerated.
5. It requires upto 20% less power to cut by this method.
6. It may be used when cutting off stock or when milling deep, thin slots.

Disadvantages

1. It cannot be used unless the machine has a backlash eliminator and the table jibs have been tightened.
2. It cannot be used for machining castings or hot rolled steel, since the hard outer scale will damage the cutter.

7.4 MILLING OPERATIONS

7.4.1 Work Holding

Milling machine table comes with precision parallel T-slots along the longitudinal axis. The work piece therefore can be mounted directly on the table using these T-slots. Alternatively a variety of work holding devices can be used for holding the work piece, depending upon the type of work piece and the type of milling to be done.

Vice is the most common form of work holding device used for holding small and regular work pieces. The vice is mounted on the table using the T-slots. A variety of vice jaws are available to suit different work piece geometries.

Universal chuck is used for holding round work pieces for machining of end slots, splines, etc.

Fixtures are the most common form of work holding devices used in production milling operations. These become almost a necessity to reduce the setup time and increase the locational accuracy and repeatability.

Clamps For large and irregular work pieces, clamps in a variety of shapes are available as shown in Fig. 7.16. These clamps can be applied in a variety of ways. Some of the methods are shown in Fig. 7.17. However, care has to be taken while the clamps are used for work holding. The work piece should not be shifted under the action of the cutting forces. Also the clamping force should not be too high such that the distortion of work piece takes place.

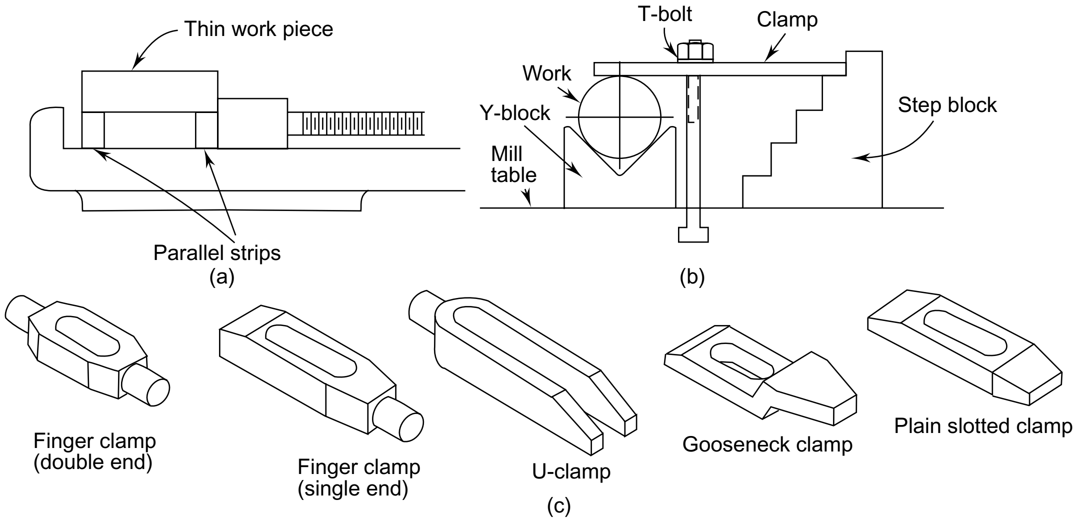


Fig. 7.16 *Common work holding methods in Milling*

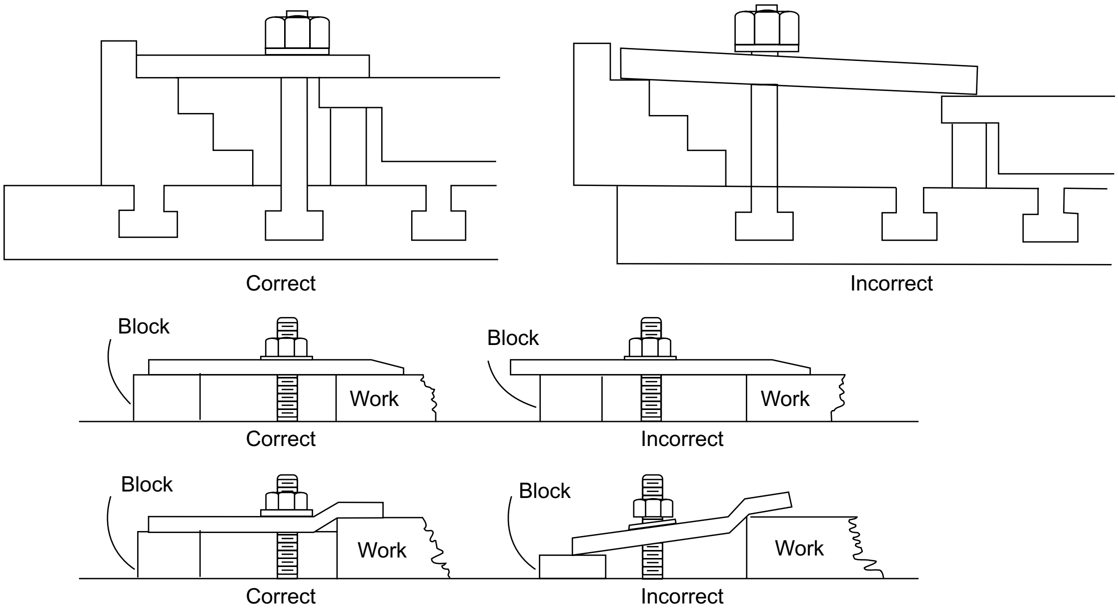


Fig. 7.17 *Work holding principles in Milling*

7.4.2 Milling Setups

Depending upon the situation, a large variety of milling methods have been developed in view of its versatility. These methods are more of a convention followed on the shop floor, rather than following any other scientific basis. The methods are-

String Milling

In this method small work pieces which are to be milled are fed into the milling cutter one after the other. In other words a number of the work pieces will be kept on the machine table in a line, hence are called 'string milling' or 'line milling'. The main advantage is that if individual work pieces are milled, the milling cutter will have to keep the approach distance, which is substantial. By having a number of work pieces kept in line the approach distance will be only at the beginning and end of the line, thus considerably saving the machine time.

Abreast or Reciprocal Milling

This is an operation done with special milling fixtures, which have a capability for indexing 180°. While one component is being machined at position 1, at the second position, which is at 180° to the first one, the second component will be loaded. When the machining is completed at location one, the fixtures indexes bring the already clamped component ready for machining. In this case the machine need not remain idle during the unloading, loading and setup of the blank for machining.

Rotary or Circular Milling

Rotary milling takes the reciprocal milling to a greater length. A number of fixtures depending upon their size are located on a rotary table such that a number of work pieces can be loaded simultaneously on the machine table. This will save the setting time of the work pieces and keeps the machine cutting all the time except during the indexing of the rotary table. The rotary can be integral with the milling machine or a separate accessory fixed to the milling machine table.

Gang Milling

In gang milling a number of milling cutters are fastened to the arbor to suit the profile of the work piece to be machined. For example, two side and face milling cutter with a slab milling cutter at the centre, to mill an inverted U-shape. The advantage of gang milling is that several surfaces are machined at the same time. It is also possible to combine form cutters along with the general purpose cutters.

One of the major problems is the choice of the cutting speed, which is determined by the largest cutter diameter. Hence it is desirable that all the cutters should be similar in size and shape to allow for larger speeds and feeds. In production milling operations, gang milling is generally preferred.

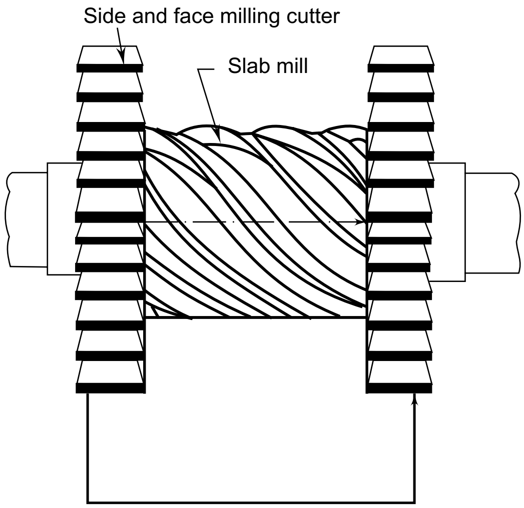


Fig. 7.18 *Gang milling*

Straddle Milling

Straddle is a special form of gang milling where only side and face milling cutters are used.

A typical sequence of processes used and the milling cutters required for a component machined in a milling machine is shown in Fig. 7.19.

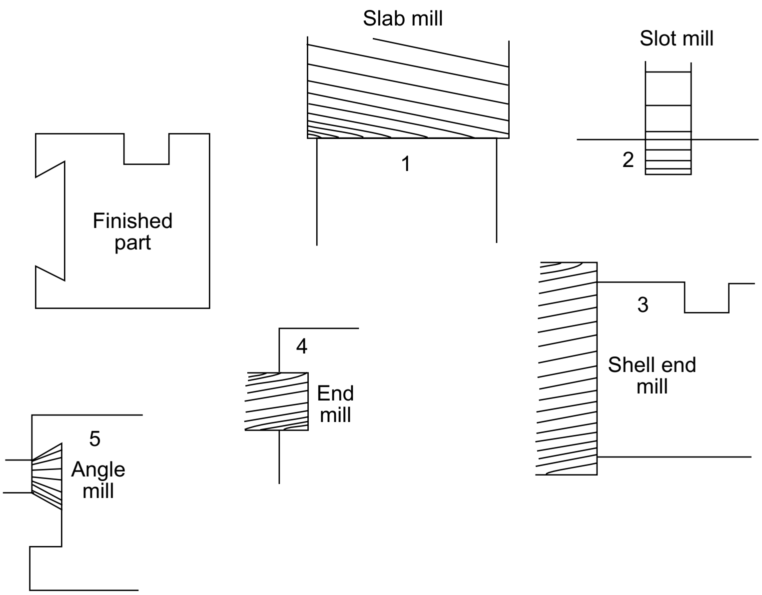


Fig. 7.19 *Typical process sequence in milling*

7.5 DIVIDING HEAD

Dividing head is one of the most important attachments with the milling machine and it is almost indispensable. The typical construction of the dividing head is shown in Fig. 7.20. The main spindle of the dividing head drives the work piece by means of a 3-jaw universal chuck or a dog and live centre similar to a lathe.

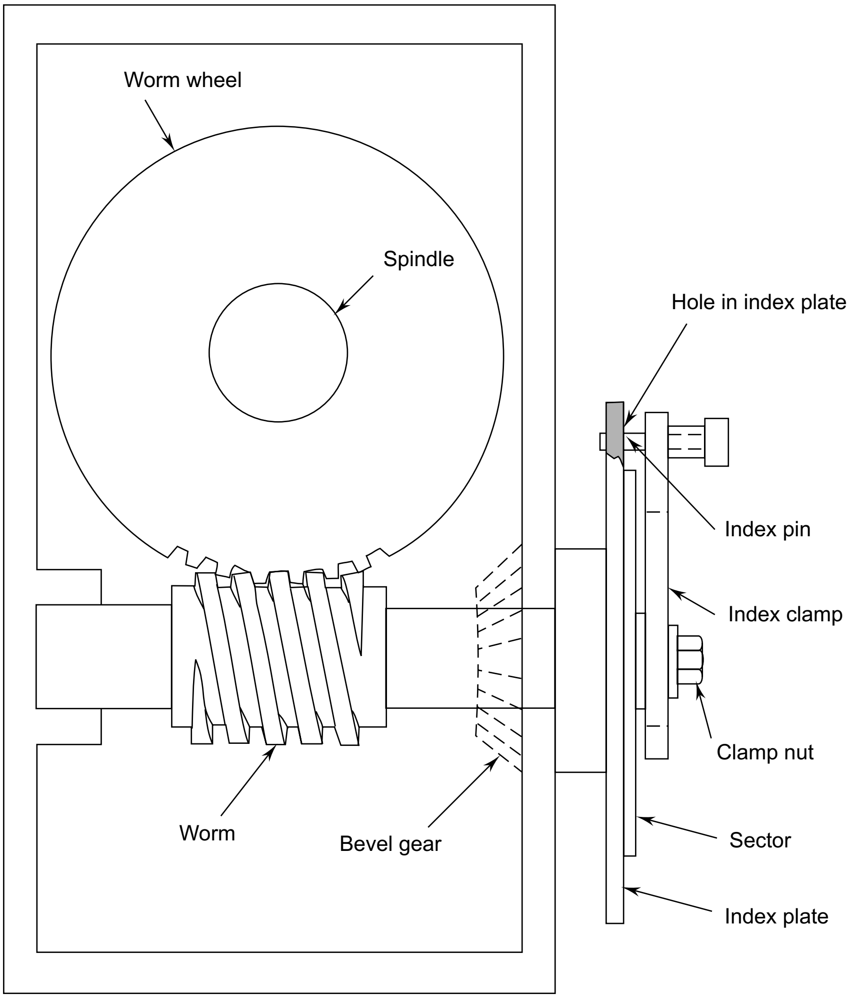


Fig. 7.20 *Dividing head construction*

The index plate of a dividing head consists of a number of holes with a crank and pin. The index crank drives the spindle and the live centre through a worm gear, which generally has 40 teeth as shown in Fig. 7.21. As a result, a full rotation of the work piece is produced by 40 full revolutions of the index crank. Further indexing is made possible by having the index plates with equi-spaced holes around various circles. This would allow for indexing the periphery of the work piece to any convenient number of divisions.

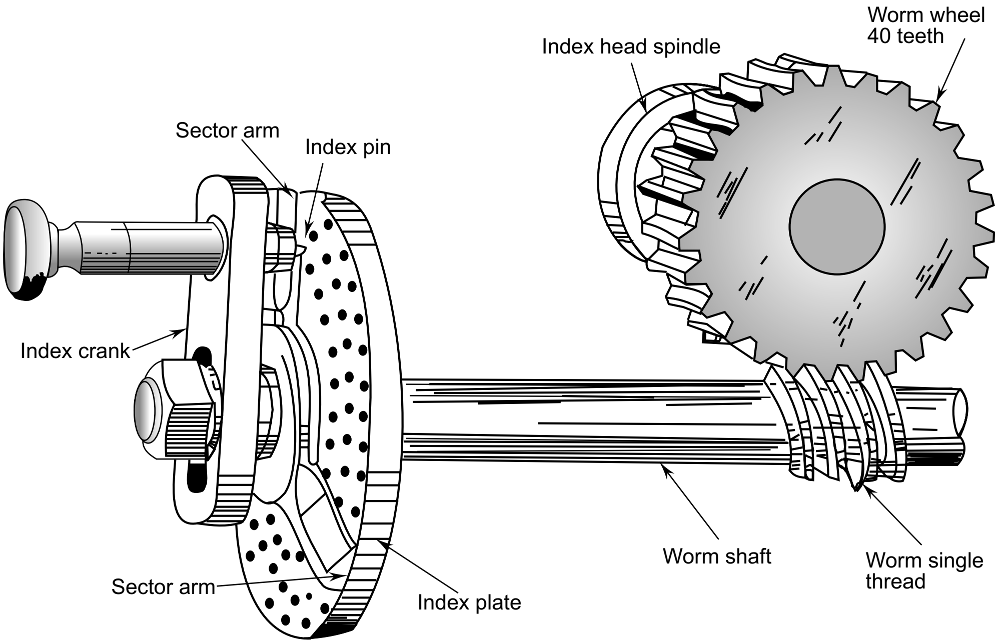


Fig. 7.21 *Indexing method of the Dividing head*

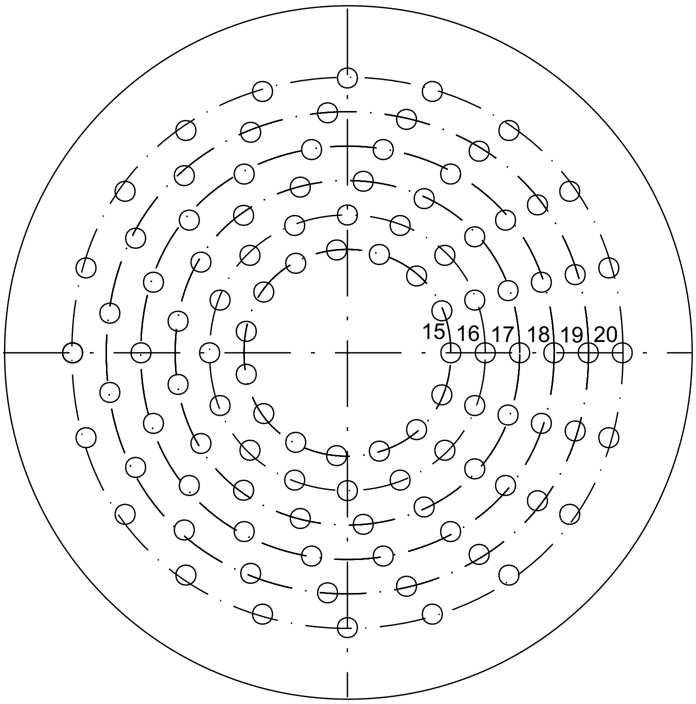


Fig. 7.22 *Index plate no. 1 of Brown and Sharpe Dividing head*

The index plates available with the Brown and Sharpe milling machines are

Plate no. 1: 15, 16, 17, 18, 19, 20 holes
Plate no. 2: 21, 23, 27, 29, 31, 33 holes
Plate no. 3: 37, 39, 41, 43, 47, 49 holes

The index plate used on Cincinnati and Parkinson dividing heads is

Plate 1: Side 1 24, 25, 28, 30, 34, 37, 38, 39, 41, 42 and 43 holes
Side 2 46, 47, 49, 51, 53, 57, 58, 59, 62 and 66 holes

It is also possible to get additional plates from Cincinnati to increase the indexing capability as follows:

Plate 2: Side 1 34, 46, 79, 93, 109, 123, 139, 153, 167, 181, 197 holes
Side 2 32, 44, 77, 89, 107, 121, 137, 151, 163, 179, 193 holes

Plate 3: Side 1 26, 42, 73, 87, 103, 119, 133, 149, 161, 175, 191 holes
Side 2 28, 38, 71, 83, 101, 113, 131, 143, 159, 173, 187 holes

7.5.1 Simple or Plain Indexing

Plain indexing is the name given to the indexing method carried out using any of the indexing plates in conjunction with the worm. With this it is possible to obtain relatively simple divisions. To explain the procedure let us consider a gear that is to be milled with 20 teeth. This means that the gear blank held in the spindle of the dividing head is to be divided equally into 20 divisions. Since 40 revolutions of the index crank produces one full revolution of the work piece, we need to rotate the index crank two full turns for cutting each tooth of the gear.

Suppose we want to have 6 equal divisions to be made.

The rotation of the index crank = $\frac{40}{6} = 6\frac{2}{3}$ turns.

This means that the index crank should be rotated 6 full turns followed by two-thirds of a rotation. The fraction of a rotation required is to be obtained with the help of the index plates as given above. This can be done as follows, using any of the Brown & Sharpe plates.

Plate no. 1: 10 holes in 15-hole circle
12 holes in 18-hole circle

Plate no. 2: 14 holes in 21-hole circle
18 holes in 27-hole circle
22 holes in 33-hole circle

Plate no. 3: 26 holes in 39-hole circle

Example 7.1

Indexing 28 divisions.

Solution The rotation of the index crank = $\frac{40}{28} = 1\frac{3}{7}$ turns.

This can be done as follows using any of the Brown & Sharpe plates

One full rotation + 9 holes in a 21-hole circle in plate no. 2.
One full rotation + 21 holes in a 49-hole circle in plate no. 3.

Example 7.2

Indexing 62 divisions.

Solution The rotation of the index crank = $\frac{40}{62} = \frac{20}{31}$ turns.

This can be done as follows using any of the Brown & Sharpe plates

20 holes in a 31-hole circle in plate no. 2.

7.5.2 Compound Indexing

Using the above method a majority of the indexing jobs could be completed. However when the available capacity of the index plates is not sufficient to do a given indexing job, the compound indexing method could be used. In order to obtain more complex indexing the following method is used. First the crank is moved in the usual fashion in the forward direction. Then a further motion is added or subtracted by rotating the index plate after locking the plate with the plunger. This is termed as compound indexing.

For example, if the indexing is done by moving the crank by 5 holes in the 20 hole circle and then the index plate together with the crank is indexed back by a hole with the locking plunger registering in a 15 hole circle as shown in Fig. 7.23.

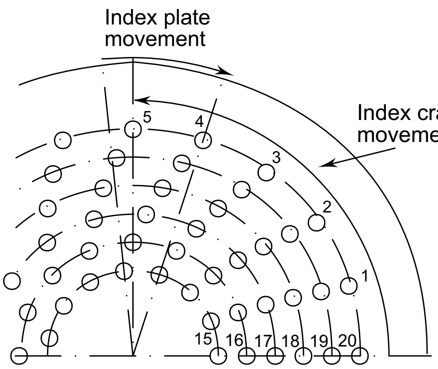


Fig. 7.23 *Compound indexing using the Index plate no. 1 of Brown and Sharpe Dividing head with 5 holes in 20-hole circle minus 1 hole in 15-hole circle*

The total indexing done is then

$$\frac{5}{20} - \frac{1}{15} = \frac{11}{60}$$

i.e., 11 holes in a 60-hole circle. Unfortunately the 60-hole circle is not available in the Brown & Sharpe range of index plates. Similarly it is possible to have the two motions in the same direction as well. In that case the total indexing will be

$$\frac{5}{20} + \frac{1}{15} = \frac{19}{60}$$

i.e., 19 holes in a 60-hole circle.

It is therefore possible that by following this method any other indexing can also be done.

Example 7.3

Indexing 77 divisions.

Solution The indexing required is $\frac{77}{40}$ considering the worm.

It is necessary to convert this fraction into two fractions corresponding to the two hole circles in the same plate. Use trial and error method to obtain the same.

First step in compound indexing is to factorise the number into a suitable hole circles available in a single plate.

$$77 = 11 \times 7$$

This can be achieved by using plate 2 with

3 holes in 33-hole circle, and

3 holes in 21-hole circle.

This means that 33 and 21 are the hole circles that have been identified for this indexing. The next step is to find the exact indexing required.

$$\frac{X}{21} \pm \frac{Y}{33} = \frac{360}{693} = \frac{40}{77}$$

$$33 \times X \pm 21 \times Y = 360$$

By trial and error, we get, $X = 9$, and $Y = 3$

$$33 \times 9 + 21 \times 3 = 360$$

$$\frac{9}{21} + \frac{3}{33} = \frac{40}{77}$$

Hence the indexing required is 9 holes in the 21-hole circle added to 3 holes in the 33-hole circle to get 77 divisions.

Example 7.4

Indexing 141 divisions.

Solution The indexing required is $\frac{141}{40}$ considering the worm.

It is necessary to convert this fraction into two fractions corresponding to the two hole circles in the same plate. Use trial and error method to obtain the same.

First step in compound indexing is to factorise the number into suitable hole circles available in a single plate.

$$141 = 47 \times 3$$

This can be achieved by using plate 3 with 39 and 47-hole circles.
The next step is to find the exact indexing required.

$$\frac{X}{39} \pm \frac{Y}{47} = \frac{13 \times 40}{39 \times 47} = \frac{40}{141}$$

$$47 \times X \pm 39 \times Y = 520$$

By trial and error, we get, $X = 26$, and $Y = 18$

$$47 \times 26 - 18 \times 39 = 520$$

$$\frac{26}{39} - \frac{18}{47} = \frac{40}{141}$$

Hence, the indexing required is 26 holes in the 39-hole circle subtracted by 18 holes in the 47-hole circle to get 141 divisions.

7.5.3 Angular Indexing

Sometimes it is desirable to carry out indexing using the actual angles rather than equal numbers along the periphery, then angular indexing would be useful. The procedure remains the same as the previous cases, except that the angle will have to be first converted to equivalent divisions. Since the 40 revolutions of the crank equals to a full rotation of the work piece, which means 360°, then one revolution of the crank is equivalent to 9°.

Example 7.5

Calculate the indexing for 41°.

Solution Indexing required = $\frac{41}{9} = 4\frac{5}{9}$

This is equivalent to 4 full rotations of the crank followed by 10 holes in the 18-hole circle in plate no. 1.

Example 7.6

Calculate the indexing for 15° 30′.

Solution Indexing required = $\frac{15.5}{9} = 1\frac{6.5}{9} = 1\frac{13}{18}$

This is equivalent to 1 full rotation of the crank followed by 13 holes in the 18-hole circle in plate no. 1.

Example 7.7

Calculate the indexing for 16° 40'.

Solution Indexing required = $\frac{16\frac{2}{3}}{9} = 1\frac{7\frac{2}{3}}{9} = 1\frac{23}{27}$

This is equivalent to 1 full rotation of the crank followed by 23 holes in the 27-hole circle in plate no. 2.

7.5.4 Differential Indexing

Though compound indexing as explained above is a convenient way to get any indexing required, it is fairly cumbersome to use in practice. Hence differential indexing is used for that purpose which is an automatic way to carry out the compound indexing method. The arrangement for differential indexing is shown in Fig. 7.24.

In differential indexing, the index plate is made free to rotate. A gear is connected to the back end of the dividing head spindle while another gear mounted on a shaft, is connected to the shaft of the index plate through bevel gears as shown in Fig. 7.24. When the index crank is rotated, the motion is communicated to the work piece spindle. Since the work piece spindle is connected to the index plate through the intermediate gearing as explained above, the index plate will also start rotating. If the chosen indexing is less than the required one, then the index plate will have to be moved in the same direction as the movement of the crank, to add the additional motion. If the chosen indexing is more then the plate should move in the opposite direction to subtract the additional motion.

The direction of the movement of the index plate depends upon the gear train employed. If an idle gear is added between the spindle gear and the shaft gear in case of a simple gear train, then the index plate will move in the same direction to that of the indexing crank movement. In the case of a compound gear train an idler is to be used when the index plate is to move in the opposite direction. The procedure of calculation is explained with the following example.

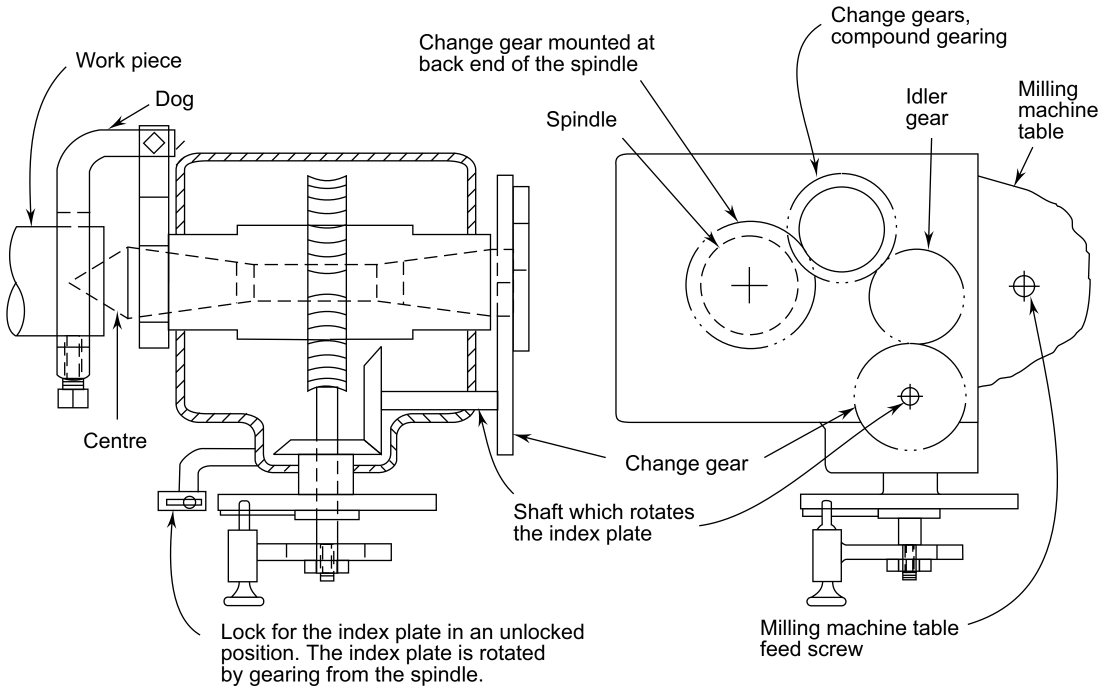


Fig. 7.24 *Dividing head setup for differential indexing*

The change gear set available is

24, 24, 28, 32, 40, 44, 48, 56, 64, 72, 86 and 100

Example 7.8

Obtain the indexing for 97 divisions.

Solution Required indexing is $\frac{40}{97}$ which cannot be obtained with any of the index plates available. Choose a nearest possible division. For example the indexing decided is $\frac{40}{100} = \frac{2}{5} = \frac{8}{20}$

The actual indexing decided is 8 holes in 20-hole circle. This indexing will be less than required. Ideally the work piece should complete one revolution when the crank is moved through the 97 turns at the above identified indexing. Actual motion generated when the crank is moved 97 times is

$$40 - \frac{97 \times 40}{100} = \frac{3 \times 40}{100}$$

Hence, the index plate has to move forward by this amount during the 97 turns to compensate for the smaller indexing being done by indexing crank. Hence, the gear ratio between the spindle and the index crank is

$$\frac{3 \times 40}{100} = \frac{6}{5}$$

Change gear set used is $\frac{\text{Driven}}{\text{Driver}} = \frac{6}{5} = \frac{48}{40}$

An idler gear is to be used since the index plate has to move in the same direction.

Example 7.9

Obtain the indexing for 209 divisions.

Solution Required indexing is $\frac{40}{209}$ which cannot be obtained with any of the index plates available. Choose a nearest possible division. For example, the indexing decided is $\frac{40}{200} = \frac{4}{20}$.

The actual indexing decided is 4 holes in 20-hole circle. This indexing will be more than required. Ideally the work piece should complete one revolution when the crank is moved through the 209 turns at the above identified indexing. Actual motion generated when the crank is moved 209 times is

$$40 - \frac{209 \times 40}{200} = -\frac{9 \times 40}{200}$$

Hence, the index plate has to move in the reverse by this amount during the 209 turns to compensate for the larger indexing being done by indexing crank. Hence the gear ratio between the spindle and the index crank is

$$\frac{9 \times 40}{200} = \frac{36}{20}$$

Change gear set used is $\frac{\text{Gear on spindle}}{\text{Gear on index crank}} = \frac{36}{20}$

A few points to be remembered during the differential indexing is

(a) Use the hole circles for indexing which will easily factorise with the available gear set. For example, in the case of Brown & Sharpe, 18, 20, 21 and 27 hole circles should be used.
(b) The difference from the actual to the approximate indexing should be a small value such that the change gear set can accommodate this ratio. For example a total difference of 0.5 to 1.5 will be most convenient.
(c) The idler gear has to be provided when the index plate has to move in the opposite direction to that of the crank movement.

7.6 MILLING TIME AND POWER ESTIMATION

7.6.1 Milling Time Estimation

Typical process parameters used in milling operations are given in Table 7.1. The cutting speed in milling is the surface speed of the milling cutter. Thus

$$V = \frac{\pi D N}{1000}$$

TABLE 7.1 **Data for milling**

Work Material	Hardness	HSS		Carbide	
	BHN	Speed m/min	Feed mm/tooth	Speed m/min	Feed mm/Tooth
C20 Steel	110–160	20	0.13	90	0.18
C35 Steel	120–180	25	0.13	80	0.18
C50 Steel	160–200	20	0.13	60	0.18
Alloy Steel	180–220	30	0.10	60	0.18
Alloy Steel	220–300	18	0.08	90	0.18
Alloy Steel	220–300	14	0.08	60	0.15
Alloy Steel	300–400	14	0.05	60	0.13
Stainless steel	200–300	20	0.10	85	0.13
Cast iron	180–220	16	0.18	58	0.20
Malleable iron	160–240	27	0.15	85	0.18
Cast steel	140–200	16	0.15	50	0.18
Copper	120–160	38	0.15	180	0.15
Brass	120–180	75	0.28	240	0.25
Bronze	160–200	38	0.18	180	0.15
Aluminium	70–105	120	0.28	240	0.25
Magnesium	40– 60	210	0.28	380	0.25

Where, V = cutting speed (surface), m/min

D = diameter of the milling cutter, mm

N = rotational speed of the milling cutter, rpm

Flat surfaces can be generated by using the slab milling as well as face milling. However, each of these operations is different in terms of the actual machining time required. Schematically the slab milling operation is shown in Fig. 7.25. All other milling operations using the arbor mounted milling cutters will be similar in approach. The milling cutter will have to traverse beyond the actual work piece by a distance termed as the approach allowance, A, which is given by

$$\text{Approach distance, } A = \sqrt{\left(\frac{D}{2}\right)^2 - \left(\frac{D}{2} - d\right)^2} = \sqrt{d\,(D - d)}$$

Where D = diameter of the slab milling cutter

d = depth of cut

$$\text{Time for one pass} = \frac{1 + 2 \times A}{fZN} \text{ minutes}$$

Where Z = number of teeth in the milling cutter

f = feed per tooth, mm

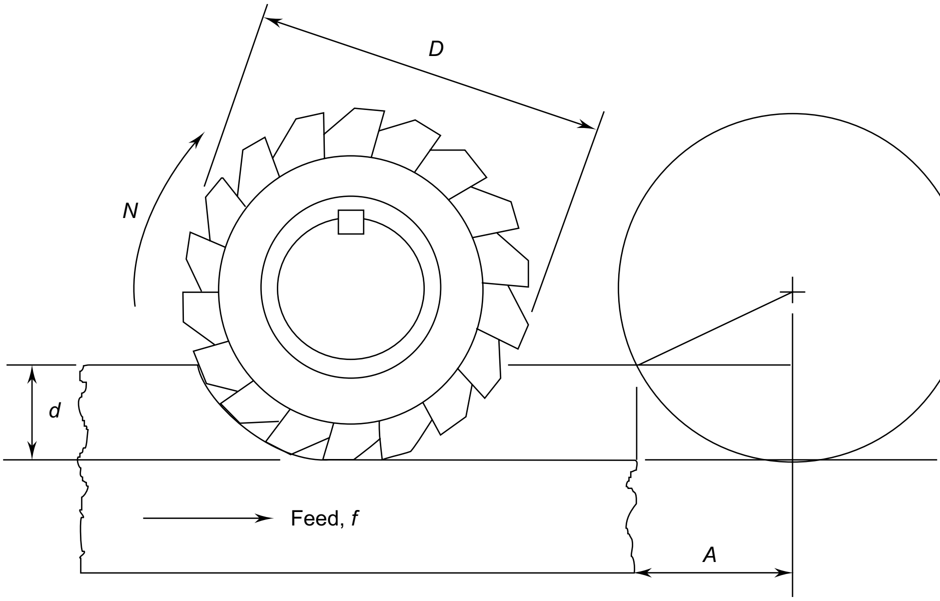


Fig. 7.25 *Slab milling operation*

Example 7.10

A C50 steel flat surface of 100 × 250 mm is to be produced on a horizontal axis milling machine. A HSS slab mill of 100 mm diameter and 150 mm width is to be used for the purpose. The milling cutter has 8 teeth. Calculate the machining time assuming that entire stock can be removed in one depth of 2 mm.

Solution Given $Z = 8$

$D = 100$ mm

$D = 2$ mm

From the table,

Cutting speed, $V = 20$ m/min

Feed rate, $f = 0.13$ mm/tooth

Approach distance, $A = \sqrt{d(D-d)} = \sqrt{2(100-2)} = 14$ mm

$$\text{Spindle speed, } N = \frac{1000 \times 20}{\pi \times 100} = 63.66 \approx 65 \text{ rev/min}$$

$$\text{Time for machining} = \frac{150 + 2 \times 14}{0.13 \times 8 \times 65} = 2.633 \text{ minutes}$$

Fig. 7.26 shows the situation of face milling operation using a vertical axis milling machine.

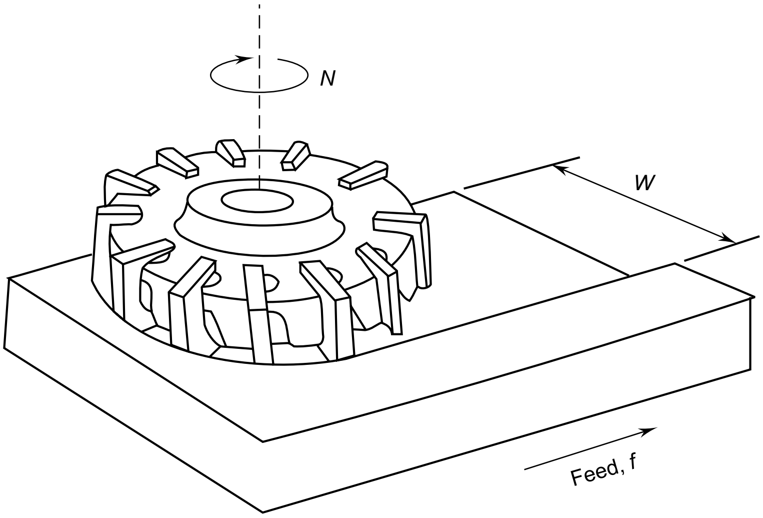


Fig. 7.26 *Face milling operation*

Approach distance for the face milling case is given as

$$A = \frac{D}{2} \qquad \text{for} \quad W = \frac{D}{2} \text{ up to } D$$

$$A = \sqrt{W(D-W)} \qquad \text{for} \quad W < \frac{D}{2}$$

Where W = width of cut

Example 7.11

A surface 115 mm wide and 250 mm long is to be rough milled with a depth of cut of 6 mm by a 16-tooth cemented carbide face mill 150 mm in diameter. The work material is alloy steel (200 BHN). Estimate the cutting time.

Solution Given $Z = 16$

$D = 150$ mm

$d = 6$ mm

$W = 115$ mm

From the table,

Cutting speed, $V = 60$ m/min

Feed rate, $f = 0.18$ mm/tooth

Spindle speed, $N = \dfrac{1000 \times 60}{\pi \times 150} = 127.32 \approx 125$ rev/min

Since $W < \dfrac{D}{2}$

Approach distance, $A = \sqrt{115(150-115)} = 63.44 \approx 65$ mm

Time for machining $= \dfrac{250 + 2 \times 65}{0.18 \times 16 \times 125} = 1.06$ minutes

7.6.2 Milling Power Estimation

Similar to the turning, milling power can be calculated based on the empirical relations that have been developed. See Machinery's handbook for detailed information related to the calculations. It is assumed that the milling power is proportional to the material removal rate. Material removal rate (Q) in milling is given by

$$Q = \frac{f_m\, w\, d}{60000}\ \text{cm}^3/\text{s}$$

Where f_m = feed rate in mm/min = fZN
w = width of cut in mm
d = depth of cut in mm
f = feed rate in mm/tooth as normally given in cutting tables
Z = number of teeth in the milling cutter
N = rotational speed of the spindle in rpm

Milling power (P_m) in horse power units at the cutting tool is given by

$$P_m = K_p QCWhp$$

Where P_m = milling power in hp
K_p = power constant as given in Table 7.2.
C = Feed factor given in Table 7.3
W = Tool wear factor given in Table 7.4

Table 7.2 Power constant for Milling (Machinery's handbook)

Work Material	Hardness BHN	Power Constant
Plain carbon Steel	100–120	1.80
	120–140	1.88
	140–160	2.02
	160–180	2.13
	180–200	2.24
	200–220	2.32
	220–240	2.43
Alloy Steel	180–200	1.88
	200–220	1.97
	220–240	2.07
	240–260	2.18
Cast iron	120–140	0.96
	140–160	1.04
	160–180	1.42
	180–200	1.64
	200–220	1.94
	220–240	2.48
Malleable iron	150–175	1.15
	175–200	1.56
	200–250	2.24
	250–300	3.22

TABLE 7.3 **Feed factors for power calculation (Machinery's handbook)**

Feed, mm/Tooth	Feed Factor	Feed, mm/Tooth	Feed Factor
0.02	1.70	0.22	1.06
0.05	1.40	0.25	1.04
0.07	1.30	0.28	1.01
0.10	1.25	0.30	1.00
0.12	1.20	0.33	0.98
0.15	1.15	0.35	0.97
0.18	1.11	0.38	0.95
0.20	1.08	0.40	0.94

TABLE 7.4 **Tool wear factors for power calculation (Machinery's handbook)**

Operation	Tool-wear Factor
Slab milling and end milling	1.10
Light and medium face milling	1.10 to 1.25
Heavy face milling	1.30 to 1.60

Example 7.12

Calculate the power required to rough mill a surface 115 mm wide and 250 mm long with a depth of cut of 6 mm by a 16-tooth cemented carbide face mill that is 150 mm in diameter. The work material is alloy steel (200 BHN).

Solution Given $Z = 16$; $d = 6$ mm; $W = 115$ mm

From Table 7.1, Cutting speed, $V = 60$ m/min

Feed rate, $f = 0.18$ mm/tooth

Spindle speed, $N = \dfrac{1000 \times 20}{\pi \times 100} = 63.66 \approx 65$ rev/min

Where f_m = feed rate in mm/min = $fZN = 0.18 \times 16 \times 65 = 187.2$ mm/min

Material removal rate (Q) is

$$Q = \frac{f_m wd}{60000} \text{ cm}^3/\text{s} = \frac{187.2 \times 115 \times 6}{60000} = 2.1528 \text{ cm}^3/\text{s}$$

From Table 7.2, $K_p = 1.88$

From Table 7.3, $C = 1.11$

From Table 7.4, $W = 1.30$

Milling power (P_m) in horse power units at the cutting tool is

$$P_m = 1.88 \times 2.1528 \times 1.11 \times 1.30 = 5.84 \text{ hp}$$

7.6.3 Milling Trouble Shooting

Milling is a relatively slow speed machining operation. However the material removal rate is generally high and in view of the production operations in which it is generally used, it is important to achieve the accuracy and finish desired. The following table lists some of the problems encountered and their likely causes.

Table 7.5 Problems encountered in milling operations

Problem Encountered	Possible Cause and Remedy
Chatter is characterised by large amplitude vibrations which spoil the surface finish of the component as well as reduce the tool life	The machine tool, fixture and the work piece are less rigid The cutting force is high. Reduce the depth of cut. Milling cutter needs re-sharpening. No cutting fluid is applied. The milling cutter has straight teeth. Change to helical cutter.
Poor dimensional tolerance	High cutting force, which causes the deflection leading to inaccuracy. Reduce the depth of cut. The chips fill the gaps in the cutter teeth and are not flushed properly by the cutting fluid. Chips in the work holding surfaces causing locational errors.
Lower milling cutter life	Higher cutting forces causing the cutter to dull quickly. Reduce the depth of cut. Insufficient cutting fluid applied.
Poor surface finish on milled work pieces	Higher feed rates used not commensurate with the required finish. The milling cutter needs re-sharpening. The cutting speed is low. The number of teeth on the milling cutter is low which is suitable for roughing operation. Use a cutter with larger number of teeth.

7.7 SPECIAL SETUPS

7.7.1 Spiral (Helical) Milling

The use of dividing head for dividing the periphery of the work piece has been discussed earlier. Since the dividing head has the ability to move the periphery in a continuous fashion, by linking the table feed screw with the indexing crank, it would be possible to get a helical motion. This is termed as spiral or helical milling.

In order to achieve the helical milling the worm spindle of the dividing head is geared to the lead screw of the machine table, such that when the lead screw is turned the worm is also rotated by a ratio depending upon the gearing provided. The gearing has to be fixed based on the lead of the helix to be milled as well as the lead of the lead screw.

The work piece will rotate a full revolution when the index crank is moved by 40 full revolutions. Hence

$$\text{Lead of the machine} = 40 \times \text{Lead of the table feed screw}$$

If the lead of the machine is known, the gearing to be connected between the table and index shaft is given by

$$\frac{\text{Gear on index plate shaft}}{\text{Gear on table lead screw}} = \frac{\text{Lead of the machine}}{\text{Lead of the helix to be cut}}$$

The hand of the helix whether right hand or left hand is controlled by the presence of an idler gear in the gear train.

Example 7.13

The milling machine has a table lead screw with a lead of 5 mm. Calculate the gear train necessary to cut a helix of lead of 480 mm.

Solution Lead of the machine = 40 × 5 = 200 mm

$$\frac{\text{Gear on index plate shaft}}{\text{Gear on table lead screw}} = \frac{200}{480} = \frac{40}{48} \times \frac{32}{64}$$

When the dividing head is used in a horizontal axis milling machine, it is necessary to orient the arbor mounted milling cutter to be rotated by an angle equivalent to the helix angle such that the milling cutter will not interfere with the side of the helix being cut a shown in Fig. 7.27. For this purpose it is necessary to use a universal machine where the table will be swivelled by the helix angle. Helix angle, α is given by

$$\tan \alpha = \frac{\pi \times \text{Diameter of the work}}{\text{Lead of helix}}$$

If the helix is cut by a vertical milling machine using an end mill, then no table swivelling is required since the cutting portion of the end mill is perpendicular to the cut surface.

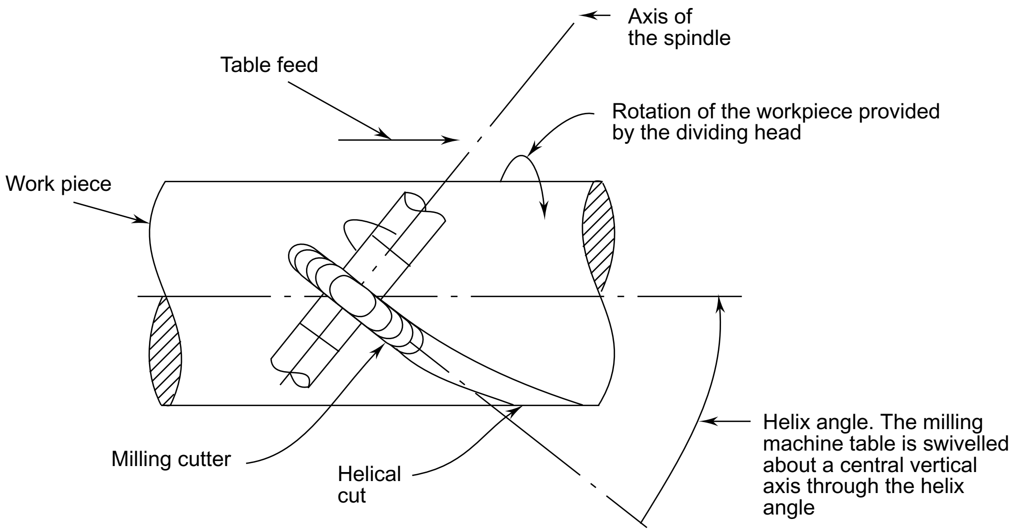


Fig. 7.27 *Helical milling operation*

Example 7.14

A left hand helical flutes are to be milled in a reamer that is 40 mm in diameter with a lead of 800 mm. If the flutes are 8 mm deep and the machine lead screw has a lead of 5 mm, calculate the necessary settings required.

Solution Lead of the machine = 40 × 5 = 200 mm

$$\frac{\text{Gear on index plate shaft}}{\text{Gear on table lead screw}} = \frac{200}{800} = \frac{24}{48} \times \frac{32}{64}$$

To calculate the helix angle, α, take the average depth of the flute

$$\tan \alpha = \frac{\pi \times \text{Diameter of the work}}{\text{Lead of helix}} = \frac{\pi \times 32}{800} = 0.1257$$

or, Helix angle, $\alpha = 7° \, 10'$

7.7.2 Cam Milling

A setup similar to the spiral milling explained above could be utilised for machining plate cams with uniform rise or fall. Typical setup used is shown in Fig. 7.28. The cam milling is carried in a universal milling machine with a vertical milling attachment. The tool used is the end mill with sufficiently long cutting length and having a diameter equal to the diameter of the follower to be used with the cam.

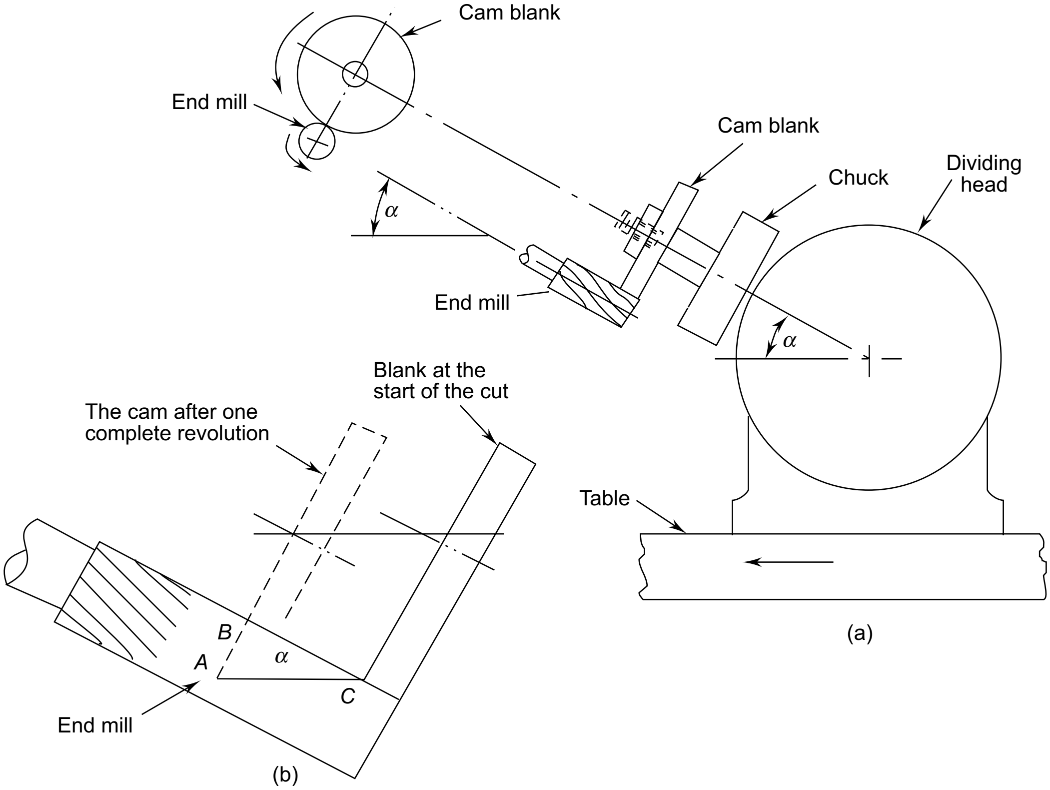


Fig. 7.28 *Cam milling setup*

The shaft of the dividing head index plate is connected to the table lead screw by gears as in spiral milling. The plate cam blank is held in the spindle of the dividing head, which is inclined to the horizontal at an angle, α. The end mill is also inclined at the same angle such that the axis of the cutter and axis of the cam to be milled are parallel. As the table starts moving, the plate cam rotates through the dividing head spindle and the distance between the cutter and the cam becomes smaller, thereby machining the cam profile. Since the lead of the table lead screw is constant, the rise or fall of the cam machined also becomes constant, the actual value depending upon the machine setting.

The inclination angle, α, that is set can be calculated as follows:

$$\sin \alpha = \frac{\text{Lead of the cam} \times \text{Gear ratio}}{\text{Machine lead}}$$

If the setting angle is zero, the lead of the cam becomes zero as shown in Fig. 7.29. Any suitable gear ratio can be used.

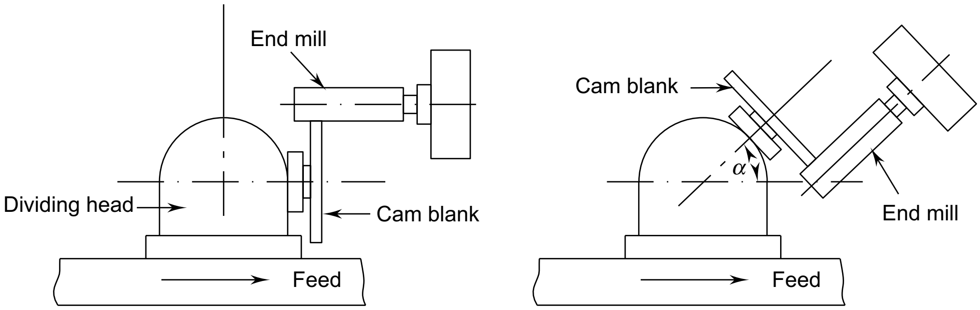


Fig. 7.29 *Cam milling setup at two different angles*

Example 7.15

Calculate the necessary settings for milling a plate cam with 10 mm rise in 60° revolutions. Lead screw of the machine has a lead of 5 mm.

Solution Machine lead = 40 × 5 = 200 mm

$$\text{Lead of the cam} = \frac{10 \times 360}{60} = 60 \text{ mm}$$

Assume a gear ratio of 2. Then the angular setting required is

$$\sin \alpha = \frac{\text{Lead of the cam} \times \text{Gear ratio}}{\text{Machine lead}} = \frac{60 \times 2}{200} = 0.6$$

Setting angle, $\alpha = 36° \, 52'$

7.7.3 Thread Milling

Cutting threads in the parts is a common exercise for most machine shops. Threads can be cut using taps discussed in chapter 8 or milled using a milling cutter. Thread milling cutters look similar to taps, but function entirely differently. The thread milling is a process that produces threads by circular ramping movement of thread shaped rotating milling cutter as it moves round the work piece. One of the major disadvantages of tapping is that a different size tap is required for each size of hole that requires thread. This is particularly inconvenient because it can consume a large number of valuable, but limited positions in the CNC tool magazine. Plus, having to switch tapping tools for all of the various sized holes increases the cycle time.

The shape of the thread milling cutter is shown in Fig 7.30 where the similarity to the tap can be seen. The thread milling cutter is inserted into the hole along the axis of the spindle, and follows a helical path with both circular and vertical feed motions as shown in Fig 7.30. During this circular motion the thread mill must be moved along the Z-axis of the machine one thread pitch (or lead) to produce a thread. This combination of circular and vertical feed motions is termed as "helical interpolation". The CNC controller of the machine to be used should have the capability of helical interpolation. After completing the machining, the cutter returns to the centre of the hole and is extracted from the part.

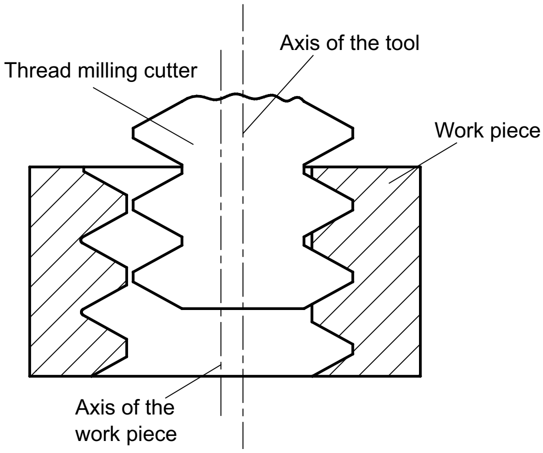


Fig. 7.30 *Schematic of internal thread milling operation*

Thread milling cutters are available in HSS, indexable carbide and solid carbide depending upon the application and the hole sizes. HSS thread mills are generally produced by powder metallurgy and are coated with TiN, TiCN, or TiAlN to enhance the tool life. These are generally used for softer materials such as those with hardness less than HRC 30/32. Solid carbide is generally used for production of threads for materials up to HRC 62. These are used for smaller thread sizes. Indexable carbide thread mills are used for sizes 20 mm or larger, and accommodate a variety of pitches by replacing the insert.

The quality of thread obtained by threading, particularly the small size holes will appear a bit jagged because the chips do not have sufficient space to be removed during the machining process.

Some of the situations where thread milling is preferred compared to tapping:

- Cutting threads in thin-walled components
- Cutting threads in tough materials that are likely to generate high cutting forces
- Materials that cause chip breaking and chip removal problems
- Cutting threads all the way to the bottom of a blind hole
- Likelihood of the tap breaking inside expensive parts as thread milling cutters can be easily removed unlike a broken tap
- To reduce the tool inventory

SUMMARY

Milling is a machining operation using a multi-tooth cutter that rotates and generates short chips.

- Milling machines can be classified based on the axis orientation, or the machine tool structure.
- There are a variety of milling machines used for various production applications.
- A large variety of milling cutters are used to generate some of the most complex shapes in the parts.
- Arbour mounted cutters such as plain milling cutter and side and face cutters are used in horizontal axis machines to generate flat surfaces.
- End mills are the most common tools used in the vertical axis machines to generate a variety of surfaces.
- Milling operations are classified, based on the direction of rotation of the cutter and the movement of the table, as conventional and climb milling.
- Common work holding devices used in milling are the vice, and a variety of clamps.
- A number of milling setups are used to cater to the wide range of machining applications that are used with the milling machines.
- Dividing head is an important attachment that is used to divide the periphery of the work piece into a number of divisions to machine surfaces such as gear teeth.
- Machining time calculation is similar to turning based on the cutting speed and feed and the part geometry.
- Milling power can be estimated based on the material properties using empirical relations.
- Special setups such as cam milling can be used for milling special surfaces.

Questions

7.1 Explain the characteristics that distinguish a milling process from other machining processes.

7.2 Describe the differences between a lathe and milling machine in terms of the types of surfaces generated, the types of tools used, and applicability for general and production applications.

7.3 Give a brief classification of various milling machines used in the industry giving a brief note on their application.

7.4 How is a milling machine specified?

7.5 What are the various types of milling cutters that are used in milling?

7.6 Describe the application and relative merits of various types of milling cutters that are used in milling.

7.7 List the motions of the arbour mounted milling cutter with respect to the work piece.

7.8 What are the various work holding devices used in milling? Explain their relative applications and disadvantages.

7.9 What are the various types of end mills used in milling? Explain their applications.

7.10 Differentiate between up-milling and down milling. Explain their applications mentioning the most commonly used method.

7.11 Explain the difference between straight and helical slab mills bringing out the advantages of the use of helical teeth.

7.12 Explain the applications and differences with neat sketches, the following with reference to milling:
(a) Straddle milling (b) Gang milling

7.13 Sketch typical setups for
(a) Reciprocal milling (b) String milling

7.14 Explain the construction of a dividing head giving the applications for which it can be used.

7.15 What are the differences between compound indexing and differential indexing? Explain the relative merits.

7.16 Explain with a sketch what you understand by the words 'helix angle' and 'direction of cut' in the case of milling. What is their importance with respect to machining performance? Explain the basis on which these are selected.

7.17 Briefly explain some of the problems caused in milling. Give their causes and probable remedies.

7.18 Describe a method used for manufacturing (machining) the flutes on a twist drill with a neat sketch.

7.19 Describe the setup that one can use for milling cams in a milling machine. Explain neatly with a sketch, the various attachments that one needs to use for such machining. Explain the limitations of such a setup.

7.20 Make a sketch for milling cams on a horizontal knee and column type milling machine. If the lead on the cam is to be r, show the necessary setup. Explain any necessary precautions.

Problems

7.1 A 20 mm × 150 mm diameter HSS side and face milling cutter is to be used to cut a groove into a piece of brass with one cut. The groove is 20 mm wide, 4 mm deep and 250 mm long. Calculate the total machining time. Justify the assumptions made if any. [0.558 minutes]

7.2 In a slab milling operation, the milling cutter has 20 teeth and is 100 mm in diameter. The rotational speed of the cutter is 5 RPS. If the flat surface to be generated is 200 mm by 50 mm and feed per tooth is 0.013 mm/rev., calculate the machining time required for 100 pieces. The depth of cut may be taken as 6 mm. Specify any assumptions made. [1.15 minutes]

7.3 A surface 115 mm wide and 250 mm long is to be rough milled with a depth of cut of 6 mm by a 16-tooth cemented carbide face mill 150 mm in diameter. The work material is medium hard cast iron (220 – 260 BHN). Estimate the cutting time and justify the process parameters used. [4.6296 minutes]

7.4 A flat surface of 250 × 350 mm is to be produced on a horizontal axis milling machine. A slab mill of 100 mm diameter and 150 mm width is to be used for the purpose. Calculate the machining time assuming that entire stock can be removed in one depth of 1 mm. Make only the requisite assumptions. [6.25 minutes]

7.5 A grey cast iron casting plane surface, which is 150 mm wide and 450 mm long is to be finished by milling. It can be machined using a face mill of 200 mm diameter with 10 teeth made of cemented carbide. The cutting speed is 70 m/min and feed per tooth is 0.25 mm. Calculate the machining time for finishing the job if all the stock is removed in a single cut. [17 500 mm^3/min]

7.6 The job in Problem 7.5 can also be done by using a high speed steel slab mill of 150 mm diameter and 200 mm wide with 8 teeth. If the cutting speed is 40 m/min and feed per tooth is 0.25 mm, compare the machining times and material removal rates in both cases.

[3.75 minutes, 10 000 mm^3/min]

7.7 The above job can also be completed by shaping using a high speed steel single point cutting tool on a hydraulic shaper. The feed can be taken as 0.4 mm/stroke while the cutting speed is 40 m/min. Compare the machining times and material removal rates in all cases.

[6.68 minutes, 16 000 mm^3/stroke]

7.8 Calculate the following indexing requirements:
(a) 41 divisions (b) 76 divisions
(c) 187 divisions

[40 in 41; 10 in 19; Differential – 4 in 18 with change gears 56 and 40 with idler gear]

7.9 Calculate the following indexing requirements:
(a) 10° 26′ (b) 41° 34′
(c) 55° 45′

[1 rotation plus 4 in 27; 1 rotation plus 11 in 18; 1 rotation plus 5 in 27]

7.10 Calculate the indexing requirement for 127 divisions on a milling machine equipped with a differential indexing head. The index plates available are

Plate no. 1: 15, 16, 17, 18, 19, 20 holes
Plate no. 2: 21, 23, 27, 29, 31, 33 holes
Plate no. 3: 37, 39, 41, 43, 47, 49 holes

The change gear set available is

24, 24, 28, 32, 40, 44, 48, 56, 64, 72, 86, 100

[6 in 18 plus change gear 56 and 24 with idler gear]

7.11 A plate cam of thickness 10 mm and radius 75 mm is to be milled on a horizontal axis milling machine. The cam has a uniform rise of 40 mm over 90°. Explain with a neat sketch how this can be accomplished. What modifications in the setup are required, if dwell is to be machined over part of the cam periphery? If the table lead screw is having 6 mm lead, calculate the actual machining time for cutting the uniform rise portion. Make any necessary assumptions, but justify them.

[23° 34′ 41″]

7.12 A helix of lead 150 mm on a shaft of diameter 80 mm is to be milled on a horizontal axis milling machine. Show the setup used with a neat sketch. Explain the precautions to be taken during the setup. If the table lead screw is having 6 mm lead, calculate the actual machining time for cutting the helix considering the length of helix to be one half of the lead. Make any necessary assumptions, but justify them. [59° 10’]

7.13 A C20 steel disc of 300 mm diameter and 10 mm thick is to be cut at 93 equally spaced points on the periphery. The milling machine is equipped with a simple dividing head without any change gear set. Calculate the necessary setting required. Assume that the dividing head is provided with a 40 teeth worm wheel. [9 in 27 plus 3 in 31 – Compound indexing]

Multiple Choice Questions

7.1 An important characteristic of a milling process unlike any other machining processes is
(a) Interrupted cutting
(b) Small size chips
(c) Variable chip thickness
(d) All of the above

7.2 Identify the machine tool that is most versatile from the following list of machine tools
(a) Gap bed lathe
(b) Vertical axis milling machine
(c) Pillar drilling machine
(d) Surface grinding machine

7.3 The following type of milling machine is normally used for very high production rates
(a) Horizontal knee and column milling machine
(b) Vertical knee and column milling machine
(c) Simplex bed type milling machine
(d) Duplex bed type milling machine

7.4 The following milling cutter is used for machining rectangular slots on a horizontal knee and column milling machine
(a) Slab milling cutter
(b) Face mill
(c) Side and face milling cutter
(d) Shell end mill

7.5 The following milling cutter is used for machining rectangular slots on a vertical knee and column milling machine
(a) End mill
(b) Ball end mill
(c) Slitting saw
(d) Side and face milling cutter

7.6 Advantage of down milling (climb milling) compared to up milling (conventional milling) is
(a) Can be used in machines without backlash eliminator
(b) Can be used for machining castings or rolled steel directly
(c) Work piece need not be clamped tightly
(d) Can only be used for rigid parts

7.7 Disadvantage of down milling (climb milling) compared to up milling (conventional milling) is
(a) Cannot be used in machines without backlash eliminator
(b) It requires more power to cut
(c) Work piece need to be clamped tightly
(d) Can only be used for rigid parts

7.8 String milling is used for
(a) Large work pieces
(b) Small work pieces
(c) Heavy work pieces
(d) All of the above

7.9 Gang milling is used for
(a) Large work pieces
(b) Small work pieces
(c) A number of milling cutters are used to cut simultaneously
(d) Only one milling cutters is used to cut heavy work piece

7.10 To cut an involute gear on a milling machine the following is required
(a) Angle milling cutter
(b) Differential indexing head
(c) Slab milling cutter
(d) None of the above

7.11 Poor surface finish on the milled work pieces is caused by
(a) Higher feed rates
(b) Milling cutter is worn out and needs resharpening
(c) Cutting speed is low
(d) All of the above

7.12 Chatter (large amplitude vibration) during milling is caused by
- (a) Milling machine is less rigid
- (b) High depth of cut increasing the cutting force
- (c) No cutting fluid is applied
- (d) All of the above

7.13 An end mill having 4 teeth is rotating at 250 RPM. If the feed per tooth is given as 0.1 mm, what is the table feed in mm/min?
- (a) 100 mm/min
- (b) 10 mm/min
- (c) 250 mm/min
- (d) 25 mm/min

Answers to MCQs

7.1 (d)	7.2 (b)	7.3 (d)	7.4 (c)	7.5 (a)
7.6 (c)	7.7 (a)	7.8 (b)	7.9 (c)	7.10 (b)
7.11 (d)	7.12 (d)	7.13 (a)		

CHAPTER 8

Hole Making Operations

Objectives

Practically all components have some holes that need to be completed before the part could be used. Thus, hole making operations are an important part of a machine shop. After completing this chapter, the reader will be able to

- Understand the various types of holes that are used in manufactured parts
- Know the various elements of a twist drill geometry as well as the types of drills used in industry
- Understand the variety of machine tools and their construction
- Calculate the drilling time and the power required for the drilling operation
- Understand and take precautions while drilling deep holes
- Select the appropriate finishing operations such as reaming and boring
- Use tapping with appropriate speeds to get good screw threads in holes
- Know the differences between various other hole making operations

8.1 INTRODUCTION

Machining round holes in metal stock is one of the most common operations in the manufacturing industry. It is estimated that of all the machining operations carried out, about 20% are hole making operations. Practically any work piece will not leave the machine shop without making a hole in it. The various types of holes are shown in Fig. 8.1.

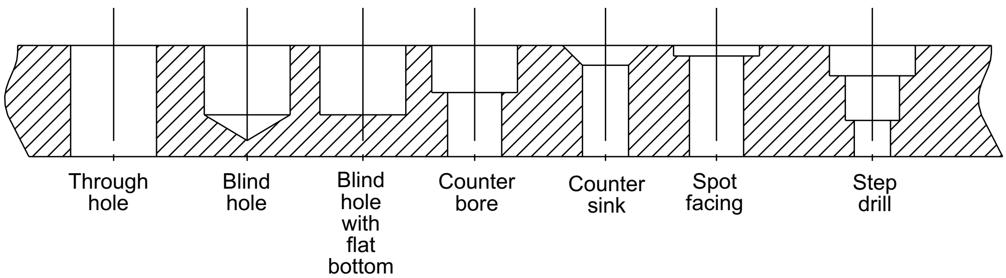


Fig. 8.1 *Various types of holes*

The types of hole making operations performed on the holes are:

- Drilling
- Boring
- Reaming
- Counter sinking
- Counter boring and
- Tapping

Whereas drilling is used for making a hole in solid material, all the other operations are used to enlarge the hole or improve the quality of the hole, depending upon the requirement. Comparative process capabilities of these processes are given in Table 8.1. In the Table l is the length of the hole and D is the diameter of the hole. In this chapter the basic details related to all these operations will be discussed.

TABLE 8.1 Comparative characteristics of hole making operations

Hole Parameter	Drilling	Reaming	Boring	Counter Boring
Smallest size, mm	1.6	1.6	9.5	6
Largest size, mm	50	100	250	75
Negative tol., mm	$0.896 \times D^{0.5}$	0.010	0.008	$0.512 \times D^{0.5} + 0.064$
Positive tol., mm	$0.896 \times D^{0.5} + 0.075$	0.010	0.008	$0.640 \times D^{0.5} + 0.075$
Straightness, mm	$0.013 \times (l/D)^3 + 0.050$	2.5	0.013	0.250
Roundness, mm	0.100	0.013	0.025	0.075
Parallelism, mm	$0.025 \times (l/D)^3+0.075$	0.25	0.025	0.250
Depth limit, mm	300	400	225	500
True position, mm	±0.200	±0.25	±0.003	±0.003
Surface finish, µm	2.54	0.41	0.20	1.25

8.2 DRILLING

8.2.1 Twist Drill Geometry

The cutting tool used for making holes in solid material is called the twist drill. It basically consists of two parts; the body consisting of the cutting edges and the shank which is used for holding purpose. This has two cutting edges and two opposite spiral flutes cut into its surface as shown in Fig. 8.2. These flutes serve to provide clearance to the chips produced at the cutting edges. They also allow the cutting fluid to reach the cutting edges.

The drill blanks are made by forging and then twisted to provide the torsional rigidity. Then the flutes are machined and hardened before the final grinding of the geometry. Twist drill geometry is shown in Fig. 8.2. These are made with either straight or taper shank. Straight shank drills are held in the machine spindle in a drill chuck. The taper shank drills are directly held in the spindle with the help of the self-holding taper. The tang at the end of the taper shank fits into a slot in the spindle. The tang helps to drive the drill, prevents it from slipping and provides a means of removing it from spindle.

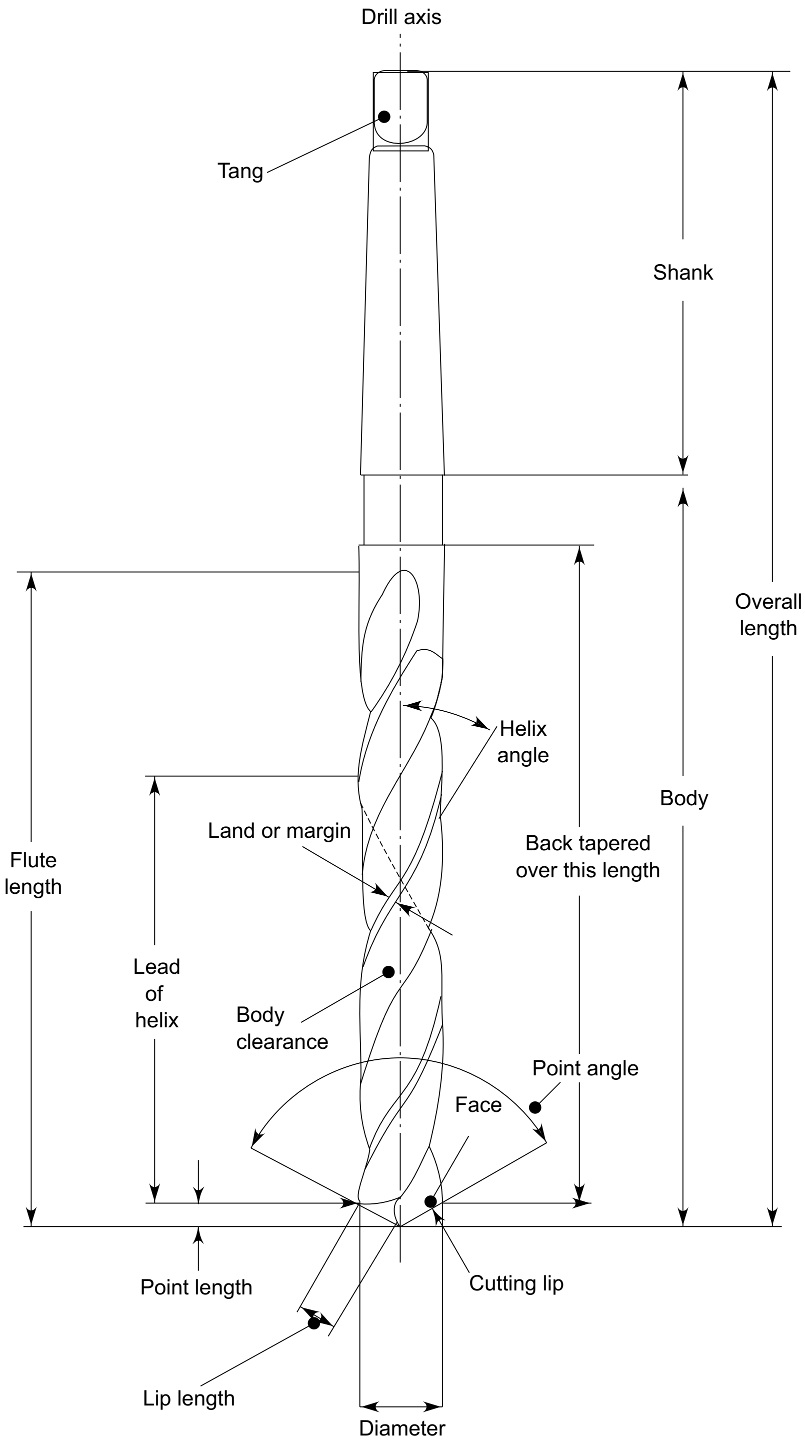


Fig. 8.2 *Twist drill geometry*

The surface on the drill, which extends behind the cutting lip to the following flute, is termed as flank. Face is the portion of the flute surface adjacent to the cutting lip on which the chip moves as it is cut from the work piece. The cutting lip is the edge formed by the intersection of the cutting edge or face and the flank face. Land or margin is the cylindrically ground body surface on the leading edge of the drill sometimes also termed as cylindrical land. The cutting edge is reduced in diameter after the margin to provide a body clearance.

Axial rake angle is the angle between the face and the line parallel to the drill axis. At the periphery of the drill, it is equivalent to the helix angle. Helix angle is the angle between the leading edge of the land and the axis of the drill. Sometimes it is also called spiral angle. The lip clearance angle is the angle formed by the portion of the flank adjacent to the land and a plane at right angles to the drill axis measured at the periphery of the drill. Lead of the helix is the distance measured parallel to the drill axis, between corresponding point on the leading edge of the land in one complete revolution.

The chisel edge is formed by the intersection of the two flanks. In order to provide strength to the drill the cutting edge is thickened gradually from the bottom. It is termed as web. Also a back taper is provided on the body towards the shank to provide longitudinal clearance.

The shape of the drill point is the most important. The lip angle should be correct for the given application. In general 118º is found to be suitable for mild steel and other general materials. Larger values are used for hard and brittle materials, while smaller values are used for soft materials. Some representative values are given in Table 8.2.

Table 8.2 **Lip angles for various work materials**

Work Material	Lip Angle, Degrees
Deep-hole drilling	128
Hard materials	136
Soft nonferrous materials	90
Hardened steel	125
Wood and nonmetals	60

The two cutting edges of the drill should be equal in length as well as the same angle with the drill axis. Otherwise, there will be unequal cutting forces along the cutting edges causing a torsional load. This will cause the drill to wear out quickly. Also the holes produced with such a drill tend to be oversized.

The rake angle in case of drilling is complex since it depends upon the helix angle of the flutes, the point angle and the feed rate. The various rake angles possible are shown in Fig. 8.3. Thus the thrust force is the maximum at the web where the material is compressed and extruded rather than sheared to the minimum value at the end of the cutting edge.

Since the web does not cut, sometimes direct drilling of large diameter holes makes it difficult to achieve the positional tolerance. For these situations a centre hole is made first with a centre drill or a small hole drill as a pilot hole. The size of the pilot hole drilled takes care of the web portion and thereby allows for more accurate location of the hole.

Twist drills are designed with the web, which gradually thickens as it moves from the point along the length of the flutes as shown in Fig. 8.4. This is necessary for providing strength and rigidity to the cutting tool. A twist drill is re-ground at the cutting lip to remove the worn out portion. This gradually decreases the total length of the drill. However, along with it the web thickness also increases as shown in Fig. 8.4. As explained above, the web will only compress the material and as a result, the thrust on the drill increases with an increase in the web thickness. Also, it is likely that out of round and over sized holes may result, because of the additional thrust.

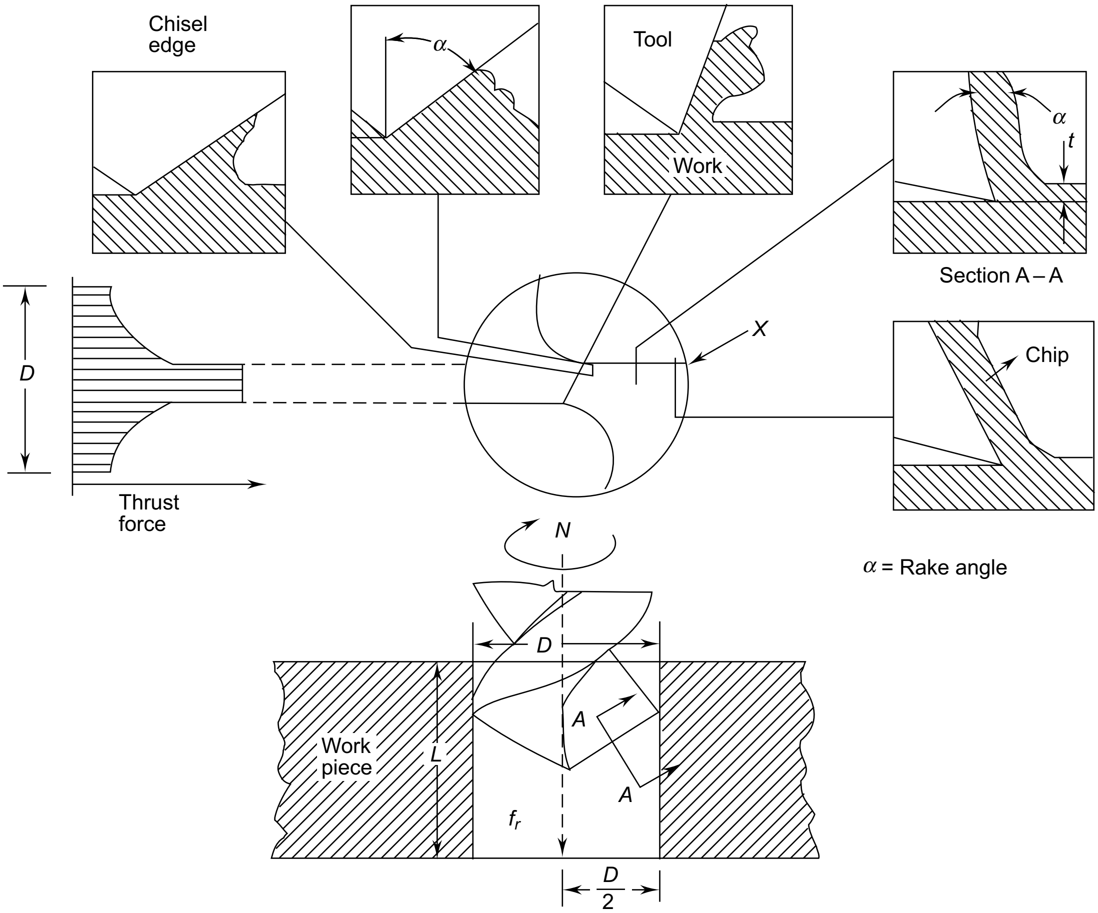


Fig. 8.3 *The rake angle in case of a twist drill*

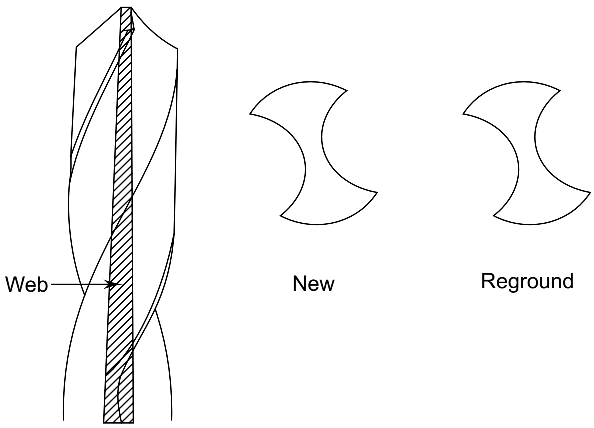


Fig. 8.4 *Change in thickness of the web due to the regrinding process*

After sometime, it therefore becomes necessary to thin the web. Web thinning has to be carefully done such that the thinning is blended evenly into the flutes. Further, the chisel edge should not be excessively reduced by carefully removing equal amounts of material from either side. Some of the ways in which the web thinning is done are shown in Fig. 8.5.

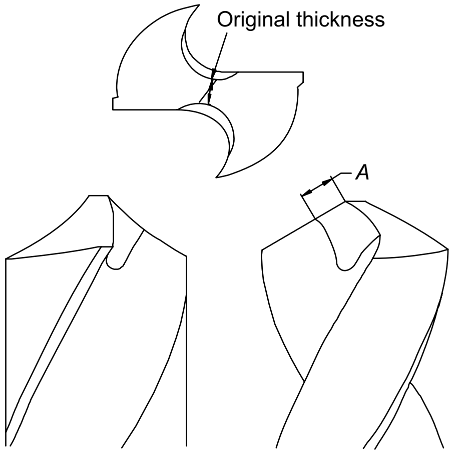


Fig. 8.5 *Thinning of the web by different methods*

8.2.2 Types of Drills

A large variety of drills are developed in addition to the standard twist drill, as detailed above, for specific applications.

Oil hole drills These are most useful for deep hole drilling. These are provided with two internal holes extending through the length of the drill through which the cutting fluid can be pumped under pressure. This keeps the cutting edge cool while flushing away the chips as well.

Step drills A variety of step drills are developed for combining machining of operations such as multiple hole drilling, counter boring and counter sinking.

Core drills These are special holes meant for enlarging already existing holes such as those in castings. These are either three-flute or four-flute type. The four-flute type is used for enlarging the drilled holes while the three-flute type is used for punched or cored holes. The three-flute type keeps the chatter to a minimum due to the fact that the cutting lips are not diametrically opposite.

Shell core drills These are similar to the core drills, but do not have a normal shank for the purpose of holding and are for the large diameters. This needs to be mounted using a stub arbour similar to the shell end mills with the help of the central hole present.

Spade drills Spade drills are used to make smaller diameter holes with low cutting speeds and high feed rates. These have a long supporting bar with the cutting blade attached at the end. These are less expensive since the support structure can be made more rigid using ordinary steel with no spiral flutes. Spade drills are also used to machine small conical shapes for subsequent drilling or making a bevel (similar to countersinking) on the existing holes to facilitate the subsequent tapping and assembling operations.

Carbide tipped drills Most of the drills are made of high speed steel. However, for machining hard materials as well as for large volume production, tungsten carbide tipped drills are available as shown in Fig. 8.6. The tungsten carbide tips of suitable geometry are clamped to the end of the tool to act as the cutting edges.

As explained earlier in Chapter 2, coatings provide a better alternative in improving the cutting tool life. This is more so in the case of a high speed steel drill. The titanium nitride (TiN) coating on the drills improves the drill tool life on an average by 2 to 10 times while drilling steel.

8.2.3 Drilling Machine Construction

In order to carry out the drilling operation, the motions required are the rotation of the drill while it is fed linearly into the work piece. Drilling machines come in a variety of shapes and sizes.

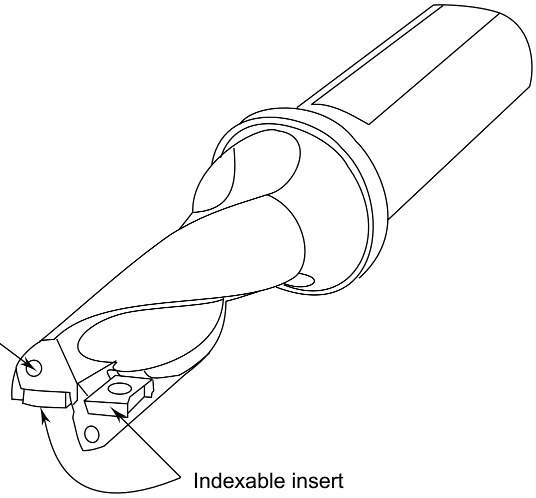


Fig. 8.6 *Drill with carbide inserts*

Drill Press

Typical drill press is shown in Fig. 8.7. The cutting tool in this case called the drill bit is mounted into the spindle either with the help of the drill chuck for small sized drills that are straight shank type, or by means of the spindle taper. The spindle is located inside a quill, which can reciprocate by means of manual operation or by means of power feed. The work piece is normally placed on the table and clamped using a suitable work holding device. These are relatively simple and less expensive in operation. However, these are not suitable for mass production.

Radial Drilling

The radial drilling machine is more versatile than the drill press as described earlier. The schematic of radial drilling machine showing the principal parts and motions is shown in Fig. 8.8. The drill head can move along the radial arm to any position while the radial arm itself can rotate on the column, thus reaching any position in the radial range of the machine. They are more convenient to be used for large work pieces, which cannot be moved easily because of their weight, such that the drill head itself will be moved to the actual location on the work piece, before carrying the drilling operation. In addition to the twist drills other hole making tools will also be used.

Multiple Spindle Drilling

For production operations, it is necessary to carry out a large number of operations simultaneously, which can be done through the multiple-spindle drilling machines. In the drilling heads of these machines more than one drill can be located with each of them getting the power from the same spindle motor. The use of these machines becomes more economical for large volume production of identical parts. These machines are capable of producing a large number of holes in a short time. Some machines have a fixed number of spindles in fixed locations while the others have the number fixed but their locations can be changed to suit the work piece geometry. The latter type of machines are more versatile.

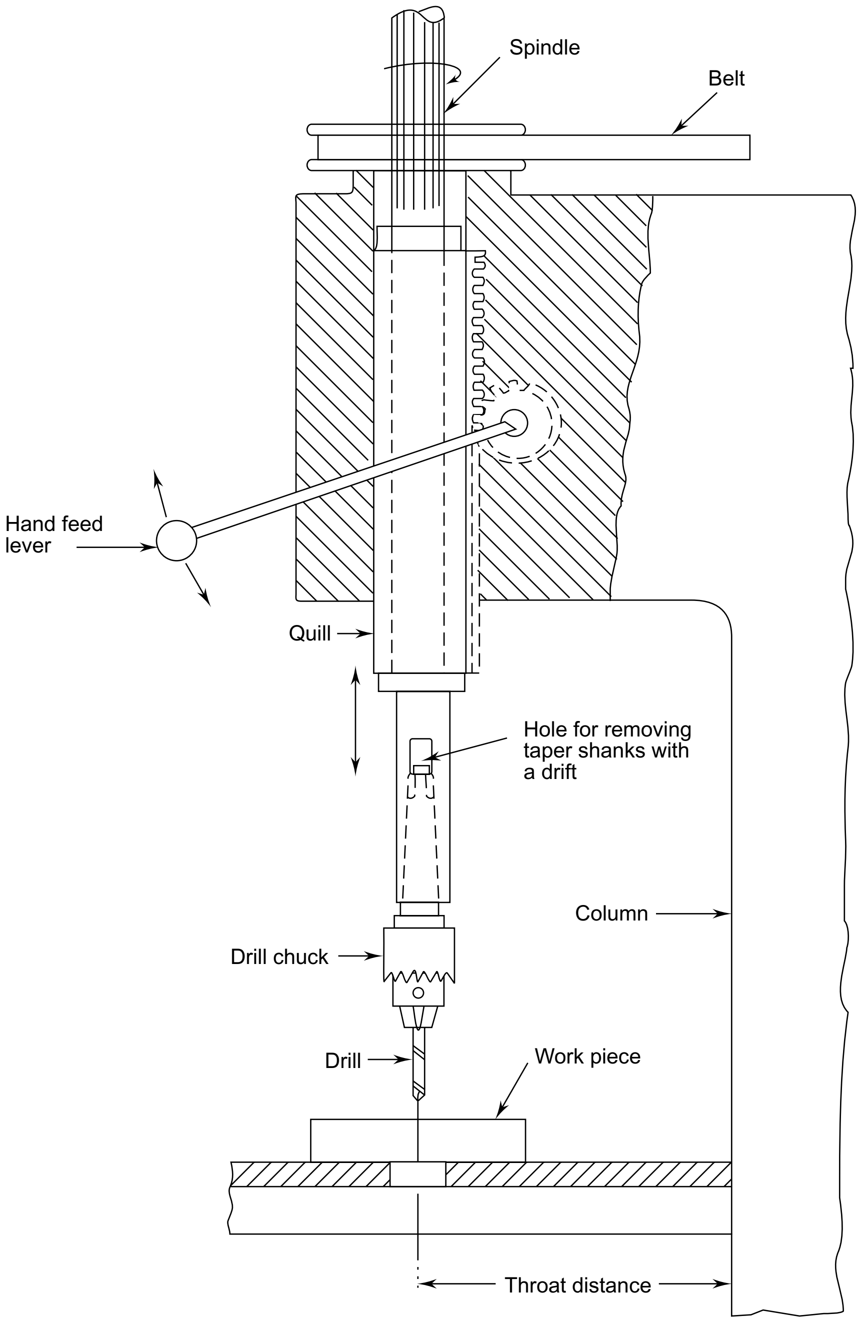


Fig. 8.7 *The drill press*

Gang Drilling

Gang drilling machines are the equivalent of the progressive action type multiple spindle lathes. These machines have a number of spindles (often equal to four) laid out in parallel. Each of the spindles can have different drills or other hole making operation tools fixed in sequence. The work piece will move from one station to the other, with each completing the designated hole making operation. These are used for volume production with the work pieces located in a jig, with reasonable size to allow the operator to move the part with jig to the next station generally on a roller conveyor.

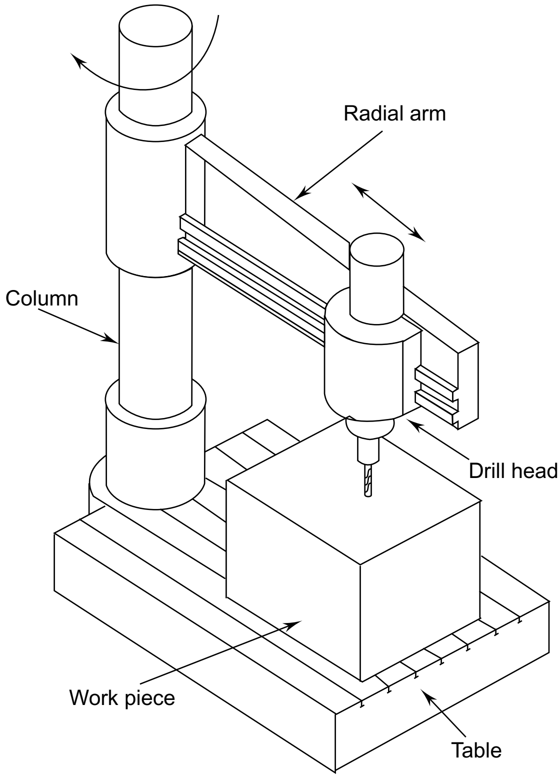


Fig. 8.8 *The radial drilling machine*

Work Holding

Work holding in drilling machines is similar to milling. Most of the small components are held in vices for drilling in job shops. However, for production operations, it is not only necessary to locate and clamp the work piece properly, but also to locate and guide the drill. Hence jigs are used to serve this function. An example of a drilling jig is shown in Fig. 8.9.

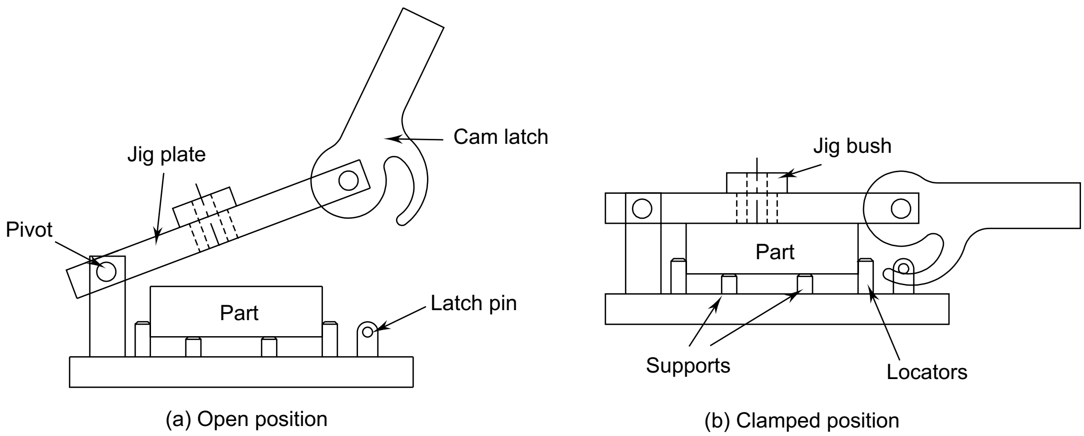


Fig. 8.9 *Drilling Jig for drilling a hole in a part*

8.2.4 Drilling Time Estimation

Typical process parameters used in drilling operation are given in Table 8.3, 8.4 and 8.5. The cutting speed in drilling is the surface speed of the twist drill. Thus,

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

Where, V = cutting speed (surface), m/min
D = diameter of the twist drill, mm
N = rotational speed of the drill, rev/min

Table 8.3 Cutting process parameters for drilling

Work Material	Hardness BHN	HSS	
		Speed m/min	Feed mm/rev
Cast iron	200	25–35	0.13–0.30
Cast steel	280–300	12–15	0.06–0.19
AISI 1020	110–160	35	0.20–0.50
AISI 1040	170–200	25	0.13–0.30
Manganese steel	185–215	5	0.06–0.19
Nickel steel	200–240	18	0.06–0.19
Stainless steel	150	15	0.13–0.30
Spring steel	400	6	0.06–0.19
Tool steel	150	23	0.20–0.50
Tool steel	200	18	0.13–0.30
Tool steel	215	15	0.13–0.30
Tool steel	300	12	0.06–0.19
Tool steel	400	5	0.06–0.19
Malleable iron	110–130	26	0.20–0.50
Aluminium	95	275	0.13–0.90
Aluminium alloys	170–190	18	0.13–0.30
Copper	80–85	21	0.06–0.19
Brass	190–200	70	0.20–0.50
Bronze	180–200	54	0.20–0.50
Zinc alloys	110–125	70	0.20–0.50
Glass		4.5	0.06–0.19

TABLE 8.4 **Cutting speeds for drilling using tungsten carbide tip drills**

Work Material	Cutting Speed m/min
Aluminium	50 to 150
Brass	50 to 100
Bronze	50 to 100
Cast iron soft	30 to 55
Cast iron chilled	10 to 15
Cast iron hard	30 to 45
Steel over 450 BHN	25 to 35

TABLE 8.5 **Feed rates for drilling using tungsten carbide tip drills**

Drill Size	Feed mm/rev
Up to 1.5 mm	0.010 to 0.025
1.5 to 3.0 mm	0.025 to 0.075
3 to 5 mm	0.05 to 0.10
6 to 8 mm	0.08 to 0.13
9 to 11 mm	0.10 to 0.20
12 to 18 mm	0.15 to 0.25
19 to 25 mm	0.20 to 0.30

The drill will have to approach the start of the hole from a distance and also traverse beyond the actual hole by a distance termed as the total approach allowance, A. The initial approach is generally a small value for positioning the drill above the hole. This distance, A_I can generally be taken as 2 to 3 mm. The traverse distance beyond the hole is often termed as the breakthrough distance and is required because of the conical shape of the twist drill as shown in Fig. 8.10. This value is dependent upon the drill diameter and the lip angle and is given by

Breakthrough distance, $A = \dfrac{D}{2 \tan \alpha}$

For the most common case of $\alpha = 59°$, it is given by

$$A = \frac{D}{3.3286}$$

Total length of tool travel, $L = l + A + 2$ mm

Where l = length of the hole, mm

Time for drilling the hole = $\dfrac{L}{fN}$ minutes

Where f = feed rate, mm/rev

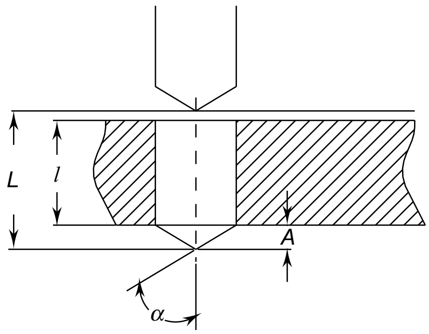


FIG. 8.10 *End of the twist drill showing the breakthrough distance*

The total volume of the material present in the hole indicates the material removal rate. In the case of a solid material without coring, the material removal rate MRR is given by the area of cross-section of the hole times the tool travel rate through the material. Thus,

$$\text{MRR} = \frac{\pi D^2 f N}{4}$$

Example 8.1

A hole of 40 mm diameter and 50 mm deep is to be drilled in mild steel component. The cutting speed can be taken as 65 m/min and the feed rate as 0.25 mm/rev. Calculate the machining time and the material removal rate.

Solution Given, $V = 65$ m/min

$f = 0.25$ mm/rev

$D = 40$ mm

$L = 50$ mm

Spindle speed, $$N = \frac{1000 \times 65}{\pi \times 40} = 517.25 \text{ rev/min}$$

$$= 520 \text{ rev/min}$$

Breakthrough distance, $$A = \frac{40}{2 \tan 59} = 12.02 \text{ mm}$$

Total length of drill travel, $L = 50 + 12 + 3 = 65$ mm

Time for drilling the hole = $$\frac{65}{0.25 \times 520} = 0.50 \text{ minutes}$$

The material removal rate is MRR = $$\frac{\pi \times 40^2 \times 0.25 \times 520}{4} = 163362.82 \text{ mm}^3\text{/min}$$

$$= 163.363 \text{ cm}^3\text{/min}$$

8.2.5 Drilling Force Estimation

The two major forces acting on the drill during the machining operation are the torque and the thrust. The torque acting on a twist drill is given by

$$M = C\, d^{1.9} f^{0.8} \text{ N mm}$$

Where d is the diameter of the drill in mm

F is the feed rate of the drill in mm/rev

And C is a constant whose values are given in Table 8.6.

The thrust force is given by

$$T = Kdf 0.7 \text{ Newtons}$$

The values of K are given as

Steel = 84.7

Cast iron = 60.5

Table 8.6 The constant *C* for torque calculations

Material	Hardness, BHN	Constant, *C*
Steel	200	616
	300	795
	400	872
Aluminium alloys		180
Magnesium alloys		103
Brasses		359

Example 8.2

Estimate the force and thrust for a 20 mm drill with a feed of 0.2 mm/rev rotating at 150 rev/min, while drilling mild steel.

Solution From the table $C = 616$

Torque acting is $M = 616 \times 20^{1.9} \times 0.2^{0.8} = 50392.0$ N.mm

$= 50.392$ Nm

Thrust is $T = K\,d\,f^{0.7} = 84.7 \times 20 \times 0.2^{0.7} = 549.1$ Newtons

The tolerance on the achieved dimension in drilling depends upon the geometry of the drill to a great extent, but also on the diameter of the drill. Typical tolerances that can be achieved in drilling with a properly ground drills is given in Table 8.7.

Table 8.7 Limits of tolerance on drilling

Diameter, mm	Limits of Tolerance, mm	
	High (+)	Low (–)
Up to 3	0	0.014
3 to 6	0	0.018
6 to 10	0	0.022
10 to 18	0	0.027
18 to 30	0	0.033
30 to 50	0	0.039
50 to 80	0	0.046
80 to 120	0	0.054

Some common problems that are found in the case of drilling are shown in the following Table 8.8.

Table 8.8 Common problems in drilling

Drilling Problem	Probable Cause and its Remedy
Chipping of cutting lips	High feed rate High lip relief angle
Breaking of drill	Drill point improperly ground High feed rate Drill needs regrinding Flutes clogged with chips
Drill chatter	Loose moving parts of the machine Reduce overhang Fixture not properly clamped
Hole over size	Unequal angle of cutting edges Loose spindle Unequal lengths of cutting edges
Rough hole	Point improperly ground High feed rate Fixture not rigid No cutting fluid
Drill will not enter the work	Drill needs regrinding The drill web is too thick Lip relief is too small

8.2.6 Deep Hole Drilling

The deep hole is normally classified as hole, which is longer than three times its diameter. Since the depth is longer, it creates some problems. The rigidity of the drill becomes low because of the extra length required. Further the chip space becomes small compared to the amount of chips generated. The cutting speed and feed used for deep hole drilling should be reduced. The cutting speed for deep holes, V_{deep} is given by

$$V_{\text{deep}} = V_{\text{drill}} \times \left[1 - \frac{\text{Depth of hole}}{40 \times \text{Dia. of hole}}\right]$$

Similarly the feed rate is also reduced as follows:

$$f_{\text{deep}} = f_{\text{drill}} \times \left[1 - \frac{\text{Depth of hole}}{50 \times \text{Dia. of hole}}\right]$$

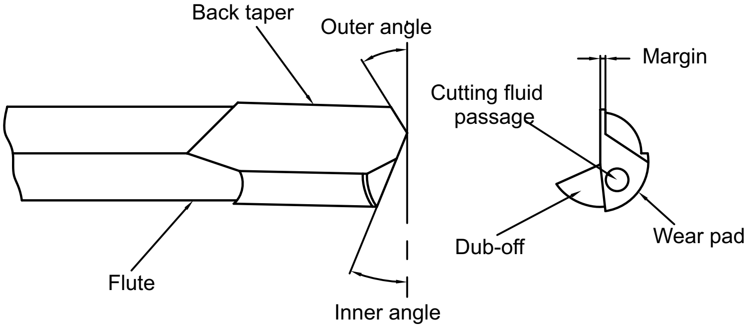
The major problem with deep hole drilling is total volume of chips generated. The flute space will be simply jammed if no proper care is taken to prevent it. One of the methods to be adopted is the withdrawal of the drill from the hole at periodic intervals such that the drill can be flushed with the cutting fluid. The recommended method is that the first drill to be used is the normal drill of short length and completes the hole up to 3 to 4 times the diameter. Later change to an extra long drill and continue drilling with the care to see that the drill is withdrawn from the hole every 1 to 2 times the diameter for really deep holes. This is sometimes called as peck drilling (similar to the wood pecker action of boring the tree trunk). The problem gets further compounded when the work piece material generates ductile and continuous chips. In such cases, it is necessary to have some form of chip breaking arrangement. One common method used is to grind grooves in the flank of the point.

Very small hole drilling such as those with diameters ranging from 0.4 to 1.0 mm will always be classified as deep hole drilling because of the length of the hole involved. To maintain the strength of these drills, the web is generally thicker than that for the normal size drills. As a result the cutting conditions have to be carefully adjusted to get a reasonable tool life.

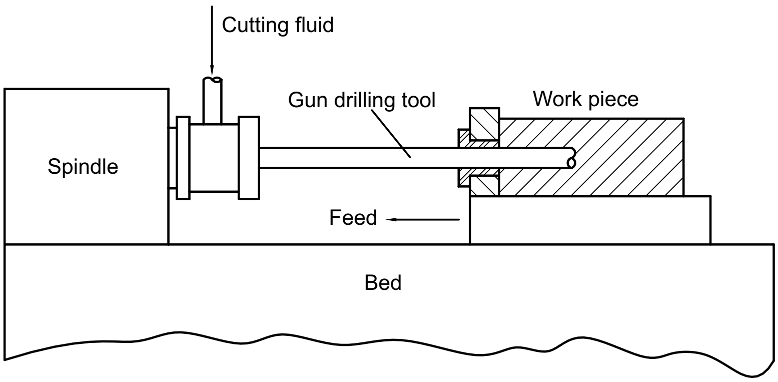
8.2.7 Gun Drilling

Gun drilling is the process of drilling extremely long or deep holes. It was originally developed for making the gun barrels from which it derives its name. This process is widely applied to produce precision long and short holes up to 200 mm dia and 30 m long. A typical gun drill consists of a hollow tube with a *V* shaped groove or flute along its length with a carbide cutting tip which also acts as its own guide bushing as it drills the hole as shown in Fig. 8.11(a). A gun drill can produce holes up to 100 times its diameter. The success in gun drilling is achieved by choosing suitable speeds and feeds which facilitate proper chip management and adequate coolant pressure for lubrication of the brazed carbide tip along with the chip evacuation through the drill's single flute. High-pressure oil or water-soluble coolant flows through an internal coolant passage of the gun drill to the cutting area.

Cutting fluid forces the chips formed by the cutting action up the flute and out of the hole. The high-pressure (140 MPa or higher) coolant system includes chip separation and filtration units to maintain clean coolant. If not done properly chips in the coolant directed to the tool will impair its function and clog small drills. Drilling action takes place off the centre point, and as a result considerable pressure acts on the wear pad that is used to help guide the drill and create a burnishing effect as the hole is drilled. Typical surface finish of a gun drilled hole is from 0.8 to 1.5 μm. Rotate the tool instead of the work wherever feasible. Gun drilling machines in which the work rotates are more sensitive to misalignment than those in which the tool is rotated. Gun-drilling machines should have the highest rigidity to provide accurate alignment.



(a) Gun-drilling tool



(b) Schematic of a gun-drilling machine

Fig. 8.11 *Gun drilling operation, (a) Gun drilling tool, (b) Schematic of a gun drilling machine*

8.2.8 Trepanning

Trepanning produces a hole in solid material by directly cutting the circumference of the hole and not the full hole. After the trepanning operation the core at the centre of the hole which forms bulk of the material is left intact and can be re-used for other purpose. There are two forms of trepanning: one type used for thin sheets while the other is used on much heavier material.

The tool used for thin sheet trepanning consists of a single point tool that is fixed to an adjustable arm that revolves around a pointed centre shank as shown in Fig. 8.12. The centre locates the tool to generate the required geometry and also keeps it rigid. This operation is very similar to drawing circles using a compass for engineering drawing. Using this tool disks up to 6 mm thick and 150 mm in diameter can be machined. The main advantage of the process is that the centre disk can be re-used for other purpose instead of converting into chips as in solid drilling.

To make deep and large holes (greater than or equal to 50 mm in diameter), another trepanning tool has to be used. This process is similar to gun drilling where once the cutting starts forced lubrication and cooling is used. This process is faster (10 to 15% compared to solid drilling) with very few chips produced and gives a straighter bore because the tool is self-truing. Straightness and diametric tolerances are almost comparable to gun drilled holes. Since the central portion of the bore is not converted into chips, it can be salvaged which is an advantage in case of expensive alloys.

The trepanning tool is essentially a tube with one end fixed with a shank and the other end consists of a tool bit wider than the width of the tool to provide the clearance. Similar to gun drilling cutting fluid is introduced around the tool and the chip and spent fluid evacuates through the tool's centre. The cutting fluid flow rate increases as the tool advances into the work material. For re-circulating the cutting fluid it has to be filtered to remove all the chips with a fine mesh and cooled to improve the efficiency of machining. For this purpose the coolant tank is larger than that used in conventional machine tools. This process can deliver tolerance on the hole diameter up to ±0.050 mm and surface finishes from 2 to 3 μm.

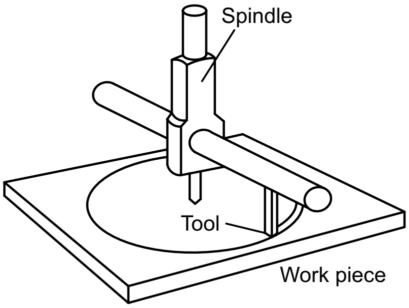


Fig. 8.12 *Thin sheet trepanning operation*

8.3 REAMING

Reamer is a multi-tooth cutter, which rotates and moves linearly into an already existing hole. The previous operation could be drilling or preferably boring. Reaming will provide smooth surface as well as close tolerance on the diameter of the hole. Generally the reamer follows the already existing hole and therefore will not be able to correct the hole misalignment. Typical geometry of a reamer along with its nomenclature is shown in Fig. 8.13. Typical limits of tolerances that can be obtained by reaming are given in Table 8.9, which can be used for process selection purpose.

Table 8.9 Limits of tolerance on reaming

Diameter, mm	Limits of Tolerance, mm	
	High (+)	Low (–)
1 to 3	0.009	0.002
3 to 6	0.012	0.004
6 to 10	0.015	0.006
10 to 18	0.018	0.007
18 to 30	0.021	0.008
30 to 50	0.025	0.009
50 to 80	0.030	0.011
80 to 120	0.035	0.013

A reamer is more like a form tool, since the cylindrical shape and size of the reamer is reproduced in the hole. In reaming very little material is removed. The normal reaming allowances are shown in Table 8.10. At the bottom of the reamer the flutes (cutting edges) are made slightly tapered to facilitate its entry into the existing hole. Generally the reamer is expected to cut from the sides and not from the end. These are most suitable for reaming through holes. However for reaming blind holes with flat bottom, special end cutting reamers that look similar to end mills will have to be used. They have the cutting edges also formed in the end.

Table 8.10 Reaming allowances

Size of Reamed Hole, mm	Stock Allowance, mm	
	When Predrilled	When Precore Drilled
Up to 10	0.30	0.20
10 to 14	0.40	0.25
14 to 18	0.50	0.25
18 to 30	0.50	0.30
30 to 50	1.00	0.40

The reamer flutes are either straight or helical. The helical flutes promote smoother cutting and should be used specifically for the holes that are not continuous such as those with keyways parallel to the axis of the hole. The cutting action of the helical flutes is smoother and helps in preventing chatter, which is likely in view of the large area of contact between the tool and the work piece.

The reamers are termed as left hand or right, depending upon the direction in which they are moved, starting from the shank to the cutting portion. Also the reamer as it is rotated to the right advances towards

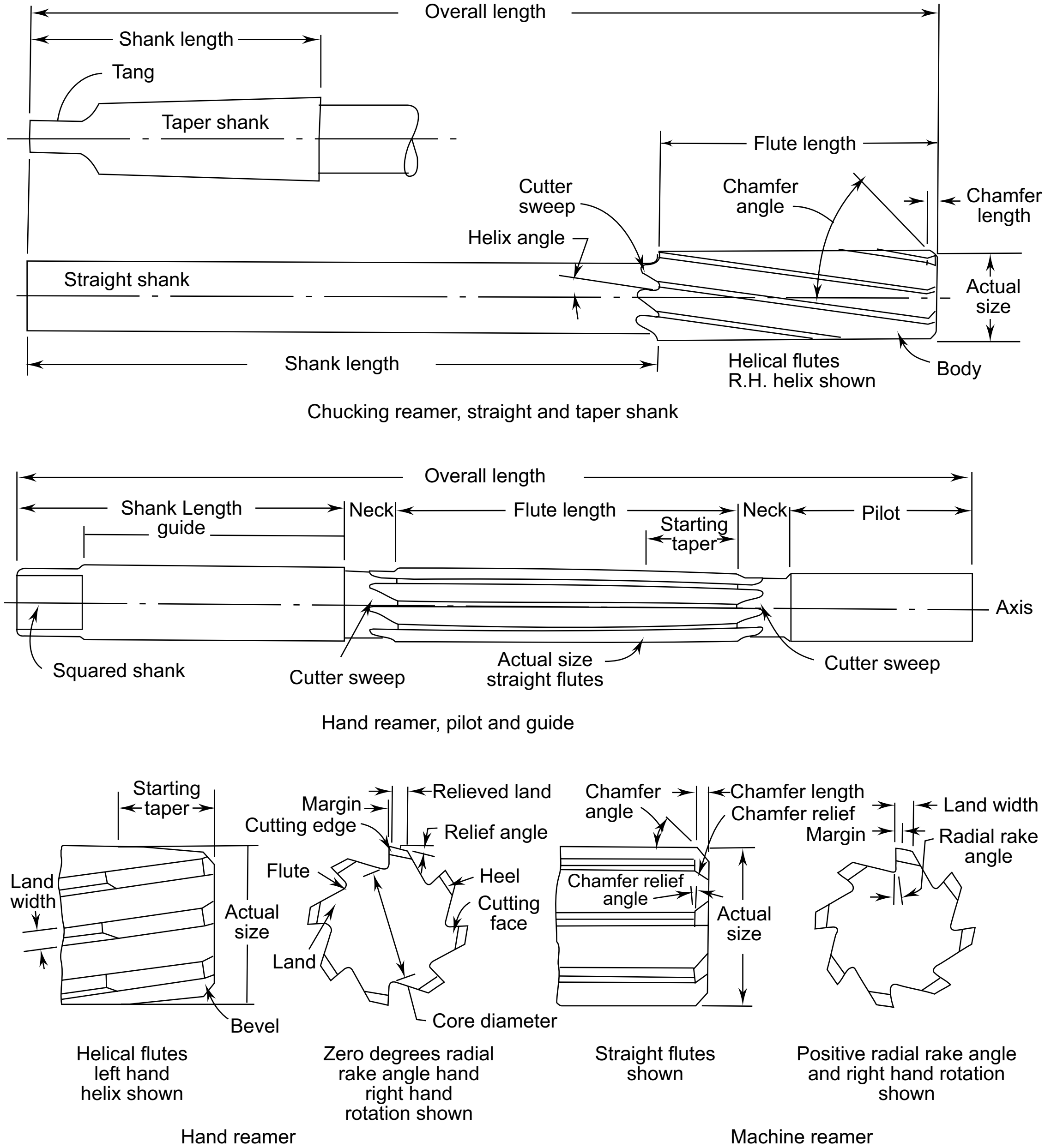


Fig. 8.13 *Typical geometry of a reamer and its nomenclature*

the cutting portion and is termed as the right hand helix. The reverse is termed as the left hand helix. The right hand reamer with right hand helix is used for roughing cuts, since the tool tends to go into the work piece more efficiently and thereby promote the material removal. Similarly a right hand reamer with left-hand flutes is used for finishing cuts.

Since the reamer follows the already existing hole, any misalignment present in the hole is likely to break the reamer if mounted in the conventional spindle. Hence floating reamer holder is used between the machine spindle and the reamer to adjust for any small misalignment between the spindle axis and hole axis.

The cutting speeds used in reaming are relatively small and the feeds generally large compared to an equivalent drilling operation to assure the required surface finish. Suggested cutting speeds are given in Table 8.11.

Since reaming is used essentially to achieve a good surface finish along with high dimensional tolerance, use of cutting fluids is essential. Suggested cutting fluids for different work materials are given in Table 8.12.

Table 8.11 Cutting speeds for reaming

Work Material	Cutting Speed, m/min
Aluminium and its alloys	45–70
Brass	45–70
Bronze	15–20
Cast iron, soft	20–35
Cast iron, hard	15–20
Steel low carbon	15–20
Steel medium carbon	12–15
Steel high carbon	10–12
Steel alloy	10–12
Stainless steel	5–20

Table 8.12 Cutting fluids for reaming

Work Material	Dry	Soluble Oil	Kerosene	Sulphurised Oil	Mineral Oil
Aluminium		X	X		
Brass	X	X			
Bronze	X	X			X
Cast iron	X				
Steel Low carbon		X		X	
Steel Alloy		X		X	
Stainless steel		X		X	

Chatter is often caused in reaming due to the lack of rigidity in the operation. To reduce the chatter, reduce the cutting speed, increase the feed rate, add a chamfer to the hole being reamed to facilitate the easier entrance of the reamer, or use a reamer with a pilot. The work piece should be properly supported during the reaming operation, otherwise over sized holes will be produced. If the reamer axis is misaligned with the axis of the hole, a bell mouthed hole will result.

8.4 BORING

Boring is an operation of enlarging a hole. The single point cutting tool used for the boring operation is shown in Fig. 8.14. Generally the single point tool bit is mounted in the boring bar of suitable diameter commensurate with the diameter to be bored. The overhang of the tool is to be maintained as small as possible to reduce the chatter, which is very common in boring.

It is possible to carry out the boring operation in a lathe for limited applications while in drilling and milling machines a large range of holes can be bored

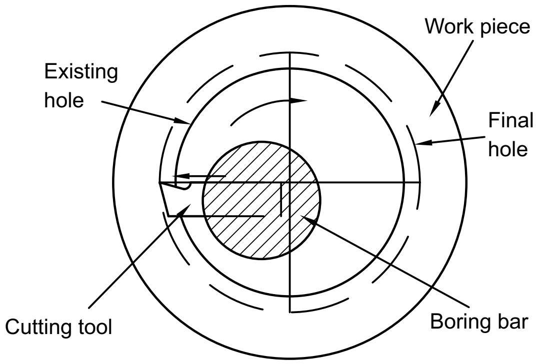


Fig. 8.14 *Boring with a single point turning tools in a boring machine*

using the multiple point cutting tools in addition to the single point tool as explained earlier. Boring with a single point cutting tool being a semi-finishing operation very small amount of stock is left out to be removed. Typical boring allowances are given in Table 8.13.

Table 8.13 Machining allowance in boring

Hole Diameter, mm	Machining Allowance, mm	
	Rough Boring	Semi Finish Boring
Up to 18	0.8	0.5
18 to 30	1.2	0.8
30 to 50	1.5	1.0
50 to 80	2.0	1.0
80 to 120	2.0	1.3
120 to 180	2.0	1.5

Major problem of boring with a boring bar with single point turning tool is lack of rigidity of the boring bar. The size of boring bar is dictated by the size of the hole to be bored, while its length depends upon the geometry of the bore. Typically up to a length of boring bar equal to 5 times the diameter of the bar, simple boring bars would be able to serve the purpose. However beyond this length, chatter becomes predominant reducing the finish of the bore produced. It is necessary to use special damped boring bars for boring above this range.

8.4.1 Horizontal Boring Machine

The work piece where a hole is already present is mounted on the table of a horizontal-boring machine. Generally heavy work pieces that are difficult to be rotated in the drilling machines, are bored in horizontal boring machines. A drilled hole, which is not properly located, can be made concentric with the axis of rotation of the spindle by the boring operation.

Horizontal boring machine or boring mill comes in a variety of configurations. The basic units that are present in any boring machine are:

- **Headstock:** It is the heart of the equipment that houses all the spindles with speed and feed arrangement. Different spindles are available depending upon the power and speed requirements of the different operations planned. The headstock also provides a station on which other attachments can be mounted. The spindle feed and hand feed controls are contained in the headstock.
- **Column:** The headstock is supported by the column and guides it up and down accurately by means of guideways. Columns are attached to bases. Some columns are stationary while others move with their bases.
- **Column base:** This base supports and secures the column. It houses the various gear and driving mechanisms.
- **End-support column:** An additional column is sometimes required for operations that involve the use of long boring bars or heavy tools. An outboard bearing in this column is utilized to support the end of the bar. This is also called backrest.
- **Runways:** These are used to carry the main column and end-support column in some machines.
- **Table and saddle:** It is provided for locating and clamping work piece. The table has accurate guideways to move the table in two perpendicular directions (X and Y in horizontal plane).
- **Bed:** Similar to the bed for heavy machine tools to support most of the machine tool parts.

- **Floor plate:** Similar to a stationary bed and is provided with T-slots for mounting work piece.
- **Cross rail:** The cross rail is similar to the cross rail of a planer.

Table-type horizontal boring machine (Fig. 8.15): This is the most common boring machine seen in the machine shops. As can be seen in Fig. 8.15 it has a headstock, column, column base, table, saddle, end support, and bed.

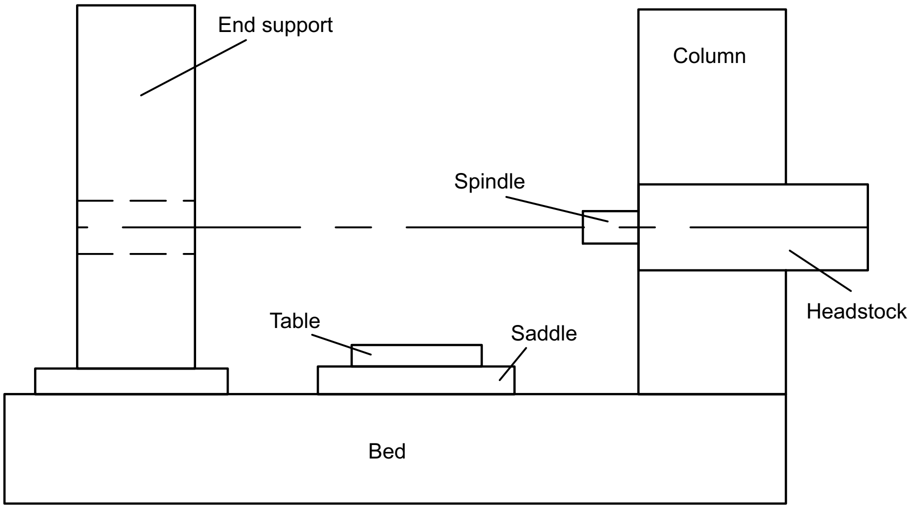


Fig. 8.15 *Schematic of a table type Horizontal Boring machine*

Floor-type horizontal boring machine (Fig. 8.16): This is normally used for very large work pieces. It has a headstock, column, column base, runway, end support, end-support runway, and floor plate to accommodate large work pieces. Because of the heavy nature of the work pieces machined on this machine they remain stationary while the spindle and its support are traversed along the runway past the work.

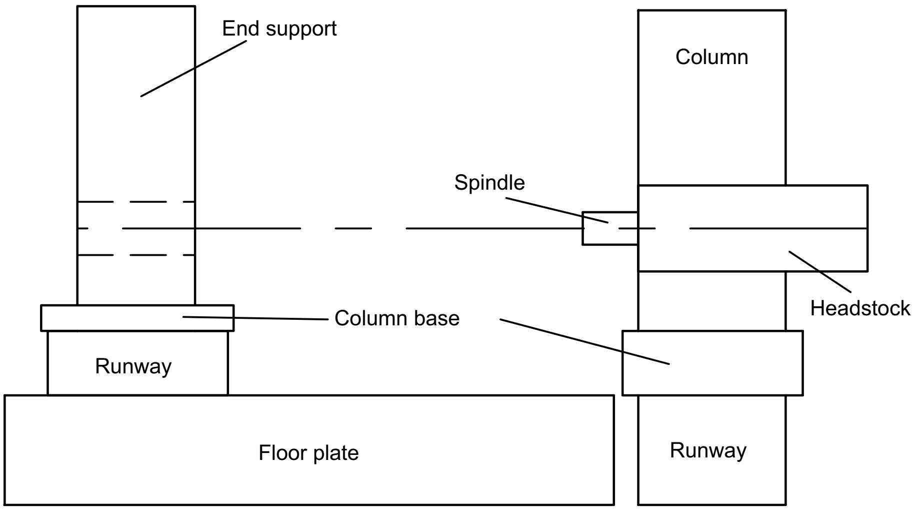


Fig. 8.16 *Schematic of a floor-type Horizontal Boring machine*

Planer-type horizontal boring machine (Fig. 8.17): The table in this machine is similar to a planer with reciprocating table. It has a headstock, column, runway, end support, end-support run-way, table, and bed. Guideways for reciprocating are provided in the bed. This machine is used for long work where exceptional rigidity is required. The tool remains stationary while the work moves past the tool.

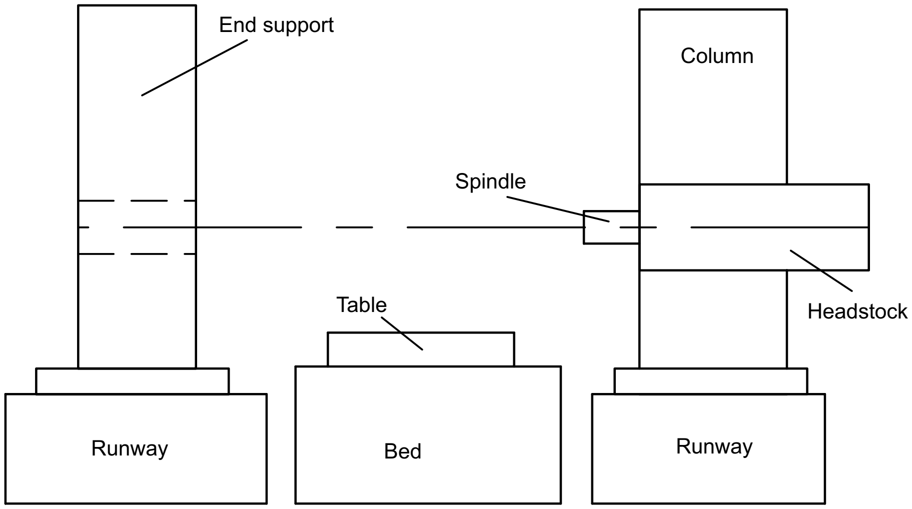


Fig. 8.17 *Schematic of a planer type Horizontal Boring machine*

Multiple-head-type horizontal boring machine (Fig. 8.18): This type of machine has multiple units of headstock (two to four), two columns with a cross rail, bed, and table. This machine looks and works like a planer-miller described in chapter 6. All the headstocks have the capability of swivelling for angular cuts. Because of the way the headstocks are positioned it can do both vertical and horizontal boring, and milling.

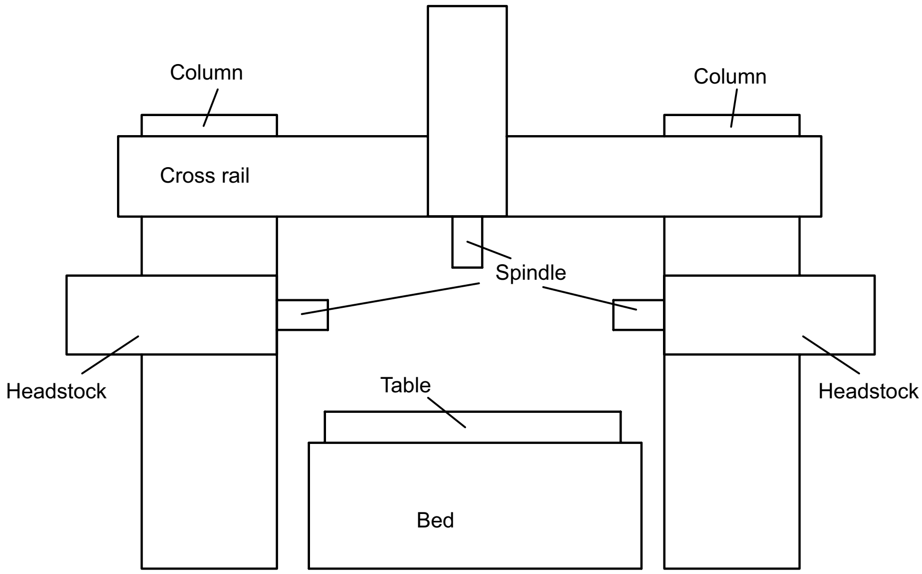


Fig. 8.18 *Schematic of a Multiple-head-type Horizontal Boring machine*

Tools Used

On horizontal boring machines practically all hole making operations such as reaming, counter boring, and tapping can be done in addition to boring. Regular boring bars as well as stub boring bars are used depending upon the hole geometry to be bored. Work always remains stationary while the tool is rotated to remove the metal. For enlarging holes which are close to the column offset boring bars are used. The offset boring bars are adjustable by means of a dovetail slide, and are provided with a fine graduate scale. The tool can be moved outward by a precise amount and clamped in any position depending upon the diameter to be bored. To reduce the machining time, regular boring bars have several slots to add multiple single point tools in sequence. These can be used for roughing, semi-finish and finish tools all in a single setup. Also all types of milling cutters can be used.

8.4.2 Vertical Boring Machine

Vertical boring mill (Fig. 8.19) is the largest machine tool. It has a number of tool heads (up to four), two columns, cross rail similar to a planer, table (rotary), and bed. It is used to machine internal and external diameters as well as to face large work pieces that are axi-symmetrical. A large rotary table is used to clamp the work piece and is rotated against a fixed tool, so only circular cuts can be made. The rotary table is mounted on the bed. Any size of the work piece that is within the size of the rotary table can be machined. Really large machines of this category are constructed with their tables flush with the floor. If necessary these tool heads can be arranged for angular cuts.

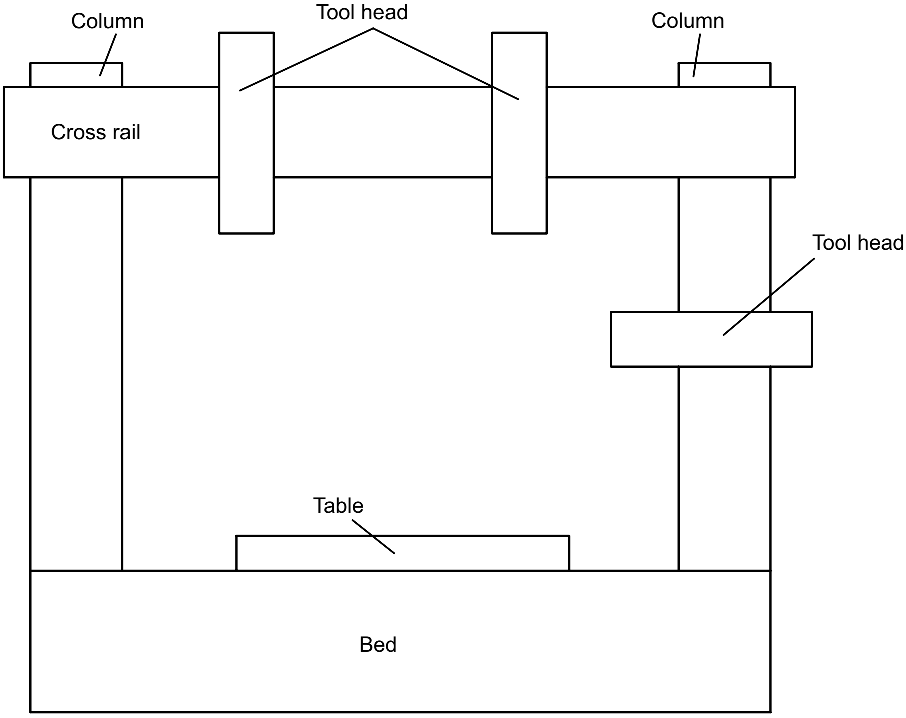


Fig. 8.19 *Schematic of a Vertical Boring machine*

8.4.3 Vertical Turret Lathe

The vertical turret lathe (Fig. 8.20) is similar to vertical boring machine except it is much smaller in size. The types of jobs that can be done in a vertical turret lathe are similar to those of its larger cousin but much smaller in size. It looks almost like a turret lathe (Chapter 5) rotated 90° from the horizontal, in such a way that its headstock is sitting on the floor and its axis is vertical.

The work piece is clamped to the table with jaws and is normally called a chuck. The tool holder is a five-sided turret mounted on the cross rail. It has five tool positions all of which can be used to machine a work piece in a single setup. Also it is possible that any of these five positions can support multiple tooling, in which up to 10 tools can be mounted, so that a work piece can be completely machined without changing any tools in the turret.

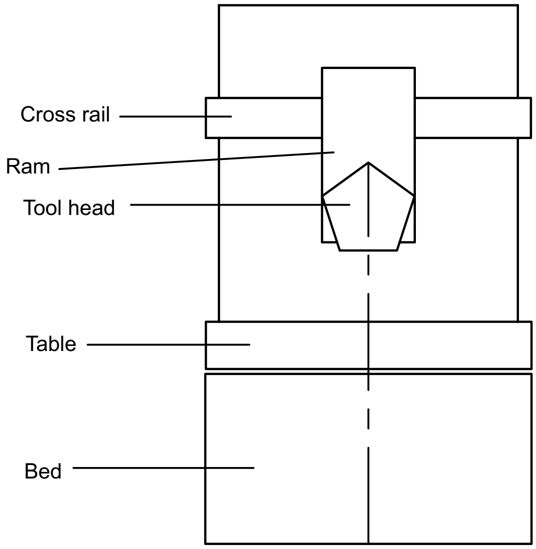


Fig. 8.20 *Schematic of a Vertical Turret Lathe*

8.4.4 Jig Boring Machine

A jig boring machine (Fig. 8.21) is similar to a vertical axis milling machine (Chapter 7) with the table closer to the floor and built with heavy, rigid and accurate construction. Table and saddle are mounted on the rigid bed that provide the necessary two axes (X and Y) movement for the work piece. A massive column supports all the necessary drives to provide power to the spindle. To provide the Z-axis movement the spindle moves inside a quill which is rigidly supported by the spindle head. The quill can also move vertically thereby providing a telescoping movement with added rigidity to extend the range of Z-axis movements for the tool. All the mechanisms involved in the three axes movement are manufactured under extremely careful and exacting conditions so that there will be no motion lost during tool movement. The materials for the housing are also carefully chosen so that thermal expansions due to the changes in temperature are minimized. The jig boring machine must be rugged enough for heavy cuts while remaining sensitive enough for light cuts. Accurate system of measurement is provided in jig boring machines for locating the tool positions precisely.

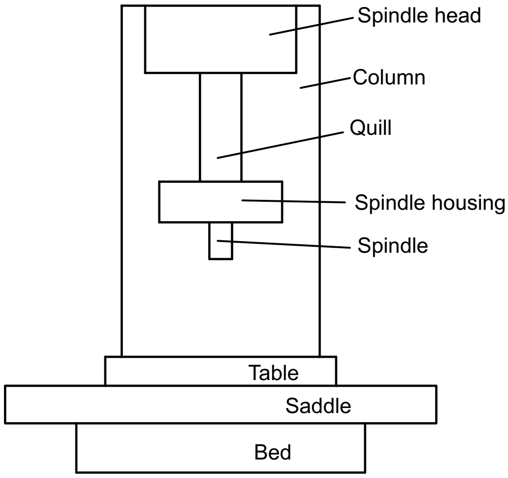


Fig. 8.21 *Schematic of a Jig Boring Machine*

8.5 TAPPING

In chapter 4 we discussed the machining of screw threads using a single point tool in a lathe. A faster way of producing internal holes is by the use of tapping operation. A tap is a multi-fluted cutting tool with cutting edges on each blade resembling the shape of threads to be cut. A tap of the required size is to be used after carrying out the pre-drilling operations. The tapping drill sizes for ISO metric threads are given in Table 8.15. Similar tables can be seen for other forms of threads from the handbooks. Important features of the taps are shown in Fig. 8.22.

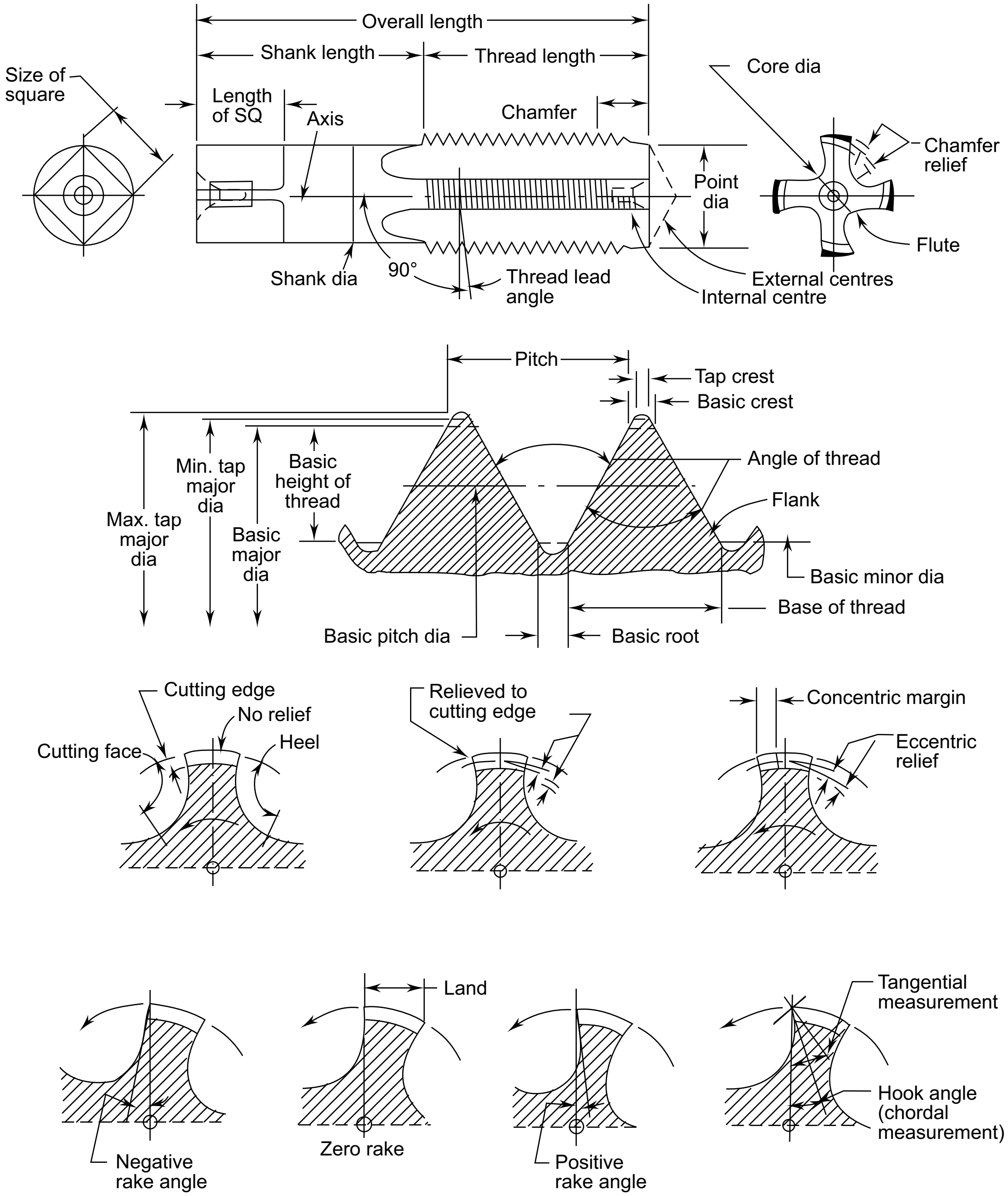


Fig. 8.22 *Important features of a tap*

Table 8.14 Recommended drill sizes for ISO metric threads

Nominal Tap Size, mm	Lead, mm	Tapping Drill Size, mm	Clearance Drill Size, mm
1	0.25	0.75	1.05
2	0.40	1.60	2.05
3	0.50	2.50	3.10
4	0.70	3.30	4.10
5	0.80	4.20	5.10
6	1.00	5.00	6.10
7	1.00	6.00	7.20
8	1.25	6.80	8.20
9	1.25	7.80	9.20
10	1.50	8.50	10.20
11	1.50	9.50	11.20
12	1.75	10.20	12.20
14	2.00	12.00	14.25
16	2.00	14.00	16.25
18	2.50	15.50	18.25
20	2.50	17.50	20.25
22	2.50	19.50	22.25
24	3.00	21.00	24.25
27	3.00	24.00	27.25
30	3.50	26.50	30.50

The cutting edges being of the same shape as that of the thread to be machined, the helical angle forces the chips to be moved ahead through the flutes. The flutes therefore are straight increasing the strength of the tap.

Tap is basically a form tool and therefore care has to be taken while re-grinding to maintain the form as well as dimensions. The leading end of the tap is tapered as shown in Fig. 8.15 to help in starting the tap and to distribute the majority of the cutting action over a number of threads. This type of tap is used for tapping through holes only. For blind holes requiring a flat bottom, special bottoming taps without taper have to be used.

While tapping, care has to be taken to see that the tap is started in proper alignment with the hole. Once started, the tap is automatically drawn into the hole by the threads and hence it should not be forced in. Sometimes it may become necessary to reverse the tap slightly to break the chips and clear the chip space and then continue in the normal way. Use of copious quantity of cutting fluid is essential since tapping is a heavy and slow material removal operation.

Table 8.15 Recommended cutting speeds to be used in tapping

Material	Tapping Speed, m/min
Aluminium	15 to 50
Brass	15 to 75
Bronze	15 to 35
Cast iron	15 to 35
Copper	10 to 20
Magnesium	25 to 65
Malleable iron	10 to 20
Steel free machining	13 to 25
Stainless steel	2 to 10
Zinc	20 to 50

Tapping speeds used are generally smaller compared to drilling in view of the large feeds that are used. Typical tapping speeds are given in Table 8.15. These are considered as the starting values, and if any problem is noticed during the tapping operation, they need to be adjusted based on the problem. The speed need to be lowered as the length of the hole increases because of the problem of the chip removal and their potential ability to get welded to the tap, thereby reducing its cutting ability.

For machine tapping the tap should be held in a floating holder to allow for sufficient flexibility for the tap to follow the existing hole. The tap should be rotated in the reverse direction to extract it from the hole after completing the tapping operation.

8.6 OTHER HOLE MAKING OPERATIONS

Counter Boring

Already existing holes in the components can be further machined for counter boring as shown in Fig. 8.23. Counter boring is done on surfaces that are rough, uneven and not evenly perpendicular to the axis of the hole. The counter boring can be done by a tool similar to the shell end mill (Fig. 8.24) with the cutting edges present along the side as well as the end, while a pilot portion is present for the tool to enter the already machined hole to provide the concentricity with the hole. The pilot should fit snugly in the hole and should have sufficient clearance facilitating the free movement of the tool. In the counter boring operation, the hole is enlarged with a flat bottom to provide proper seating for the bolt head or a nut, which will be flush from the outer surface.

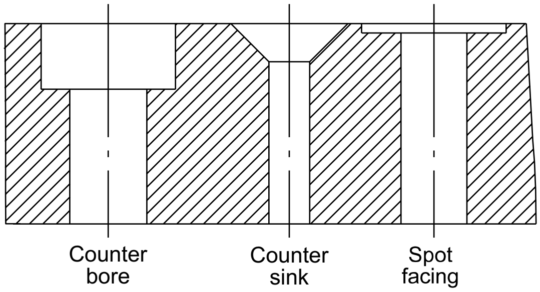


Fig. 8.23 *Other hole making operations*

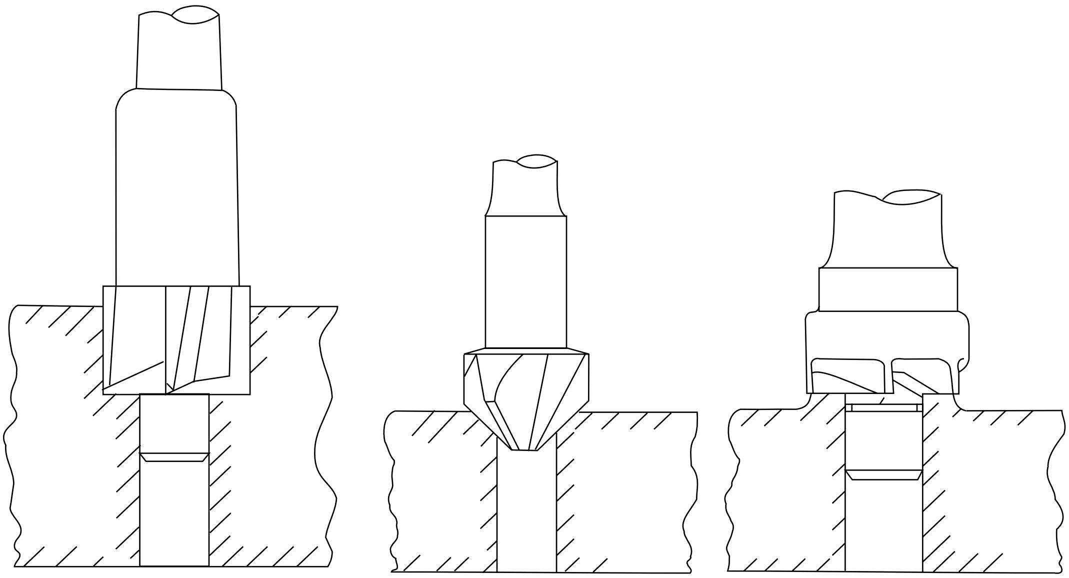


Fig. 8.24 *Tools used for other hole making operations*

Generally the speeds and feeds used for counter boring are slightly smaller than those used for the corresponding drilling operation. Typical starting values for cutting speed are given in Table 8.16, while the recommended feeds are given in Table 8.17.

TABLE 8.16 Recommended cutting speeds for counter boring

Material	Cutting Speed, m/min
Mild steel	25 to 30
Alloy steel	10 to 25
Cast iron Soft	40 to 45
Brass	50 to 100

TABLE 8.17 Recommended feeds for counter boring

Counter Bore Size, mm	Feed, mm/rev
5 to 10	0.08 to 0.13
11 to 15	0.10 to 0.15
16 to 21	0.13 to 0.18
22 to 30	0.15 to 0.20
31 to 38	0.18 to 0.23
39 to 50	0.20 to 0.25

Spot Facing

Spot facing is similar to the counter boring, but removes only a very small portion of material around the existing hole to provide a flat surface square to the hole axis. This is normally done to provide a bearing surface for a *a*-washer or a nut or the head of a bolt. This has to be done only in cases where the existing surface is not smooth. The tool used can be same as that for counter boring or the tool as shown in Fig. 8.17.

Sometimes it becomes necessary to spot face a surface that is not accessible from the outside. For this purpose inverted spot facers are developed. These have the cutting portion that can be easily attached to the holder as shown in Fig. 8.18. First the holder is passed through the hole then the spot facer is attached at the end. After completing the operation, the spot facer is removed and the holder is taken out.

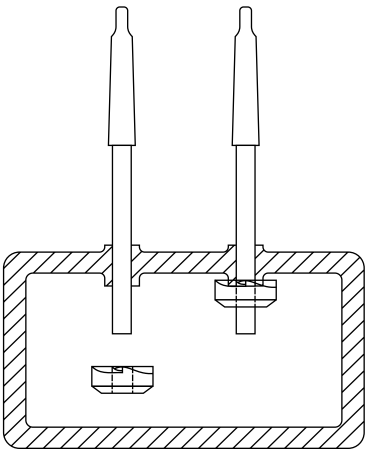


FIG. 8.25 *Inverse spot facing application*

Counter Sinking

Counter sinking is also similar to counter boring; except that the additional machining done on a hole is conical to accommodate the counter sunk machine screw head as shown in Fig. 8.25. Again the depth of counter sinking should be large enough to accommodate the screw head fully flush with the surface.

SUMMARY

There are a large number of hole making operations depending upon the geometry of the hole to be made.

- Drilling is the most common hole making operation.
- Twist drill geometry with two cutting lips arranged helically around the central web acts as the main cutting tool for drilling.
- A large variety of drill types are used such as step drills, spade drills, and shell core drills.
- Drill press, radial drilling machine and multiple spindle drilling machines are some of the varieties of drilling machines used.
- Drilling time can be estimated based on the hole length and the drilling process parameters.
- Drilling force and power can be estimated using the empirical relations.
- Deep hole drilling requires special precautions to take care of the removal of large volume of chips.
- Reaming is a finishing operation done to provide close-toleranced holes.
- Boring is an operation used to enlarge a hole using a single point cutting tool.
- Tapping is used to make inside threads where a finished hole is already present.
- Tapping is an operation using slow cutting speeds.
- Counter boring, spot facing and counter sinking are the other operations that are used to finish the holes for specific requirements.

Questions

8.1 Explain the different types of hole types and the processes used for manufacturing them.

8.2 Explain briefly the construction of a radial drilling machine with emphasis on how the requisite motions are obtained.

8.3 Explain different types of drilling machines that are used in machine shops and their features.

8.4 Show with sketches the principal features of any three hole making operations you are familiar with along with the tools used.

8.5 Show with neat sketches the constructional features of a twist drill and label the important features.

8.6 Explain briefly the construction of a drill press with emphasis on how the requisite motions are obtained.

8.7 What are the different types of drills used? Explain the function of each of the type of drill.

8.8 Define cutting speed, feed, and depth of cut as they are referred to drilling and explain how machining time in drilling is calculated.

8.9 Write a brief note on deep hole drilling operation.

8.10 Describe (a) Gang Drilling, and (b) Multiple spindle drilling operations.

8.11 Discuss about the problems faced in drilling operation with their causes and possible remedies.

8.12 Show with neat sketches the constructional features of a hand reamer and label the important features.

8.13 Explain the following terms related to reamers; a) left and right hand reamers, and (b) left hand and right hand helix in reamers. Explain their applications.

8.14 Is it possible to correct an out-of-round hole with a reamer? If not give reasons. If possible specify the type of reamer used for the purpose.

8.15 Explain the differences between drilling, reaming and tapping.

8.16 Write a brief note on trepanning.

8.17 Describe with the help of a neat diagram the construction and working of a precision horizontal boring machine? And also mention some salient design features?

8.18 Briefly discuss about the following types of boring machines:
(a) Floor type boring machine
(b) Planer type boring machine.
(c) Multiple head type boring machine.

8.19 Write short note on the following:
Lip, Helix and Rake angle in drilling

Problems

8.1 A series of 5 mm holes (total number 6) are to be drilled in a circle of 150 mm diameter on a 6 mm glass sheet. Describe the method of manufacture to be used with a neat sketch of the setup. What are the process variables to be controlled giving their effect on the final hole quality and the production rate? [Normal drilling cannot be done but use USM]

8.2 A hole of 25 mm diameter and 35 mm deep is to be drilled in mild steel component. The cutting speed can be taken as 35 m/min and the feed rate as 0.20 mm/rev. Calculate the machining time and the material removal rate. [0.511 minutes, 43 197 mm^3/min]

Calculate the drilling torque and thrust force acting in the above example.

[969.373 N mm, 686.35 N]

In C40 steel sheet of 25 mm thickness, 3 holes of 15 mm diameter are to be drilled. The cutting speed can be taken as 30 m/min and the feed rate as 0.15 mm/rev. Calculate the machining time and the material removal rate. [0.6984 minutes, 16 700 mm^3/min]

Multiple Choice Questions

8.1 Back taper is provided on a drill to
(a) Increase the strength of the drill
(b) Provide longitudinal clearance
(c) Decrease the cutting thrust
(d) Decrease the cost of the drill

8.2 Axial rake angle of a drill is
(a) The angle between the leading edge of the land and the axis of the drill
(b) The angle between the face and the line parallel to the drill axis
(c) The angle formed by the portion of the flank adjacent to the land and a plane at right angles to the drill axis
(d) None of the above

8.3 Helix angle of a drill is
(a) The angle between the leading edge of the land and the axis of the drill
(b) The angle between the face and the line parallel to the drill axis
(c) The angle formed by the portion of the flank adjacent to the land and a plane at right angles to the drill axis
(d) None of the above

8.4 The lip clearance angle of a drill is
(a) The angle between the leading edge of the land and the axis of the drill
(b) The angle between the face and the line parallel to the drill axis

(c) The angle formed by the portion of the flank adjacent to the land and a plane at right angles to the drill axis
(d) None of the above

8.5 The lip angle used in a drill for common drilling applications is
(a) 128º
(b) 136º
(c) 118º
(d) 125º

8.6 The following type of drill cannot be used for drilling into a solid material (can only enlarge an existing hole)
(a) Twist drill
(b) Core drill
(c) Spade drill
(d) Oil hole drill

8.7 A 15 mm through hole is to be drilled in a mild steel plate of 20 mm thickness. Take the over travel of the drill as 5 mm. What is the drilling time if the feed rate is 125 mm/min.
(a) 2 min
(b) 0.2 min
(c) 0.02 min
(d) 0.1 min

8.8 The cause for experiencing a lot of drill chatter (large vibrations)
(a) Loose moving parts of the machine
(b) Reduce overhang
(c) Fixture not properly clamped
(d) All of the above

8.9 The cause for very poor surface finish of the hole being drilled is
(a) Point improperly ground
(b) High feed rate
(c) Fixture not rigid
(d) All of the above

8.10 The machining operation used to enlarge an existing hole is termed as
(a) Drilling
(b) Boring
(c) Counter sinking
(d) Reaming

8.11 Boring operation is used for
(a) Drilling a hole
(b) Enlarging a hole
(c) Drilling a stepped hole in solid material
(d) None of the above

8.12 The operation to be used for obtaining smooth and close toleranced hole is
(a) Drilling
(b) Reaming
(c) Tapping
(d) Gun drilling

8.13 The type of reamer used for rough reaming operation is
(a) Left hand reamer with right hand helix
(b) Right hand reamer with left hand helix
(c) Right hand reamer with right hand helix
(d) None of the above

Answers to MCQs

8.1 (b)	8.2 (b)	8.3 (a)	8.4 (c)	8.5 (c)
8.6 (b)	8.7 (b)	8.8 (d)	8.9 (d)	8.10 (b)
8.11 (b)	8.12 (b)	8.13 (c)		

CHAPTER 9

Grinding and other Abrasive Processes

Objectives

Abrasive processes utilise very small abrasive grains to remove material in order to provide good finish on metallic parts. After completing this chapter, the reader will be able to

- Understand the basic principles of abrasive processes
- Understand the designation of the grinding wheel and the significance of the various elements of the codes
- Appreciate the different types of grinding machines and their applications
- Understand the grinding process and the variables that affect the operation
- Estimate the time and power required for the grinding operation
- Utilise various types of abrasive processes such as honing and lapping for final finishing operation

9.1 INTRODUCTION

Grinding is a process carried out with a grinding wheel made up of abrasive grains for removing very fine quantities of material from the work piece surface. The required size of abrasive grains are thoroughly mixed with the bonding material and then pressed into a disc shape of given diameter and thickness. This can be compared to a milling process with an infinite number of cutting edges.

Grinding is a process used for

- Machining materials which are too hard for other machining processes such as tool and die steels and hardened steel materials,
- Close dimensional accuracy of the order of 0.3 to 0.5 μm, and
- High degree of surface smoothness such as R_a = 0.15 to 1.25 μm.

This accounts for 25% of all the machining processes used for roughing and finishing processes.

The characteristics of some of the abrasive processes are given in Table 9.1.

The abrasive grains are basically spherical in shape with large sharp points, which act as cutting edges. All the grains are of random orientations and as such the rake angle presented to the work material can vary from positive to a large negative value. Many grit also slide rather than cut because of its orientation.

Table 9.1 Characteristics of various abrasive processes

Process	Particle Mounting	Features
Grinding	Bonded	Wheels, generally for finishing. Low material removal rate
Creep feed grinding	Bonded Open soft	Wheels, slow feed and large depth of cut
Snagging	Bonded, Belted	High material removal rate, roughing to clean and deburr castings and forgings
Honing	Bonded	Stones contain fine abrasives for hole finishing
Lapping	Free	For super finishing

The depth of cut taken by each of the grain is very small. However, a large number of grits act simultaneously, hence the material removed is large. Also cutting speeds employed are large. Chips produced as a result are very small and are red hot. Often they get welded easily to the abrasive grain or to the work piece. Thus the grinding process is a very inefficient one compared to the conventional metal cutting processes.

Specific energy of grinding ≈ 50 J/mm^3

Specific energy of other processes ≈ 2 to 5 J/mm^3

9.2 GRINDING WHEEL DESIGNATION AND SELECTION

The grinding wheels are produced by mixing the appropriate grain size of the abrasive with the required bond and pressed into shape. The characteristics of the grinding wheel depend upon a number of variables. They are described below:

9.2.1 Abrasive Types

These are the hard materials with adequate toughness so that they will be able to act as cutting edges for a sufficiently long time. They also have the ability to fracture into smaller pieces when the force increases, which is termed as friability. This property gives the abrasives the necessary self-sharpening capability. The abrasives that are generally used are:

- Aluminium oxide (Al_2O_3)
- Silicon Carbide (SiC)
- Cubic Boron Nitride (CBN)
- Diamond

Aluminium Oxide (Al_2O_3)

This is one of the natural abrasives found called corundum and emery. However the natural abrasives generally have impurities and as a result their performance is inconsistent. Hence the abrasive used in grinding wheels is generally manufactured from the aluminium ore, bauxite.

Silicon Carbide (SiC)

Silicon carbide is made from silica, sand, and coke with small amounts of common salt.

Cubic Boron Nitride (CBN)

Cubic Boron Nitride (CBN) next in hardness only to diamond (Knoop hardness ~ 4700 kg/mm^2). It is not a natural material but produced in the laboratory using a high temperature/ high pressure process similar to the making of artificial diamond. CBN is less reactive with materials like hardened steels, hard chill cast iron, and nickel base and cobalt based super alloys. CBN grains have 55 times higher thermal conductivity, four times higher the abrasive resistance and twice the hardness of the aluminum oxide abrasives. They can retain their strength above 10,000°C. CBN is very expensive, 10 to 20 times that of the conventional abrasive such as aluminium oxide.

Diamond

Diamond is the hardest known (Knoop hardness ~ 8000 kg/mm^2) material that can be used as a cutting tool material. It has very high chemical resistance along with low coefficient of thermal expansion. Also it is inert towards iron.

Table 9.2 Characteristics of abrasives used in grinding wheels

Abrasive	Vickers Hardness Number	Knoop Hardness	Thermal Conductivity, W/m K	Uses
Aluminium oxide	2300	2000 to 3000	6	Softer and tougher than SiC used for steels and high strength materials
Silicon carbide	2800	2100 to 3000	85	Nonferrous, non-metallic materials, Hard and dense metals and good finish
Cubic Boron Nitride	5000	4000 to 5000	200	Hard and tough tool steels, stainless steel, aerospace alloys, hard coatings
Diamond (synthetic)	8600	7000 to 8000	1000 to 2000	Some die steels and tungsten carbide
Hardened steel		~700		

9.2.2 Grain Size

Compared to a normal cutting tool, the abrasives used in a grinding wheel are relatively small. The size of an abrasive grain, generally called grit, is identified by a number which is based on the sieve size used. These would vary from a very coarse size of 6 or 8 to a super fine size of 500 or 600. Sieve number is specified in terms of the number of openings per square inch. Thus larger the grain number finer is the grain size.

The surface finish generated would depend upon the grain size used as shown in Table 9.3. The fine grains would take a very small depth of cut and hence a better surface finish is produced. Also fine grains generate less heat and are good for faster material removal. Though each grain cuts less, there are more grains per unit surface area of the wheel in case of fine grain size. Fine grains are also used for making the form grinding wheels.

Table 9.3 Surface finish obtained with grain size

Grain Size	Surface Finish, μm
46	0.8
54	0.6 to 0.8
60	0.4 to 0.6
80	0.2 to 0.4

Coarse grains are good for higher material removal rates. These have better friability and as a result are not good for intermittent grinding where they are likely to chip easily.

9.2.3 Bond

The function of the bond is to keep the abrasive grains together under the action of the grinding forces. The commonly used bond materials are:

- Vitrified
- Silicate
- Synthetic resin
- Rubber
- Shellac
- Metal

Vitrified

This is the most commonly used bond. The bond is actually clay mixed with fluxes such as feldspar, which hardens to a glass like substance on firing to a temperature of about 1250°C and develops the strength. This bond is strong, rigid and porous, and not affected by fluids. However, this bond is brittle and hence sensitive to impacts. This bond is also called ceramic bond.

Silicate

This is sodium silicate ($NaSiO_3$) or water glass and hardens when heated. Not as strong as vitrified. This can be used in operations that generate less heat. It is affected by dampness but less sensitive to shocks. Relatively less used.

Synthetic Resin or Resinoid

These bonding materials are thermosetting resin such as phenol formaldehyde. This bond has good strength and is more elastic than the vitrified bond. However, this is not heat and chemical resistant. Generally used for rough grinding, parting off and high speed grinding (50 to 65 m/s). It can also be used for fine finishing of roll grinding.

Rubber

Of all the bonds used, this is the most flexible. The bond is made up of natural or synthetic rubber. The strength is developed with vulcanisation. This has high strength and is less porous. This bond is affected by dampness and alkaline solutions. Generally used for cutting off wheels, regulating wheels in centre less grinding and for polishing wheels.

Shellac

This is relatively less used bond. Used generally for getting very high finish. Typical applications are rolls, cutlery, and cam shaft finishing.

Metal

This is used in the manufacture of diamond and CBN wheels. The wheel can be made of any high thermal conductive metal such as copper alloys or aluminium alloys. The periphery of the wheel up to a small depth of the order of 5 mm or less contains the abrasive grit. The choice of the metal depends on the required strength, rigidity and dimensional stability. In view of the strong bond, the grit will not be knocked out till it is fully utilised. Powder metallurgy techniques are used to make the abrasive periphery.

9.2.4 Grade

It is also called the hardness of the wheel. This designates the force holding the grains. The grade of a wheel depends on the kind of bond, structure of wheel and amount of abrasive grains. Greater bond content and strong bond results in harder grinding wheel. Harder wheels hold the abrasive grains till the grinding force increases to a great extent. The grade is denoted by letter grades as indicated in Fig. 9.1.

Soft wheels are generally used for hard materials and hard wheels are used for soft materials. While grinding hard materials the grit is likely to become dull quickly thereby increasing the grinding force, which tend to knock of the dull abrasive grains. This keeps the grinding wheel in sharp condition. In contrast the hard grinding wheel while grinding soft materials will be able to retain the grit for longer periods thus improving the material removal. Typical suggested wheel hardnesses for various materials are shown in Table 9.4.

Very soft Medium Very hard
ABCDEFGHIJKLMNOPQRSTUVWXYZ

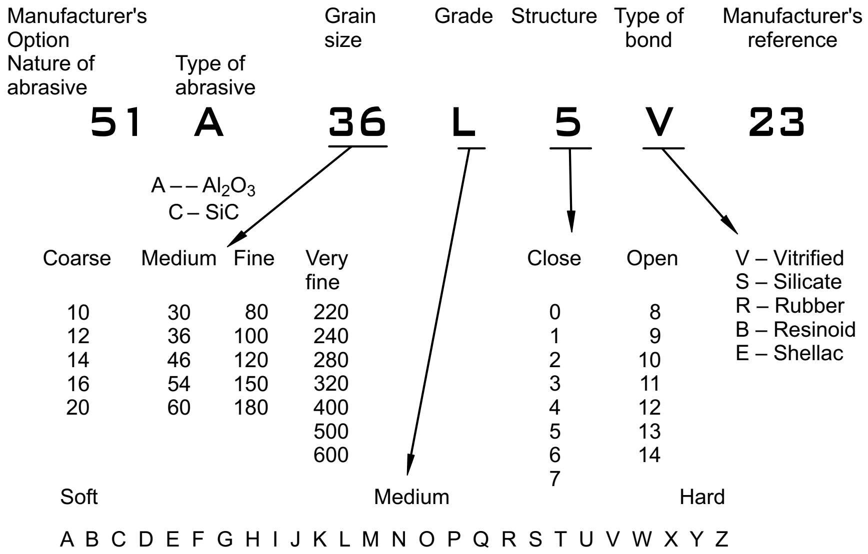


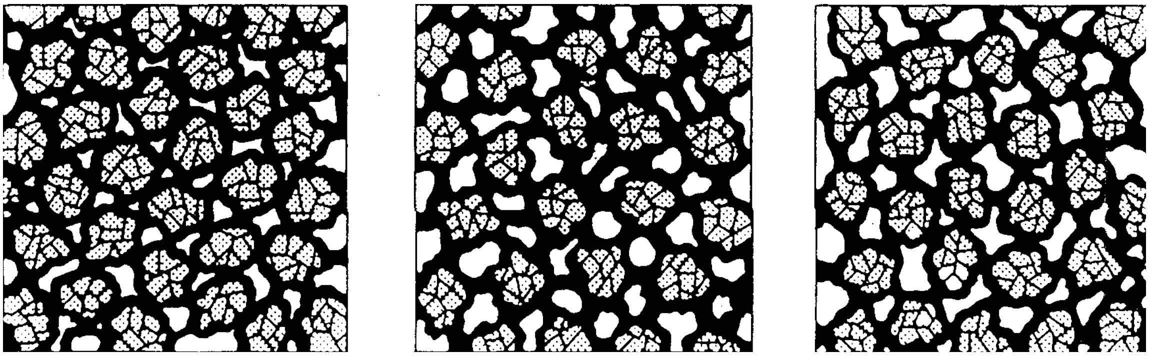
Fig. 9.1 *Grinding wheel standard marking system*

9.2.5 Structure

The structure of a grinding wheel represents the grain spacing. It can be open or dense and is shown in Fig. 9.2 conceptually. It is generally denoted by numbers as shown in Fig. 9.1. The spacing between the grains allows for chips to collect as shown schematically in Fig. 9.3. This helps avoiding the loading of the grinding wheel. Open structures are used for high stock removal and consequently produce rough finish. Dense structures are used for precision forms and profile grinding.

TABLE 9.4 Grinding wheel hardness for different work materials

Work Piece Material	Wheel Hardness			
	Cylindrical Grinding	Surface Grinding	Internal Grinding	Deburring
Steel up to 80 kg/mm^2	L, M, N	K, L	K, L	
Steel up to 140 kg/mm^2	K	K, J	J	O, P, Q, R
Steel more than 140 kg/mm^2	J	I, J	I	
Light alloys	J	I, K	I	
Cast iron	K	J	J	
Bronze, brass and copper	L, M	J, K	J	



(a) Dense (b) Medium (c) Open

FIG. 9.2 *Grinding wheel structure*

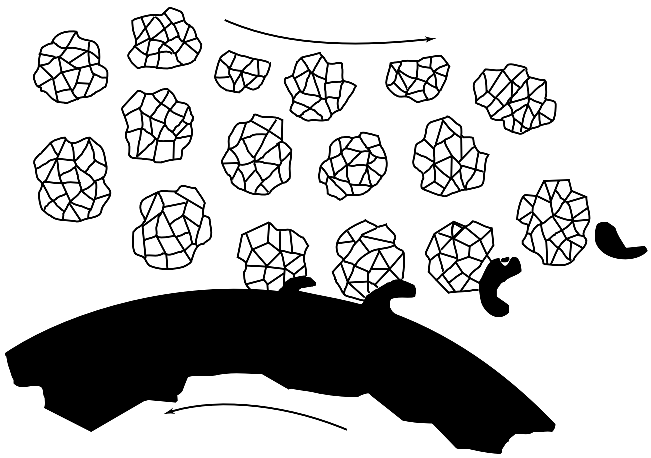


FIG. 9.3 *Illustration showing how the spaces between grit help in clearing the grinding chips*

The grinding wheel marking system should be able to specify the abrasive used, grain size, grade, structure and bond used in the sequence as shown in Fig. 9.1.

Typical grinding wheel selections are given in Table 9.5.

Table 9.5 Grinding wheel specifications for different grinding operations

Operation	Grinding Wheel Designation
Cylindrical grinding of hardened steel	A60L5V
Cylindrical grinding of soft steel	A54M5V
Cylindrical grinding of aluminium	C36K5V
Surface grinding of hardened steel	A60F12V
Surface grinding of soft steel	A46J5V
Surface grinding of grey cast iron	C36J8V
Tool grinding of high speed steel	A46K8V

9.2.6 Grinding Wheel Types

Grinding wheels come in a variety of shapes and standardised sizes as shown in Fig. 9.4. These suit various work piece shapes and sizes, and are also used in different types of grinding machines. The most common is

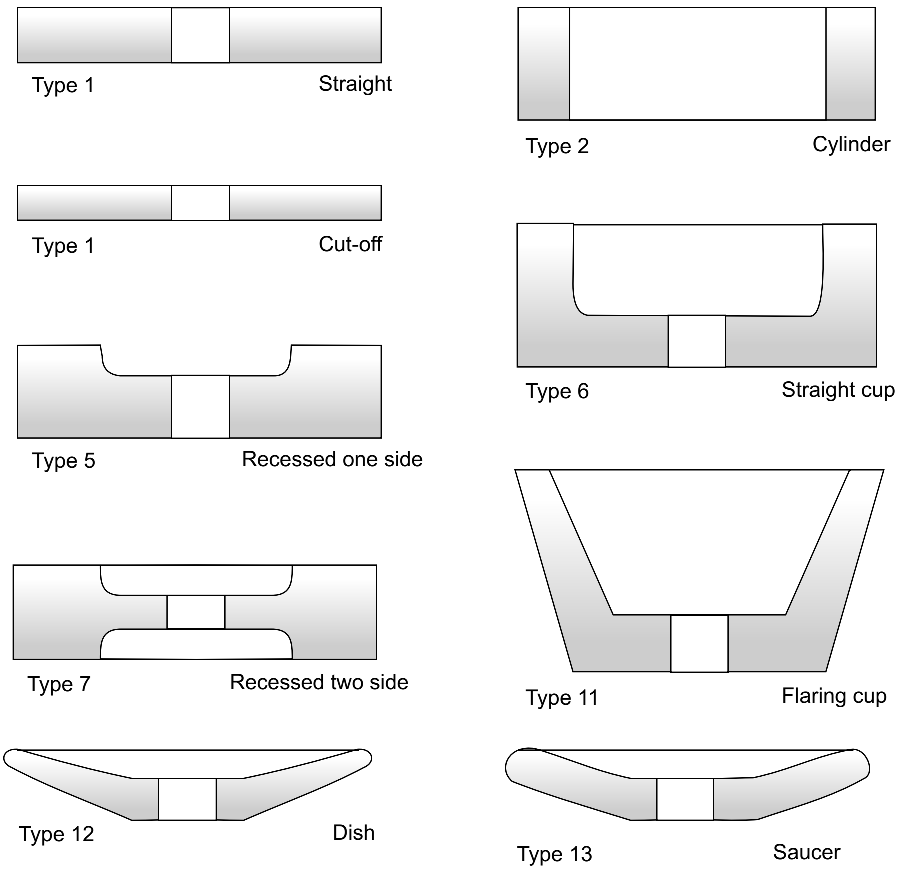


Fig. 9.4 *Grinding wheel shapes*

the straight shape, shown as type 1 in Fig. 9.4 which is used for a variety of cylindrical grinding applications. The type 1 wheel will have further modification of the end shape as shown in Fig. 9.5 to suit specific applications. The cylinder shown as type 2 is used for grinding flat surfaces. Similarly the flaring cup is used for grinding the cutting tools.

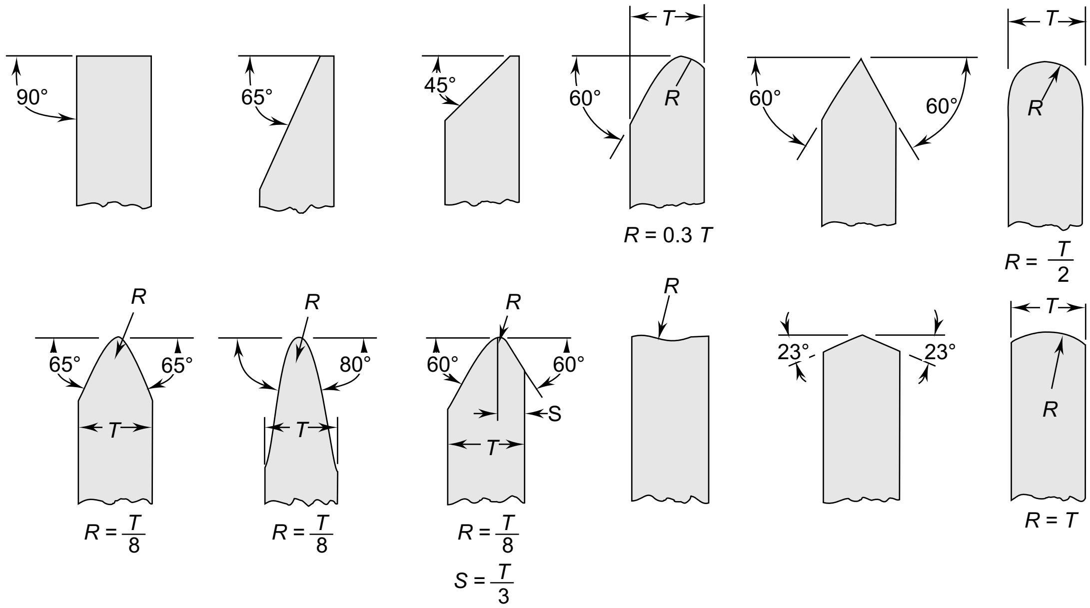


Fig. 9.5 *Various faces of grinding wheel form for the straight (Type 1) wheel shown in Fig. 9.4*

The size of the grinding wheel is normally specified by the

- Diameter of the wheel
- Diameter of the spindle hole
- Face width of the wheel

9.2.7 Wheel Balancing

New grinding wheels when used should be properly balanced. Balance of a grinding wheel also depends upon the machine spindle as well as the condition of tightening which will have to be properly taken care of. In view of the high rotational speeds used, any residual unbalance left would be harmful for the machine part and also produce poor surface finish. Such wheels are provided with movable balance weights for adjusting the balance mass location. The balancing operation can be carried in two ways:

- Static balancing
- Dynamic balancing

In static balancing the grinding wheel is rotated on an arbour and the balance weights adjusted until the wheel no longer stops its rotation in any one specific position. To do this the balance weights are removed and the wheel is kept on the balancing ways. The wheel is allowed to rotate such that the heavier portion of the wheel settles at rest. Place a chalk mark at the heavier portion (bottom most point). Try to rotate the wheel slightly and see where the wheel is resting. The chalk mark should always point to the bottom, which confirms that the heaviest portion is identified. Two weights are now inserted such that they are equidistant from the heavy mark and slightly above the horizontal mark in that position as shown in Fig. 9.6(a). If the wheel stops again at the same point, then move the weights closer. If the wheel stops in the opposite direction, move the weights further apart. It should be possible to find a point of proper balance by repeating this process. If it is not possible by any combination to find a balance, then add more balance weights as shown in Fig. 9.6(b).

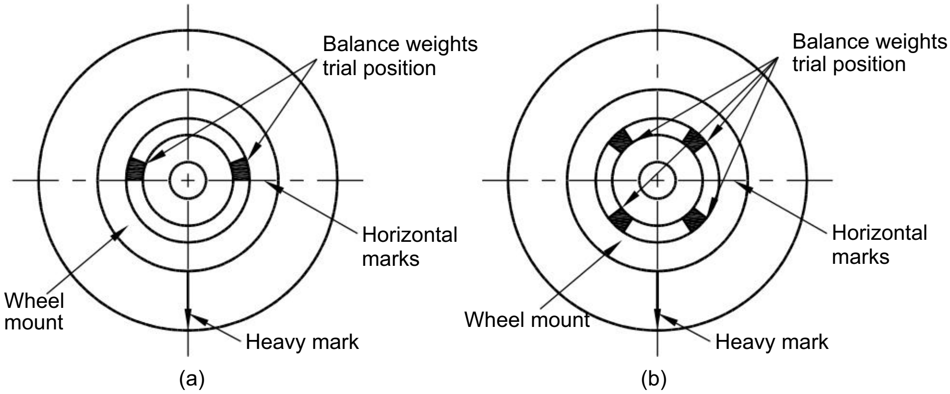


Fig. 9.6 *Static balancing of the Grinding wheel*

9.2.8 Dressing and Truing

With continuous use a grinding wheel becomes dull with the sharp abrasive grains becoming rounded. This condition of dull grinding wheel with worn out grains is termed as glazing. Further, some grinding chips get lodged into the spaces between the grit with the resulting condition known as loaded wheel. Loading is generally caused during the grinding of soft and ductile materials. A loaded grinding wheel cannot cut properly. Such a grinding wheel can be cleaned and sharpened by means of a process called dressing. A simple dressing is done by means of small steel disks, which are free to rotate at the end of a stick. When these disks contact the grinding wheel face they sharpen the wheel by removing a small portion of the face of wheel. Though the dressing is simple, it will not produce a true concentric surface because it is done manually.

Dressing can also be done by abrasive disks made of silicon carbide (less frequently boron carbide) for smaller-size wheels. The stick is applied directly to the wheel surface. A free rotating dressing wheel mounted on the table firmly with silicon carbide grains in hard vitrified bond wheel fixed on a ball bearing spindle can also be used for dressing. The wheel will crush the grinding wheel surface thus providing an improved control of the dressed surface characteristics.

A true surface of the grind wheel in terms of either the form or concentricity can be achieved with the help of a diamond dressing tool. A diamond used for truing is set in a closely fitting hole at the end of a short steel bar and is brazed as shown in Fig. 9.7. To do the truing operation, the grinding wheel is rotated at its normal speed and a small depth of 0.025 mm is given while moving the dressing tool across the face of the grinding wheel in an automatic feed. The cross feed rates are controlled depending upon the required surface. Slow feed rates are used for generating fine finishes while faster feeds are used for free cutting. The feed rate also depends upon the grain size of the wheel. The cross feed rates to be used are given in Table 9.6. Diamond dressing is also used to generate the necessary form, other than the straight form. Diamond dressing on CNC grinding machines can generate any form of wheel.

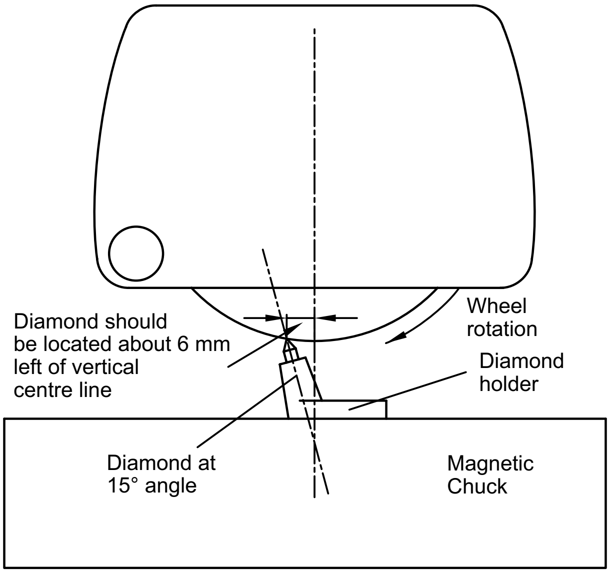


Fig. 9.7 *Truing of a Grinding wheel using a diamond dresser on a surface grinder*

Table 9.6 **Cross feed for Diamond truing**

Grain Size	Cross Feed per Wheel Revolution, mm
30	0.350 to 0.600
36	0.300 to 0.475
46	0.200 to 0.350
50	0.175 to 0.300
60	0.150 to 0.250
80	0.100 to 0.175

9.3 TYPES OF GRINDING MACHINES

Grinding operations are generally classified based on the type of surface produced. The grinding operations possible can be classified into

- Cylindrical grinding for generating cylindrical surfaces
- Surface grinding for generating flat surfaces, and
- Centre less grinding for generating axi-symmetric shapes.

Typical shape of these processes is shown in Fig. 9.8.

Grinding machines since used for precision work are generally produced with rigid frames, accurate spindles and heavy power for producing parts with close dimensional tolerances.

9.3.1 Cylindrical Grinding

The cylindrical grinding machine is used generally for producing external cylindrical surfaces. The machine is very similar to a centre lathe. Typical movements in a cylindrical grinding machine are shown in Fig. 9.9.

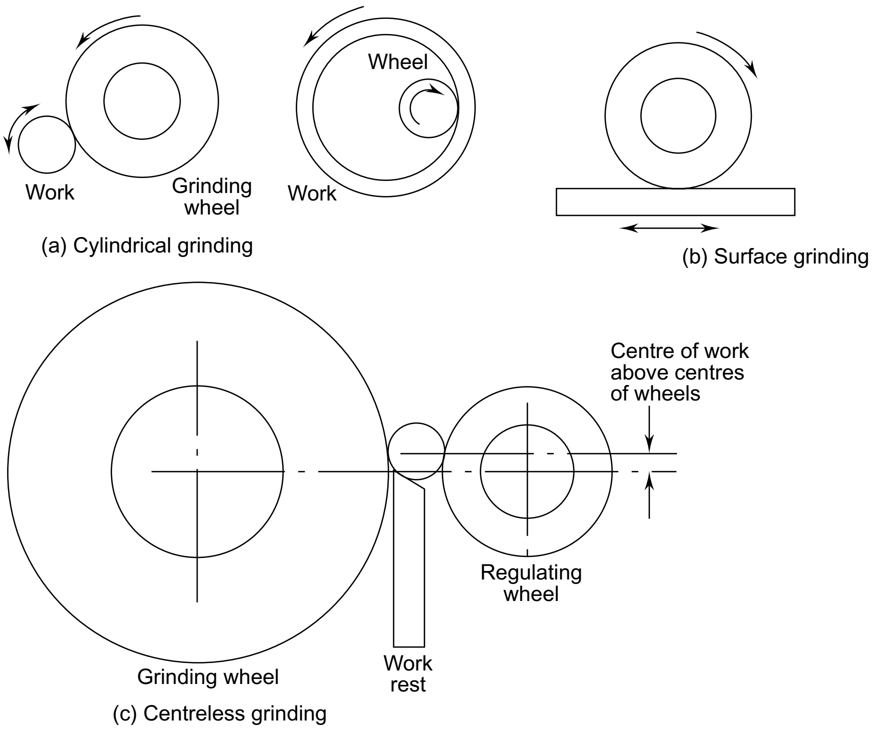


Fig. 9.8 *Typical grinding operations*

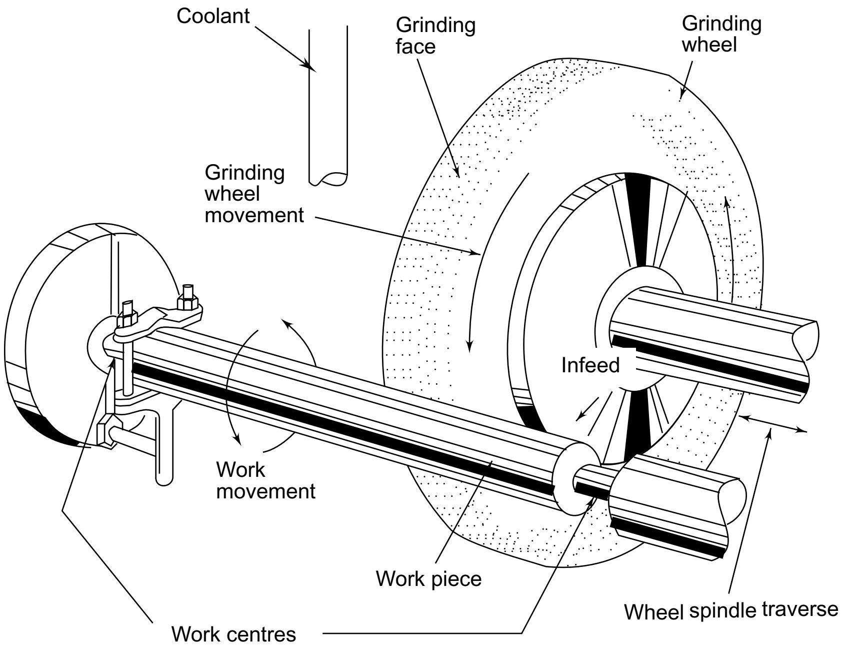


Fig. 9.9 *Cylindrical Grinding operation*

The grinding wheel is located similar to the tool post, with an independent power driven at high speed suitable for grinding operation. Both the work and the grinding wheel rotate counter clockwise. The work that is normally held between centres is rotated at much lower speed compared to that of the grinding wheel as shown in Fig. 9.10.

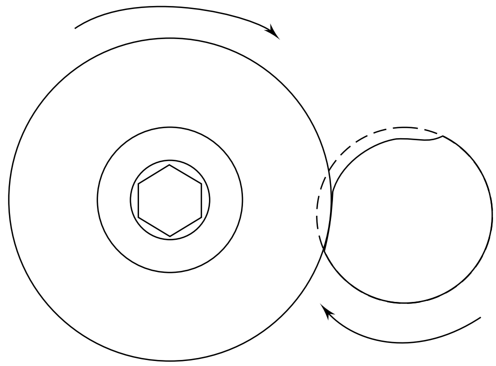


Fig. 9.10 *Relative motions of grinding wheel and the work in the cylindrical grinding operation*

If the finished section to be ground is wider than the wheel, the wheel is fed in the transverse direction. Plunge grinding is done if the part is the same size as or less than the width of the wheel. Very fine finishes are obtained with cylindrical grinding. It is possible to get accuracies within 0.25 μm with extreme care. Work pieces are normally mounted between centres and are driven by a dog. If necessary, the work should be supported by work rests, placed opposite the wheel to prevent deflection.

The traverse feed of the work piece past the grinding wheel is provided by using a hydraulic arrangement. In feed is provided by the movement of the grinding wheel head into the work piece. Economical grinding allowances that can be left are about 0.1 to 0.3 mm.

9.3.2 Surface Grinding

Surface grinding machines are generally used for generating flat surfaces. By far these are used for the largest amount of grinding work done in most of the machine shops. These machines are similar to milling machines in construction as well as motion. There are basically four types of machines depending upon the spindle direction and the table motion as shown in Fig. 9.11. They are:

- Horizontal spindle and rotating table

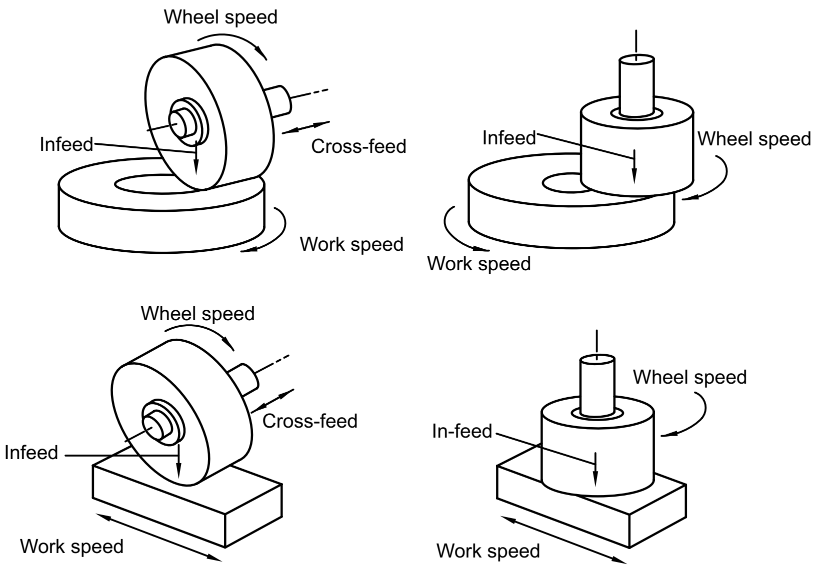


Fig. 9.11 *Different surface Grinding operations*

- Vertical spindle and rotating table
- Horizontal spindle and reciprocating table
- Vertical spindle and reciprocating table

Horizontal Spindle and Rotating Table

In this machine the grinding wheel cuts on its periphery, while the spindle traverses horizontally from the edge to the centre of the table. Feed is accomplished by moving the work mounted on the table up into the wheel with the table moving in a rotary fashion. Since the table and work rotate in a circle beneath the grinding wheel, the surface pattern is a series of intersecting arcs. This machine is used for round, flat parts because the wheel is in contact with the work at all times.

Vertical Spindle and Rotating Table

Vertical spindle machines are generally of bigger capacity. Complete machining surface is covered by the grinding wheel face. They are suitable for production grinding of large flat surfaces. In this machine both the work and the wheel rotate and feed into each other. By taking deep cuts this machine removes large amounts of material in a single pass. The side or the face of the wheel does the grinding. The wheel can be either complete solid or split into segments to save wheel material and in the process also provide cooler grinding action. In the case of small parts, the surface patterns created are a series of intersecting arcs if they are off centre around the table. It is a versatile machine and can be used to grind production parts and very large parts, as well as for grinding large batches of small parts as shown in Fig. 9.11.

Horizontal Spindle and Reciprocating Table

The table in the case of reciprocating machines is generally moved by hydraulic power. The wheel head is given a cross feed motion at the end of each table motion. In this machine, the wheel should travel over the work piece at both the ends to prevent the grinding wheel removing the metal at the same work spot during the table reversal. This is the most common grinding machine found in the tool rooms. The tables for this type of machines are rectangular and usually 150 mm wide by either 300 mm or 450 mm long. The high-production types have tables as big as 2 m by 5 m. The grinding wheels cut on their peripheries and vary in sizes from 175 mm in diameter and 12.5 mm in width to 500 mm in diameter and 200 mm in width. This type of surface grinder is the most commonly used because of its high accuracy and the fine surface finishes that it imparts. The grinding wheel traverses in a straight pattern that results in superior finish and high precision.

Vertical Spindle and Reciprocating Table

The grinding wheel in this machine is cylindrical and cuts on its side rather than on its periphery. The work is fed by the reciprocating motion of the table. Generally, the diameter of the wheel is wider than the work piece and as a result no traverse feed is required. These are generally high-production machine tools removing large amounts (as much as 10 mm) in a single pass.

9.3.3 Centre Less Grinding

Centre less grinding makes it possible to grind cylindrical work pieces without actually fixing the work piece using centres or a chuck. As a result no work rotation is separately provided. Principle of operation is shown in Fig. 9.12(a) for external cylindrical surfaces.

The process consists of two wheels, one large grinding wheel and another smaller regulating wheel. The work is held on a work rest blade. The regulating wheel is mounted at an angle to the plane of the grinding wheel. The centre of the work piece is slightly above the centre of the grinding wheel. The work piece is

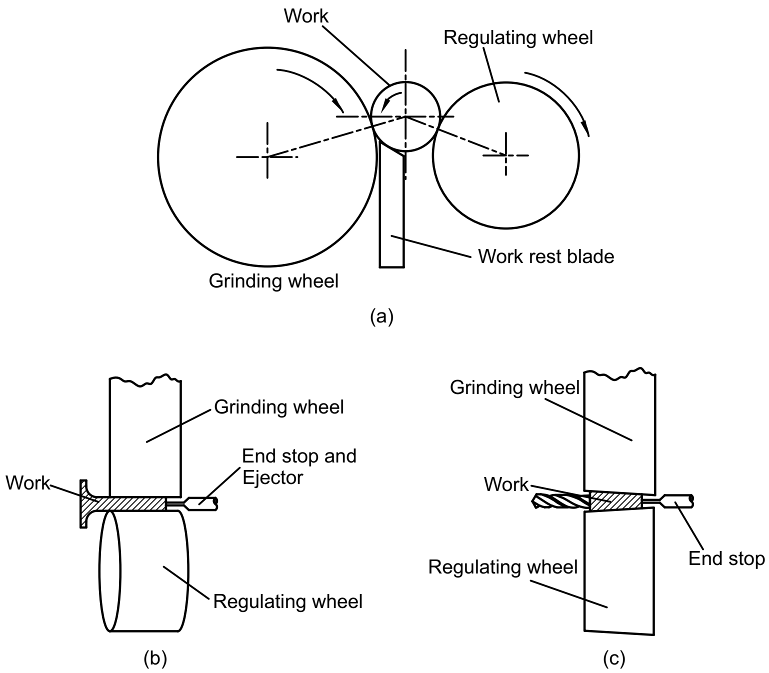


Fig. 9.12 *Centre less Grinding operations; (a) Through feed, (b) in feed and (c) end feed*

supported by the rest blade and held against the regulating wheel by the grinding force. As a result the work rotates at the same surface speed as that of the regulating wheel. The regulating wheel is generally a rubber or resinoid bonded wheel with wide face. The axial feed of the work piece is controlled by the angle of tilt of the regulating wheel. Typical work speeds are about 10 to 50 m/min.

There are three types of centre less grinding operations possible as shown in Fig. 9.12. They are

(a) Through feed centre less grinding Fig. 9.12(a) as described above.
(b) In feed centre less grinding shown in Fig. 9.12(b), where the grinding is done only by plunge feeding so that any form surface could be produced. This is useful if the work piece has an obstruction which will not allow it to traverse past the grinding wheel. The obstruction could be a shoulder, head, round form, etc. The work piece will be loaded into the machine while the work rest blade and the regulating wheel are withdrawn.
(c) End feed centre less grinding shown in Fig. 9.12(c), where tapered work pieces can be machined.

Advantages of centre less grinding

1. There is no need for having and maintaining centres and centre holes.
2. Work pieces can be loaded and unloaded from the machine rapidly. Grinding is almost continuous for through feed grinding.
3. Backing up the work piece by the regulating wheel and work rest blade practically eliminates any deflection of the work piece. This permits maximum material removal rates.
4. Minimum wear is observed in view of the large grinding wheels used. This minimises the adjustments needed for staying within dimensional tolerances and maximises the periods of time between wheel dressings.
5. Work pieces may often be loaded into the machine by the automatic feeding devices.

6. Less grinding allowance may be required, because the out-of-roundness is corrected across the diameter rather than the radius.

Limitations of centre less grinding

1. Setup time for a centre less grinding operation is usually large.
2. This process is useful only for large volume production. It may be necessary to have special equipment and additional setup time for special profiles.
3. This process is not suitable for large work piece sizes.

Internal Centre-less Grinding

It is also possible to apply centre-less grinding for internal surfaces as well. However, in this case, the work piece needs to be supported by two support rolls as shown in Fig. 9.13. The main advantage of the process is that the ground hole will be concentric with the outside diameter of the work. The process is capable of grinding straight cylindrical or tapered holes. These holes can be blind, through, interrupted, or even with a shoulder.

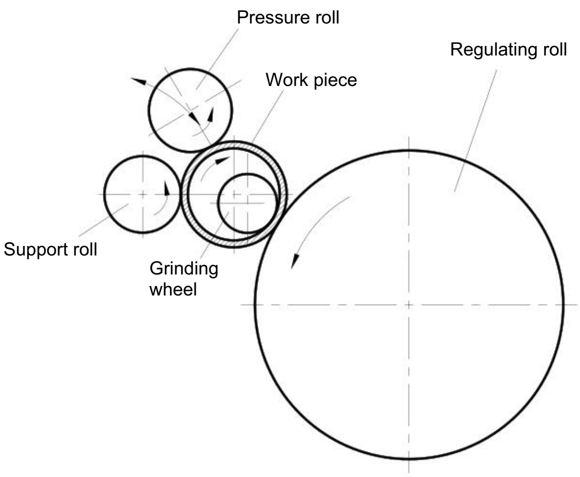


Fig. 9.13 *Internal Centre-less Grinding operation*

First support roll is mounted below the work to support it and control the distance from the work centre. The second support roll is a pressure roll and holds the work in contact with the other two. This roll moves in and out to allow for loading and unloading the machine. The third roller as shown is the regulating roll that drives the work and controls its speed and direction. The grinding wheel remains in a fixed position, and the work traverses past the grinding wheel. This is almost similar to through feed center-less grinding with simplified setup and automatic loading of work pieces.

9.4 GRINDING PROCESS

To understand the grinding process, it is convenient to study the metal removal process by the abrasive grain. For the sake of simplicity the surface grinding process is considered in the following segment.

9.4.1 Surface Grinding

The removal of metal by a single abrasive grit in surface grinding operation is shown in Fig. 9.14. The action is very similar to a milling machine. From the figure,

$$\cos\theta = \frac{OF}{D/2} = 1 - \frac{d}{D/2} \qquad (1)$$

$$d = \frac{D}{2}(1 - \cos\theta) \qquad (2)$$

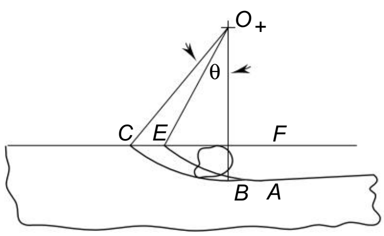


Fig. 9.14 *Geometry of single abrasive grain removing material*

Also from the figure

$$CF = \frac{D}{2}\sin\theta \tag{3}$$

Using Eq. 2, Eq. 3 may be written as

$$CF^2 = \left(\frac{D}{2}\right)^2\left[1 - \left\{1 - \frac{d}{D/2}\right\}^2\right] \tag{4}$$

Or

$$CF = \sqrt{Dd - d^2} \tag{5}$$

$$l = \text{Arc length of } BC \approx \text{Chord length } BC$$

$$l = BC = \sqrt{CF^2 + d^2} = \sqrt{Dd} \tag{6}$$

The shape of the chip can be approximated as shown in Fig. 9.15. In that case

$$t = CE\sin\theta = 2\,CE\sqrt{\frac{d}{D} - \left(\frac{d}{D}\right)^2} \tag{7}$$

From this, $\left(\frac{d}{D}\right)^2$ can be neglected, as d/D is very small.

CE is the distance moved by the table during the time the grinding wheel makes $1/K$ revolutions.

K is the number of abrasive grains on the surface.

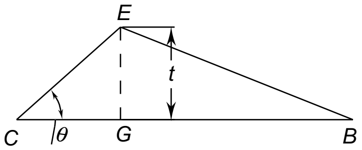


Fig. 9.15

$$CE = \frac{v_t}{KN} \tag{8}$$

where, v_t = Table speed, and

N = Grinding wheel rpm.

Hence,

$$t = \frac{2v_t}{KN}\sqrt{\frac{d}{D}} \tag{9}$$

If there are C grains per cutting unit square of the surface of the wheel, and if the average width of each cut is b, then

$$K = \pi DbC \tag{10}$$

$$t = \frac{2}{bC}\frac{v_t}{V_w}\sqrt{\frac{d}{D}} \tag{11}$$

It is possible to show for cylindrical grinding case, the expression would be as follows:

$$t = \frac{4}{bc}\frac{v_t}{V_w + v_t}\sqrt{\frac{d}{D} \pm \frac{d}{D_w}} \tag{12}$$

where

+ for external grinding
– for internal grinding
D_w = Work diameter

As t increases the grinding force increases which increase the wheel wear. This is equivalent to SOFT grade of the grinding wheel.

Let us now consider an example to get an idea of the quantities involved.

v_t = 30 m/min
V_w = 1800 m/min = 30 m/s
b = 0.1 mm
C = 2 (generally from 0.1 to 10 per mm^2)
d = 0.05 mm
D = 200 mm

$$1 = \sqrt{dD} = \sqrt{200 \times 0.05} = 3.2 \text{ mm} \tag{13}$$

$$t = \frac{2}{0.1 \times 2}\frac{30}{1800}\sqrt{\frac{0.05}{200}} = 0.00263 \text{ mm} \tag{14}$$

The basic operating conditions to be considered for the grinding process are:

- Wheel speed
- Work speed (in case of cylindrical grinding)
- Traverse feed of the wheel
- In feed
- area of contact

As can be seen from the above equations, as the wheel speed is increased, wheel wear as well as the size of the chips produced is reduced. The general range of values is from 20 to 40 m/s.

As the work speed is increased, wheel wear is increased while the heat produced is reduced. The general range of values is from 10 to 20 m/min.

As the in feed is increased, wheel wear is increased while the surface finish deteriorates.

The traverse feed is specified as a fraction of the width of the wheel. As the work speed is increased, wheel wear is increased while the surface finish deteriorates. The general range of values is from 1/2 to 3/4 of wheel width for steels while for cast irons from 3/4 to 5/6 of wheel width.

In surface grinding the stock removal rate, Q is given by

$$Q = bdv$$

In the case of cylindrical grinding it is

$$Q = 2\pi R_w df$$

where d = depth of cut
v = work velocity for surface grinding
b = width of cut for surface grinding
f = wheel traverse velocity (feed rate)
R_w = Work radius

The above equations apply provided the wheel does not wear too rapidly, so that d remains constant.

The specific cutting energy, U_s is given by

$$U_s = \frac{F_h V}{v b d}$$

where F_h = the average horizontal force (tangential to the grinding wheel for the small cuts taken during grinding) and V is the grinding wheel peripheral velocity.

The specific energy in grinding has been found to be higher than that for single point tool processes by a factor of about 10. This may be explained from the concept of 'size effect'. Due to the very small undeformed chip thickness in the case of grinding, there are few or no dislocations in the chips. Thus the deformation has to be carried on a perfect metal by breaking the metallic bond and this requires more grinding forces.

Grinding Wheel Performance

Grindability is a term used to describe the relative ease of grinding and is comparable to machinability for single point cutting tools. It is concerned with the forces and power required in grinding, wheel wear, stock removal rate and the surface finish produced. A material is considered to have good grindability if the forces, power and wheel wear are low, while stock removal is high and surface finish produced is good.

When the wheel glazes the work, considerable rubbing and little cutting occurs, causing low stock removal rates and excessive grinding temperatures. This results in thermal cracks and metallurgical changes at the work surface. Rapid wheel wear leads to a deterioration of dimensional accuracy and surface finish and increases the grinding and production costs.

As was seen above the shape of the grinding chip is triangular, and it can be assumed that the grinding force, F_m is proportional to the area of cut which in turn is proportional to the square of the undeformed chip thickness.

$$F_m \propto A \propto t_m^2$$

Glazing occurs when the worn grits are not fractured or dislodged from the grinding wheel. By increasing the forces acting on each grit, the worn grits can be fractured or pulled out of the bond, thus exposing the new and sharp cutting points. Similarly rapid wheel wear was thought to occur when high forces act on the grits.

It is interesting to note that as the wheel diameter is reduced due to the wheel wear, dressing and truing, the force on the grit will vary. Thus, as the R_w radius of the work decreases force increases for external cylindrical grinding and the wheel wear will increase. Therefore the wheel appears to be softer. The reverse occurs for internal grinding. When the work diameter (or the hole size) is decreased the wheel will appear to be softer for external or internal grinding.

The wheel wear is generally measured with a parameter called 'Grinding Ratio' which is defined as the ratio of volume of metal removed to the volume of metal worn from the grinding wheel. The Table 9.7 shows some of the influences exhibited by the operating conditions on the grinding performance.

Table 9.7 Grinding process parameters on performance

Variable	Grinding Ratio	Net Power	Surface Finish
Increase in wheel speed	Increased	Slightly increased	Improved
Increased depth of cut	Decreased	Increased	Deteriorates
Increase in work speed	Decreased	Increased	Deteriorates
Increase in work diameter	Increased	Slightly Increased	No significant change
Increase in metal removal rate	Decreased	Increased	Deteriorates
Increase in work material hardness	Optimum exhibited	Slightly Increased	Improves

9.5 GRINDING PROCESS PARAMETERS

The operating parameters of a grinding process are:

- Wheel speed
- Work speed
- Traverse speed
- In feed
- Area of contact

The recommended grinding wheel speeds based on the type of operation are given in Table 9.8. Similarly the suggested work speeds for various work materials are given in Table 9.9.

TABLE 9.8 Recommended grinding wheel speeds

Type of Grinding	Bond	Wheel Speed, m/s
Rough grinding	Vitrified	25
Rough grinding	Resinoid	45
Surface grinding	Vitrified	20–25
Internal grinding	Vitrified	20–35
Centre less grinding	Vitrified	30–80
Cylindrical grinding	Vitrified	30–35
Cutting off	Resinoid	45–80
Hand grinding of tools		20–25
Automatic grinding of tools		25–35
Hand grinding of carbide tools		18–25
Automatic grinding of carbide tools		4–20

TABLE 9.9 Recommended work speeds

Work Material	Cylindrical Grinding		Internal	Surface
	Roughing m/min	Finishing m/min	Grinding m/min	Grinding m/min
Soft steels	–15	6–8	15–20	
Hard steels	14–16	6–10	18–22	
Cast iron	12–15	6–10	18–22	8–15
Brass	18–20	14–16	28–32	
Aluminium	50–70	30–40	32–35	

9.5.1 Grinding Time Estimation

Schematically the surface grinding operation with a horizontal axis grinding machine is shown in Fig. 9.16. The grinding wheel will have to traverse beyond the actual work piece by a distance termed as the approach allowance, A which is given by

$$A = \sqrt{\left(\frac{D}{2}\right)^2 - \left(\frac{D}{2} - d\right)^2} = \sqrt{d\,(D - d)}$$

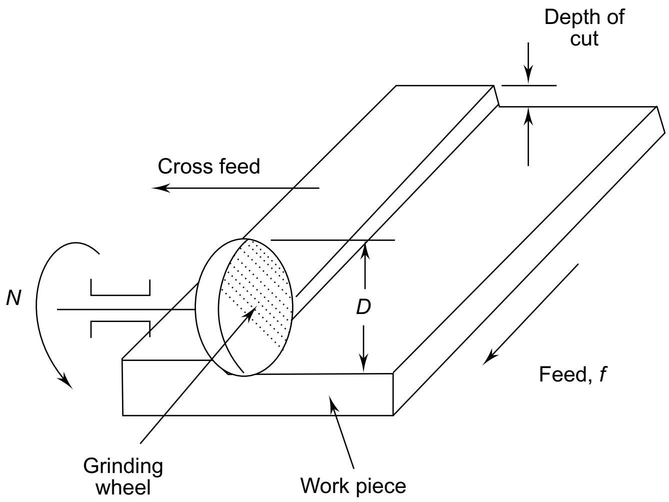


Fig. 9.16 *Grinding operation*

This value is very small since the depth of cut, d is very small in grinding. However, to allow for the table reversal at each end of the table stroke, the radius of the grinding wheel is assumed as the approach allowance. Thus

$$\text{Time for one pass} = \frac{\text{Length + Diameter}}{\text{Table feed rate}}$$

$$\text{Number of passes required} = \frac{\text{Width}}{\text{Infeed rate}}$$

Example 9.1

Using a horizontal axis surface grinder a flat surface of C65 steel of size 100 X 250 mm is to be ground. The grinding wheel used is 250 mm in diameter with a thickness of 20 mm. Calculate the grinding time required. Assume a table speed of 10 m/min and wheel speed of 20 m/s.

Solution The rpm of the grinding wheel $= \dfrac{1000 \times 20 \times 60}{\pi \times 250} = 1528$ rpm

Let approach distance = 125 mm

Time for one pass $= \dfrac{250 + 250}{10 \times 1000} = 0.05$ minutes

Assuming an in feed rate of 5 mm/pass

Number of passes required = 100/5 = 20

Total grinding time = 20×0.05 = 1 minute.

Fig. 9.17 shows the situation of a surface grinding operation using a vertical axis machine.

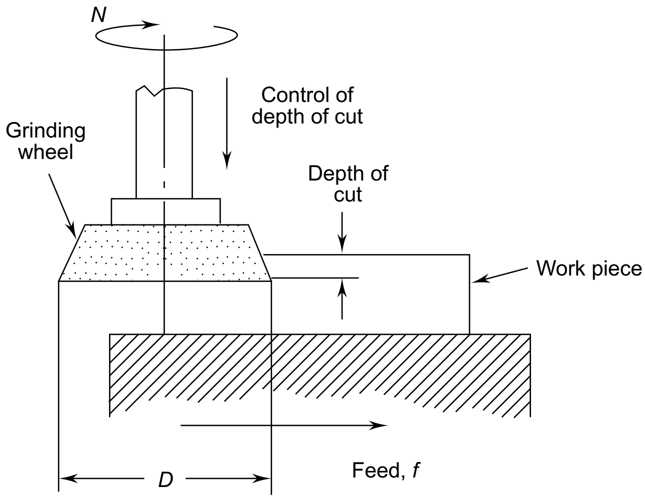


Fig. 9.17 *Grinding operation*

Approach distance for this case is given as

$$A = \frac{D}{2} \qquad \text{for } W = \frac{D}{2} \text{ up to } D$$

$$A = \sqrt{W(D - W)} \qquad \text{for } W < \frac{D}{2}$$

Where W = width of cut

Example 9.2

For the above example, if vertical axis surface grinder is to be used, calculate the grinding time. The wheel to be used is 200 mm in diameter with a wheel thickness of 20 mm.

Solution Given, W = 100 mm, and D = 200 mm

Approach distance, A = 100 mm

$$\text{Total machining time} = \frac{250 + 200}{10 \times 1000} = 0.045 \text{ minutes}$$

9.6 CREEP FEED GRINDING

Creep feed grinding is a new form of grinding operation, different from the conventional grinding process. In creep feed grinding the entire depth of cut is completed in one pass only using very small in feed rates. As shown in Fig. 9.18, this process is characterised by high depth of cut of the order of 1 to 30 mm with low work speeds of the order of 1 to 0.025 m/min. The actual material removal rates calculated from these process parameters are generally in the same range as that of the conventional grinding. However, the idle time (stopping and wheel/table reversal) gets reduced since the grinding operation is completed in one pass.

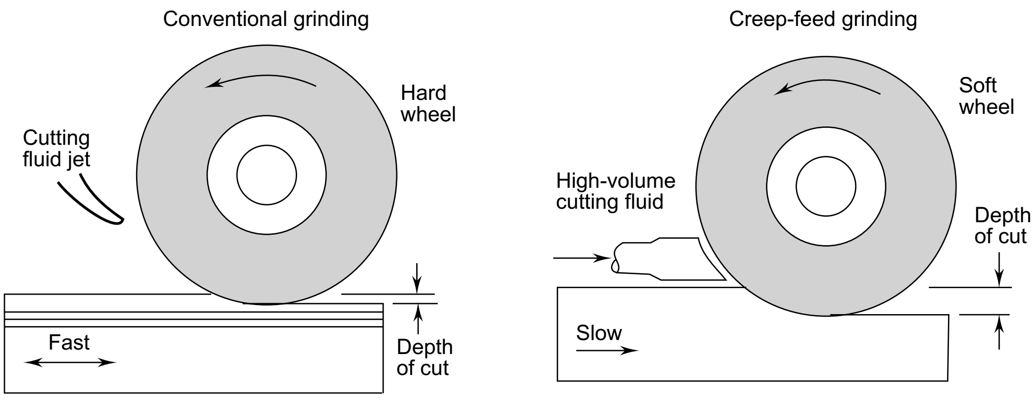


Fig. 9.18 *Creep Feed Grinding operation*

The cutting forces and consequently the power required increases in the case of creep feed grinding, but has a favourable G-ratio. It is necessary to continuously dress the grinding wheel (to reduce the wheel dullness) for efficient operation. This however causes wheel wear and the necessity to adjust the wheel head.

Use soft and open wheels to take care of the wheel dressing and accommodate large volume of chips generated in the process. The grinding wheel speeds used are also low of the order of 18 m/s compared to the 30 m/s used in the conventional grinding operations. Also the in feed rates used are low of the order of 0.005 mm/pass. The grinding fluids used are oil based in view of the low grinding speeds employed. However, the volume of grinding fluid is much more compared to the conventional grinding, in view of the high heats generated in the process. It is possible to achieve higher material removal rates by employing continuous dressing of the grinding wheel using a diamond dresser wheel as shown in Fig. 9.19.

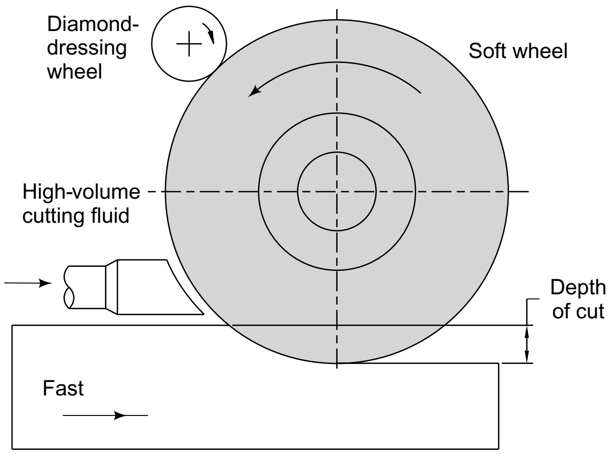


Fig. 9.19 *Creep Feed Grinding operation with continuous dressing*

9.7 HONING

Honing is a low abrading process using bonded abrasive sticks (Fig. 9.20) for removing stock from metallic and non-metallic surfaces. However, it can also be used for external cylindrical surfaces as well as flat surfaces, for which it is rarely used. Commonly it is used for internal surfaces. This is an operation performed as the final operation to correct the errors that result from the previous machining operations. The characteristics that can be achieved by the honing process are:

- Correction of geometrical accuracy
  - Out of roundness
  - Taper
  - Axial distortion
- Dimensional accuracy
- A finish surface pattern is generated by the characteristic motion of the abrasive grains that provide the best possible surface to promote optimum lubricating conditions.

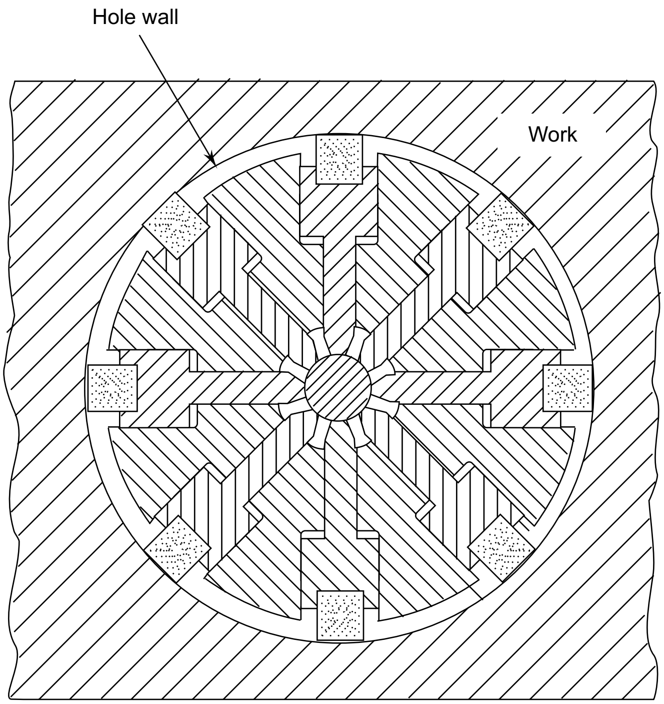


Fig. 9.20 *Honing stick in contact with the work piece*

Abrasive grains are bonded in the form of sticks by a vitreous or resin material and the sticks are presented to the work so that their full cutting forces are in contact with the work surfaces. Since a large number of abrasive grains are presented to the work surface simultaneously, substantial material removal takes place.

For cylindrical surfaces the abrasive grains are given a combination of two motions - rotation and reciprocation. The resultant motion of the grains is a cross hatch lay pattern as shown in Fig. 9.21 with an included angle of 20 to 60°.

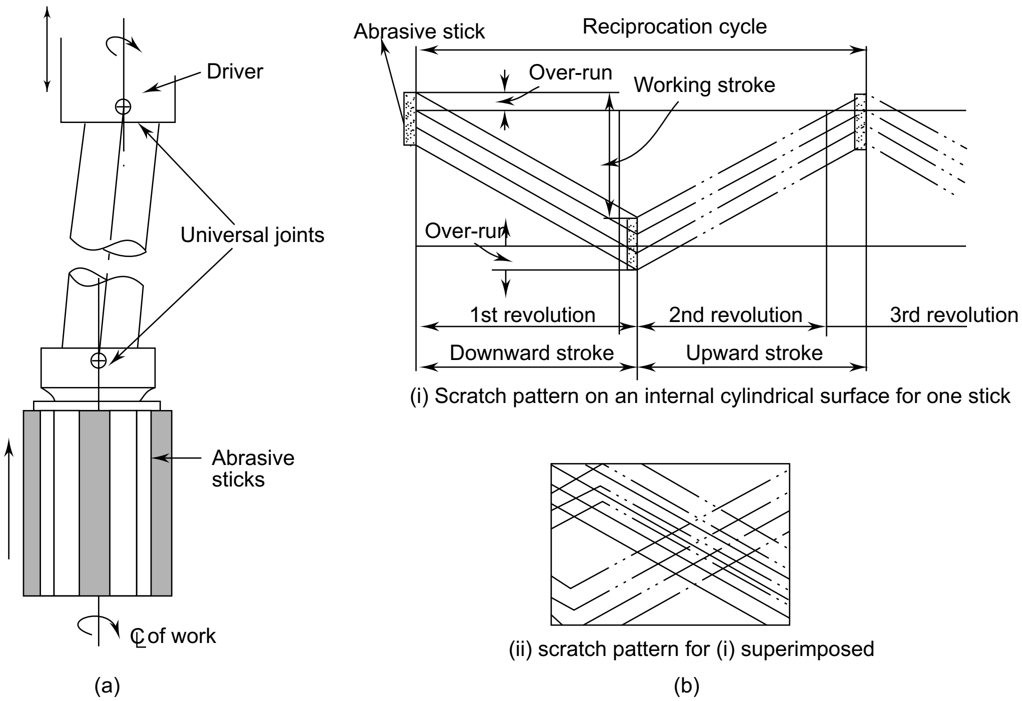


Fig. 9.21 *Honing operation*

The abrasive grains put more pressure on the high spots. After the crests are removed, the bore is made straight. Since a large number of grains are in contact with the total surface, uniform surface finish is obtained. Also the honing force and temperature are never concentrated at any one point. This results in less surface damage compared to other machining processes.

9.7.1 Honing Conditions

All materials can be honed. However, the material removal rate is affected by the hardness of the work material. The typical rates are:

- Soft material 1.15 mm/min on diameter
- Hard materials 0.30 mm/min on diameter

Maximum bore size that can be conveniently honed is about 1500 mm while the minimum size is 1.5 mm in diameter. Honing allowance should be small to be economical. However, the amount also depends upon the previous error to be corrected.

The abrasive and the grain size to be selected depend upon the work material and the resultant finish desired. Table 9.10 specifies some of these conditions for normal usage.

Generally higher cutting speeds are used for metals that shear easily such as cast iron and non-ferrous metals. Alternatively the harder work pieces require lower cutting speeds. Also the rough surfaces that dress the honing stone mechanically allow higher cutting speeds. Speeds should be decreased as the area of abrasive grain per unit area of bore increases. Higher cutting speeds usually result in finer finish. However, they decrease dimensional accuracy, over heating of work piece and dulling of the abrasive. Typical cutting speeds used in honing are given in Table 9.11.

TABLE 9.10 Selection of honing stone characteristics

Work Material	Hardness BHN	Abrasive	Grade	Grain Size for a Surface Finish, μm			
				0.01	0.025	0.3	0.4
Steel	200–300	Al_2O_3	R	600	500	400	320
	330–470		O	600	500	400	320
	50–65 R_C		J	500	400	320	280
Cast iron	200–470	SiC	Q	500	400	280	280
	50–65 R_C	SiC	J	400	280	220	150
Aluminium	120–140	SiC	R	600	500	400	320
Copper	180–200	SiC	R	600	500	400	320
		SiC	R	600	500	400	320

TABLE 9.11 Selection of honing process parameters

Material	Hardness R_C	Honing Speed, m/min			
		Rough Honing		Finish Honing	
		Rotary Speed	Reciprocating Speed	Rotary Speed	Reciprocating Speed
Cast iron	15–20	23–28	10–12	32	13.5
Steel	15–35	18–22	9–11	25	12
	35–60	14–21	12–15	28	17.5
Alloy steels	25–50	23–28	10–12	31	12
Bronze	8–15	21–26	12–26	30	17.5
Aluminium	–	21–26	12–26	31	17

Honing pressures applied are typically about 1.0 to 3.2 MPa.

9.8 LAPPING

Lapping is generally the final finishing operation done with loose abrasive grains. The process is employed to get

- Extreme accuracy of dimension
- Correction of minor imperfection of shape
- Refinement of surface finish
- Close fit between mating surfaces

The service life of components which are in close contact during machining can be greatly increased by the lapping process which removes the valleys and hills present on the machined surfaces. Typical finishes obtained in the lapping process are given in Table 9.12.

Stock removal rates with silicon carbide are generally more compared to aluminium oxide. Correspondingly aluminium oxide gives better surface finish for the same grain size. Softer non-ferrous materials require a finer grain size to produce satisfactory finish comparable with those produced on steel.

Lapping is done by charging a lap made of soft material with abrasive particles and rubbing it over the work piece surface with a slight pressure as shown in Fig. 9.19. Lapping is done manually or by specially designed machines. Pressure is applied on the lap and is moved with the loose abrasive between the lap and the work, removing the material from the work till the work confirms to the profile of the lap. The surface produced is dull in view of the random scratched pattern. Lap materials generally used are cast iron, soft steel, bronze and brass.

Table 9.12 Surface finish achieved in lapping process

Abrasive Used	Grain Size	Surface Finish, µm
Silicon carbide	220	0.75–1.00
	320	0.64–0.75
	400	0.46–0.64
	500	0.38–0.46
	600	0.25–0.38
	800	0.13–0.25
Aluminium oxide	400	0.08–0.13
	800	0.05–0.08
	900	0.03–0.08

In order to achieve uniform abrasion of the work surface, it is necessary to ensure that all the points on the work are subjected to the same amount of abrading by careful manipulation of the lap. For this purpose the manual lap is moved in a figure 8 fashion as shown in Fig. 9.22.

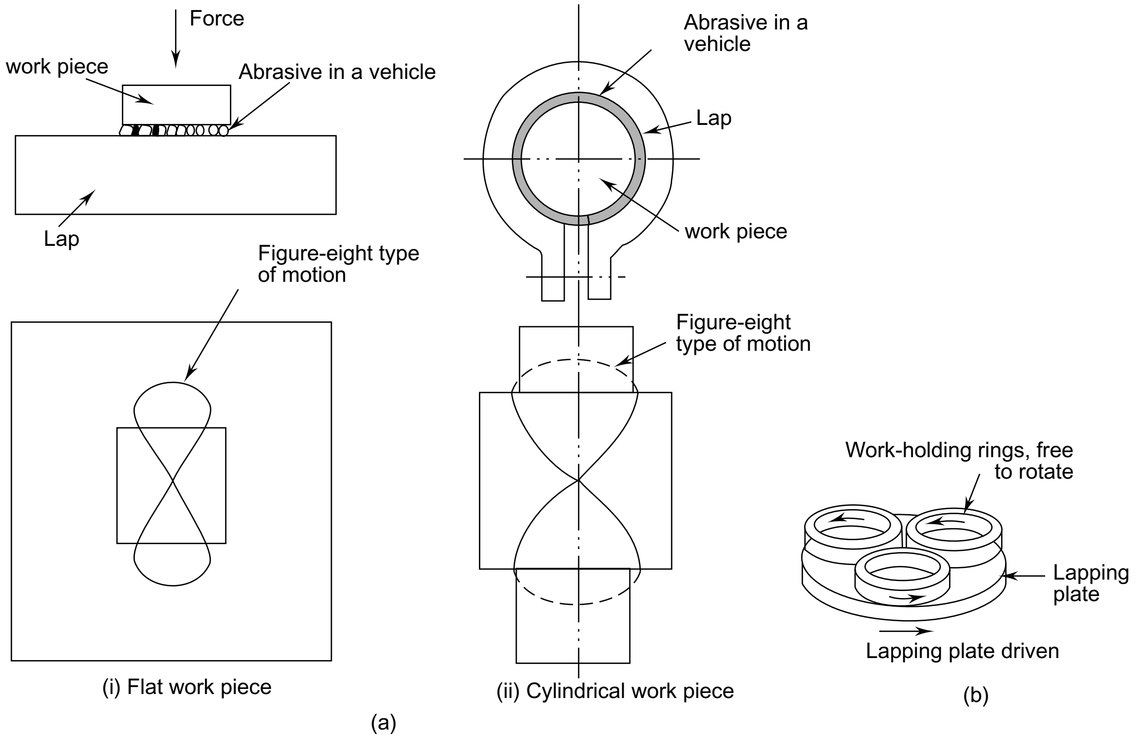


Fig. 9.22 *Lapping operation*

Special lubricants generally called vehicles are used during the lapping process. The desirable properties of fluids used as vehicles are:

1. Abrasive should be held in uniform suspension during the operation.
2. It should not evaporate easily.

3. It should be non-corrosive.
4. It can be easily removed by normal cleaning.

The materials which satisfy the above criteria are water soluble cutting fluids, vegetable oils, mineral oils and greases.

Lapping speed is 100 to 250 m/min. The material removed depends upon the lapping speed. Higher lapping allowances require higher lapping speeds. The lapping pressure applied is 0.01 to 0.03 MPa for soft materials and 0.07 MPa for hard materials. Higher pressures are likely to cause scouring of the work surface. Lapping allowance depends on the previous operation carried and the material hardness. Typical values are given in Table 9.13.

Table 9.13 Lapping allowances

Work Material	Lapping Allowance, mm
Cast iron	0.2
Aluminium	0.1
Soft Steel	0.01–0.02
Ductile Steel	0.05–0.10
Hardened Steel	0.005–0.020
Glass	0.03
Cemented Carbide	0.03–0.05
Bronze	0.03

Lapping can be carried out on flat surfaces as well as any other forms such as cylindrical or any form surfaces. The lap has to match the form surface required.

A few points to be noted with lapping operation are:

- It is more difficult to lap soft metals than hard metals.
- Lap should be softer than the work material.
- The hardness of the abrasive should be based on the hardness of the work material. Softer work materials require softer abrasives and vice versa.
- The lapping medium should preferably be a little viscous to hold the abrasive so that it resists the movement or rolling of abrasive granules to help with the removal of the material.
- The lapping medium should be used sparingly. The increase in lapping medium may increase the material removal rate but not its ability to correct the errors in the part.
- Laps with serrations or grooves are preferable for flat surfaces with large areas while laps with no serration or grooves are preferred for cylindrical lapping.
- To get higher accuracy in lapping use a hard lap that cuts slowly and wears faster with a duller finish. A soft lap on the other hand cuts faster, wears longer and gives a brighter surface.
- As a thumb rule an abrasive particle will produce a scratch that is approximately half its size. For example, a 10 micron abrasive particle will produce a scratch of 5 microns.

9.9 OTHER FINISHING PROCESSES

9.9.1 Super Finishing

Super finishing is another abrasive process utilising either a bonded abrasive like honing for cylindrical surfaces or a cup wheel for flat surfaces. It is generally used for:

- Removing surface fragmentation.
- Reduce surface stresses and burns and thus restore surface integrity.

- Correct inequalities in geometry.
- Produces high wear resistant surface on any object which is symmetrical. Typical surfaces that are surface finished are cylindrical, flat, conical and spherical.

Contact surface in super finishing is large and the tool maintains a rotary contact with the work piece while oscillating as shown in Fig. 9.23. The typical stroke of the super finishing stone is about 1 to 5 mm with an oscillating frequency of 2 kHz. Super finishing speeds used are 10 to 40 m/min while the working pressure maintained is about 0.1 to 0.3 MPa. The heat generated under these conditions is appreciably small and hence there is no metallurgical alteration of the work. The finish obtained on the surface depends upon the time for which the stone is in contact with the work as shown in Fig. 9.24.

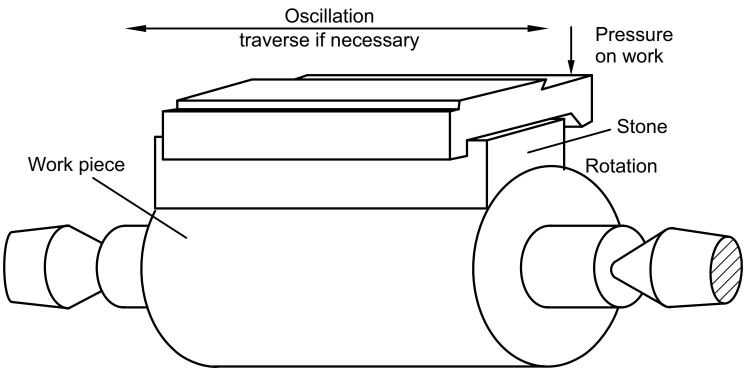


Fig. 9.23 *Typical motions in super finishing operation*

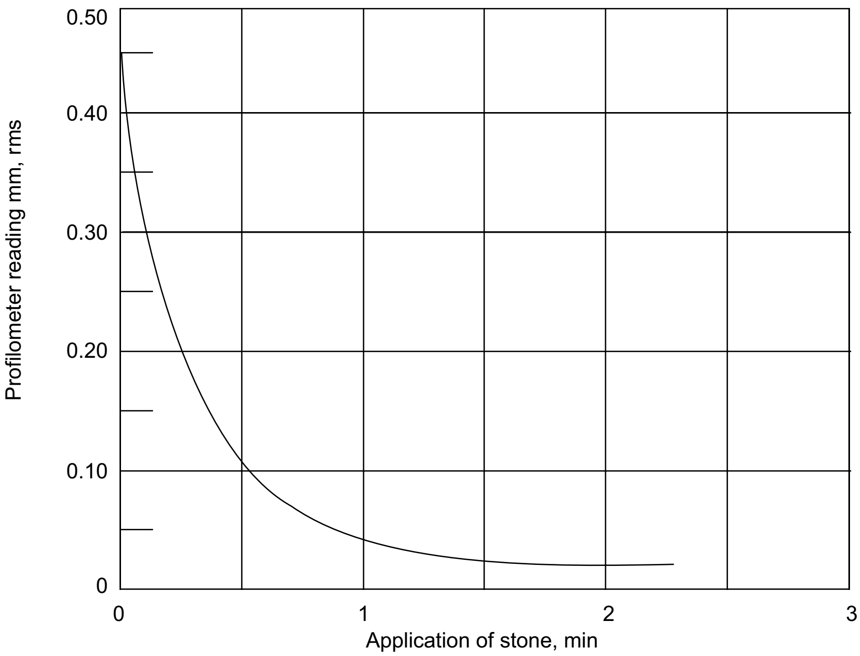


Fig. 9.24 *Variation of surface finish obtained with contact time in super finishing operation*

9.9.2 Polishing and Buffing

Both these processes are used for making the surfaces smoother along with a glossy finish. Polishing and buffing wheels are made of cloth, felt or such material, which is soft and have a cushioning effect. Polishing is done with a very fine abrasive in loose form smeared on the polishing wheel with the work rubbing against the flexible wheel. A very small amount of material is removed in polishing. In buffing the abrasive grains in a suitable carrying medium such as grease are applied at suitable intervals to the buffing wheel. Negligible amount of material is removed in buffing while a very high lustre is generated on the buffed surface. The dimensional accuracy of the parts is not affected by the polishing and buffing operations.

9.9.3 Abrasive Belt Grinding

In this process a continuous moving belt with abrasive is used for grinding the surfaces. The abrasive belt is normally passed between two wheels with one being driven while the other is idling as shown in Fig. 9.25. Use of abrasive belts results in much cooler cutting and rapid material removal rates compared to conventional grinding. The work piece is oscillated across the face of the abrasive belt to obtain a uniform belt wear and surface finish.

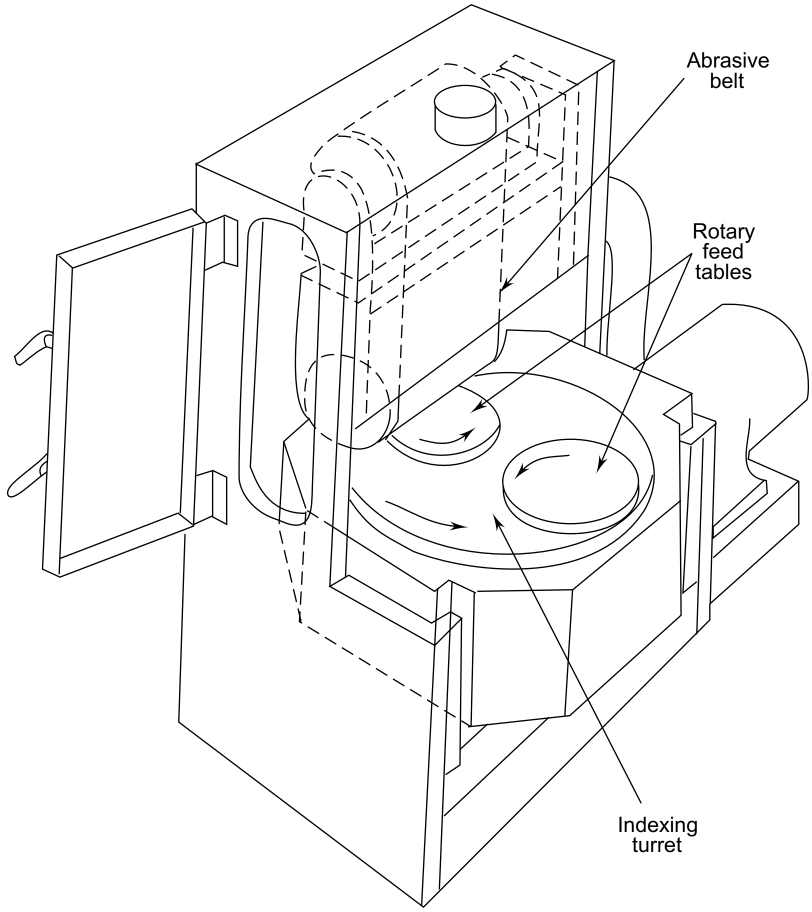


Fig. 9.25 *Abrasive belt grinding operation*

This method is most suitable for flat surfaces. However, cylindrical surfaces can also be belt ground by using a suitable contact wheel. Abrasive belt with a very fine grit may be used for polishing application.

SUMMARY

Grinding is a process utilising bonded abrasives to remove material from very hard surfaces, and in the process generates good surface finish and high dimensional accuracy. It is however inefficient in material removal consuming more energy compared to other processes such as milling.

- Grinding wheel is characterised by the type of abrasive used, as well as its size, bond used and structure.
- Conventional abrasives such as aluminium oxide and silicon carbide are more commonly used for a majority of grinding operations. Cubic boron nitride and diamond are more expensive and hence are used for special applications.
- Grinding wheel need to be properly balanced before operations. Also dressing and truing are the necessary operations carried on the grinding wheel to keep it sharp and maintain the necessary form.
- Cylindrical grinding machine is used for axi-symmetric components, while surface grinders are used for plane surfaces. Surface grinding machines have both horizontal axis as well as vertical axis.
- Centre less grinding machines are used for large volume manufactures to get very high accuracies and faster operations.
- Grinding process parameters are the wheel speed, work speed, traverse speed and in feed need to be chosen properly for a given operation such that faster grinding can be done for the required accuracy and finish.
- Creep feed grinding is an operation used for large material compared to the conventional grinding operation by using soft grinding wheels and large depths of cut.
- Honing is an operation using abrasive sticks to develop very high finish for the inside surface of cylindrical shapes.
- Lapping is an operation used to generate extremely high surface finish on flat surfaces using loose abrasive.

Questions

9.1 What are the various abrasive machining operations you are familiar with? Explain their application and limitations.

9.2 How is grinding different from other machining operations? Explain its applications in view of its capabilities.

9.3 What is the classification method that could be used for the grinding machine? Give the applications of each variety of grinding machine.

9.4 How is the abrasive selected for a grinding operation? Your answer should indicate the reasons for the selection.

9.5 What is the marking system followed in case of grinding wheels? Explain the individual elements of the marking system from the standpoint of the functioning of the wheel.

9.6 What are the grinding process parameters that are of interest? Explain their effect on the grinding performance and the wear rates.

9.7 Describe briefly about creep feed grinding. Mention the method along with the application and the precautions to be taken during its operation.

9.8 What are the various types of surface grinding approaches that are possible? What would be their individual advantages and applications? Give explanations for your reasoning.

9.9 What are the advantages and limitations of using centre less grinding?

9.10 Describe about the dressing and balancing requirement in grinding.

9.11 A high speed steel (70W18Cr4V1 variety) rod of size 50 mm diameter × 250 mm long is to be manufactured to a tolerance of j6. Describe the process to be used with a neat sketch and the suggested process parameters to be used in the manufacturing process. If the same job is to be produced in mass what would be the production process best suited? Explain your answer.

9.12 Describe in details the various arrangements of centre-less grinding with neat sketches. Mention the applications in each case.

9.13 Write short note on the following:
(a) Brazed carbide tools
(b) Grade of a grinding wheel
(c) Geometry of a single point turning tool
(d) Surface grinding machines

9.14 Mention the various types of bonds used in the making of the grinding wheels. Also mention their applications.

9.15 Write short note on the following:
(a) Lapping process
(b) Structure of a grinding wheel

9.16 Describe grinding wheel structure with the help of a neat sketch and state different bonding and abrasive materials used in it. What would you like as an abrasive for grinding steel?

9.17 Give the specifications for the wheel to be employed for external cylindrical grinding of a shaft 50 mm diameter of steel SAE 1020?

9.18 What change in specification should be made for internal grinding of the hole of 50 mm diameter in the same steel? Give reasons for the change suggested.

9.19 What is the marking system followed in case of grinding wheels? Explain the individual elements of the marking system from the standpoint of the functioning of the wheel. Which grinding wheels are used for the following applications?
(a) Rough grinding of C70 steel
(b) Finish grinding of bronze rods
(c) Cup grinding wheels for grinding single point turning tools

9.20 Specify the honing parameters to be considered for good honing practice.

9.21 Differentiate between dressing and truing along with their definitions.

9.22 Compare grinding, honing and lapping operations.

9.23 Give the advantages and limitations of honing and lapping.

Problems

9.1 A flat surface of C70 steel of size 75 × 150 mm is to be ground using a vertical axis surface grinder. The grinding wheel to be used is 200 mm diameter with a thickness of 20 mm. Calculate the machining time required for finishing. Assume any suitable values for the grinding process parameters while justifying. If the same surface is to be ground on a horizontal axis grinding machine with a grinding wheel of 250 mm diameter and wheel thickness of 25 mm, what would be the grinding time? From the two times obtained compare the performance of the two grinding machines regarding their application. [0.6875 min, 0.244 min]

9.2 A high speed steel (70W18Cr4V1 variety) rod of size 22 mm diameter × 250 mm long with a tolerance of *j*6 from a bar stock of 25 mm diameter, is to be manufactured in a lot of 100. Describe the process to be used with a neat sketch and the suggested process parameters to be used in the manufacturing process. If the same job is to be produced in quantities of 50,000 per month, what production process will be best suited? Explain your answer. [Cylindrical grinding]

Multiple Choice Questions

9.1 Grinding is a process used for
- (a) Machining materials which are too hard for other machining processes
- (b) Close dimensional accuracy
- (c) High degree of surface smoothness
- (d) All of the above

9.2 Among the conventional machining processes, the most inefficient process is
- (a) Turning
- (b) Grinding
- (c) Drilling
- (d) Milling

9.3 For grinding steels, the preferred abrasive is
- (a) Silicon carbide
- (b) Aluminium oxide
- (c) Diamond
- (d) Cubic boron nitride (CBN)

9.4 For grinding non-ferrous materials, the preferred abrasive is
- (a) Silicon carbide
- (b) Aluminium oxide
- (c) Diamond
- (d) Cubic boron nitride (CBN)

9.5 In grinding to get good surface finish, the grain size of the abrasive to be used is
- (a) Coarse
- (b) Fine
- (c) Grain size does not affect surface finish
- (d) None of the above

9.6 In grinding the grade of the grinding wheel should be chosen based on the hardness of the work material.
- (a) Yes. Hard grinding wheels are chosen for hard materials
- (b) Yes. Soft grinding wheels are chosen for hard materials
- (c) Yes. Soft grinding wheels are chosen for soft materials
- (d) No. Hard grinding wheels are preferred over soft wheels for all types of materials

9.7 In a cylindrical grinding machine
- (a) Grinding wheel rotates while the work does not rotate
- (b) Both grinding wheel and the work rotate in the same direction and same speed
- (c) Both grinding wheel and the work rotate in the opposite directions
- (d) Both grinding wheel and the work rotate in the same direction but work rotates slowly compared to the grinding wheel

9.8 The grinding machine suitable for grinding large flat areas is
(a) Surface grinder with a horizontal spindle and rotating table
(b) Surface grinder with a vertical spindle and rotating table
(c) Surface grinder with a horizontal spindle and reciprocating table
(d) Surface grinder with a vertical spindle and reciprocating table

9.9 The grinding machine suitable for grinding cylindrical work pieces without actually fixing them in the machine is
(a) Cylindrical grinding machine
(b) Surface grinding machine
(c) Centre less grinding machine
(d) Internal grinding machine

9.10 One of the main advantages of centre less grinding is
(a) There is no need for having and maintaining centres and centre holes
(b) Less grinding allowance may be required, because the out-of-roundness is corrected across the diameter rather than the radius.
(c) Work pieces may often be loaded into the machine by the automatic feeding devices.
(d) All of the above

9.11 One of the main disadvantage of centre less grinding is
(a) Setup time for a centre less grinding operation is usually large.
(b) Grinding allowance required is large.
(c) Work pieces cannot be fed automatically
(d) Work piece is not properly supported

9.12 Higher grinding wheel speeds cause
(a) Increased chip size
(b) Increased wheel wear
(c) Chip size is not affected
(d) Decreased wheel wear

9.13 Higher feed rates in grinding cause
(a) Increased chip size
(b) Increased wheel wear
(c) Improve surface finish
(d) Decreased wheel wear

9.14 Higher work speeds in grinding cause
(a) Increased heat produced
(b) Increased wheel wear
(c) Improve surface finish
(d) Decreased wheel wear

9.15 The main difference between creep feed grinding and the conventional grinding operation is
(a) In creep feed grinding hard grinding wheels are used
(b) In creep feed grinding the entire depth of cut is completed in one pass only using very small in feed rates
(c) In creep feed grinding higher feed rates are used
(d) In creep feed grinding higher work speeds are used

9.16 The characteristics of the work piece produced by honing is
(a) Out of roundness is corrected
(b) Higher dimensional accuracy
(c) Axial distortion is corrected
(d) All of the above

9.17 The characteristics of the work piece produced by lapping is
(a) Correction of minor imperfection of shape
(b) Extreme accuracy of dimension
(c) Close fit between mating surfaces
(d) All of the above

9.18 The abrasive process that uses a loose abrasive grit is
(a) Honing
(b) Lapping
(c) Grinding
(d) Creep feed grinding

Answers to MCQs

9.1 (d)	9.2 (b)	9.3 (b)	9.4 (a)	9.5 (b)
9.6 (b)	9.7 (d)	9.8 (b)	9.9 (c)	9.10 (d)
9.11 (a)	9.12 (d)	9.13 (b)	9.14 (b)	9.15 (b)
9.16 (d)	9.17 (d)	9.18 (b)		

CASE STUDY

ABRASIVE FLOW MACHINING (AFM)

Nearly 15% of the manufacturing cost is spent on finishing objects. Traditional finishing processes are developed taking the part geometry into account in providing the type of motion to the finishing tool. As a result, these processes are limited to the geometries for which it is designed and is difficult to adopt for other geometries. Also manual finishing methods are usually difficult to automate. Further internal features are difficult to finish by the traditional finishing methods. Small precision parts in hard and difficult-to-machine materials such as super alloys, ceramics, refractory materials, carbides, semiconductors, quartz, composites etc. requires special finishing methods. Abrasive Flow Machining (AFM) was developed by Extrude Hone Corporation of USA in 1960s for such purposes. The objective of this process is to produce nano level finish on the machined components. AFM is characterized by high volumes of production and ease of finishing areas that are difficult to access. In this process, the abrasive medium is passed back and forth between two opposed cylinders (Fig. 1). Abrasive action is obtained when the flow of the medium is restricted. Performance of AFM varies with the abrasive medium. Highly viscous fluids are used for larger passages while lower viscosities are fine for contour finishing. AFM produces surfaces of consistent quality compared to other finishing methods. AFM machines are available at different extrusion pressures ranging from 7-220 bar with over 380 L/min flowrate. Productivity of the process can be improved using dies with more than one passages for the abrasive medium. Usually, the machines with higher MRR would require some sort of cooling mechanism.

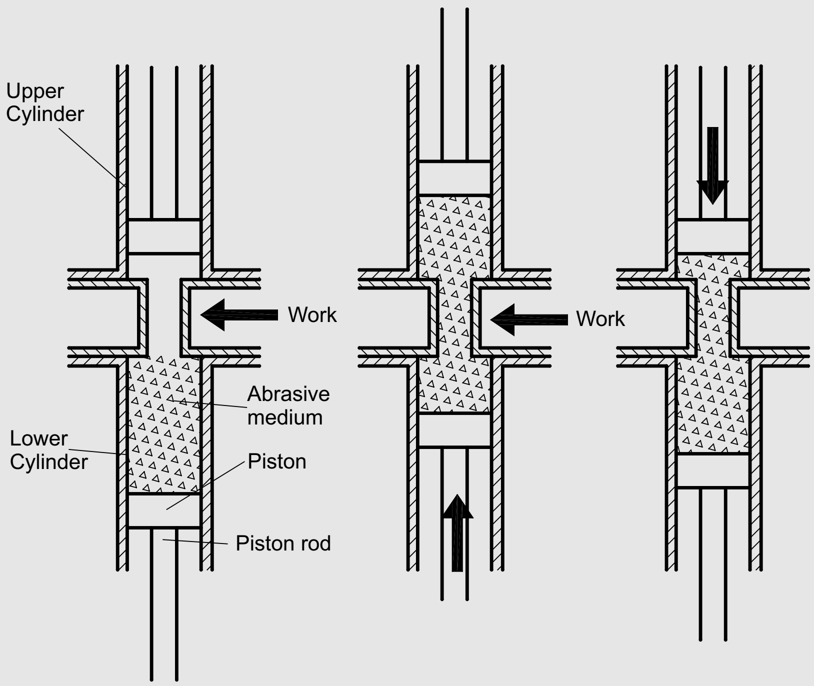


FIGURE 1 *Abrasive Flow machining process*

Another advantage of AFM is that only simple fixtures are sufficient. For finishing external surface, the abrasive medium is forced between the external surface of the work piece and the internal surface of fixture. Use of such simple fixtures makes AFM advantageous for finishing complex objects like turbine parts. A turbine engine fuel spray nozzle when processed with AFM, the surface produced is of highly predictable quality with very small dimensional reductions. Typically, about 90% improvement in surface roughness is obtained with 10% reduction in stock. AFM can be used for parts with difference sizes, typically ranging from 0.2 mm to 50 mm. AFM is used in finishing MEMS components such as micro-channels, fuel injectors, and micro filters, ink-jet printer nozzles, as well as industrial components such as gears. AFM offers various advantages and is well suited for the present scenario of automation.

1. Rhoades, L. (1991). Abrasive flow machining: a case study. *Journal of Materials Processing Technology*, *28*(1-2), 107-116.

CHAPTER 10

Other Machine Tools

Objectives

In addition to the various general purpose machines tools that were discussed so far, a few more machine tools which are used for very specific applications, but not necessarily in the mainstream of the manufacturing activity are discussed in this chapter. After completing this chapter, the reader will be able to

- Understand the sawing operation and various machines used for the purpose
- Understand the broaching operation and various machines used for the purpose
- Understand the gear cutting operation and various machines used for the purpose

10.1 SAWING

Sawing is one of the basic machining operations carried out in a narrow cutting zone through the successive removal of chips by the teeth on a saw blade. The teeth represent the cutting edges. Each of the teeth removes a part of the chip, which is contained in the chip space of the saw blade, till the tooth comes out of the material as shown in Fig. 10.1.

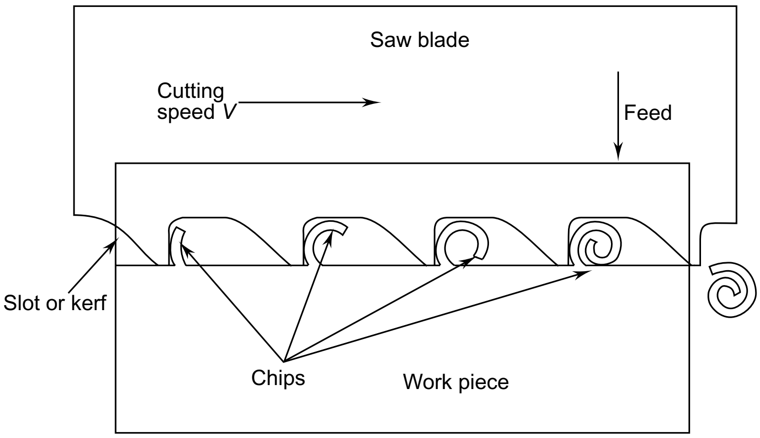


Fig. 10.1 *Saw blade in action*

Sawing is one of the most economical processes because of the removal of a very small amount of material which consumes less power and at the same time is able to cut large sections. The process can be very easily automated thereby reducing the labour cost as well.

10.1.1 Saw Blades

Saw blades are made either from carbon steel or high speed steel. The larger blades will have only the teeth portion made of high speed steel while the main portion of the blade is made of low cost alloy steel. The blade strip is electron beam welded to the main portion thereby reducing the overall cost.

Saw blades are basically of three types depending upon the applications for which they are used.

Hack saw blade Hack saw blade is in the form of a strip ranging in length from 250 to 600 mm and widths ranging from 12.5 to 50 mm. The teeth space is about 1.0 to 6.0 mm. These are used in hand hack saws as well as the power hack saw machines described later.

Band saw blades These are long and continuous and generally formed into a closed band with teeth on one edge.

Circular saw This is a large circular blade similar to the slitting saw used in milling.

Besides the type of saw blades as described earlier, the characteristics of cutting depend upon a number of geometric factors of the teeth. They are:

Tooth form The saw teeth can be simple straight teeth as shown in Fig. 10.2(a) or with an undercut-face-tooth with a positive rake angle as shown in Fig. 10.2(b). Though the form (b) is better for cutting action in view of the positive rake angle, it is difficult to obtain with smaller teeth. The straight tooth form is used for high feeding while the undercut-face-tooth type is used for coarse feeding.

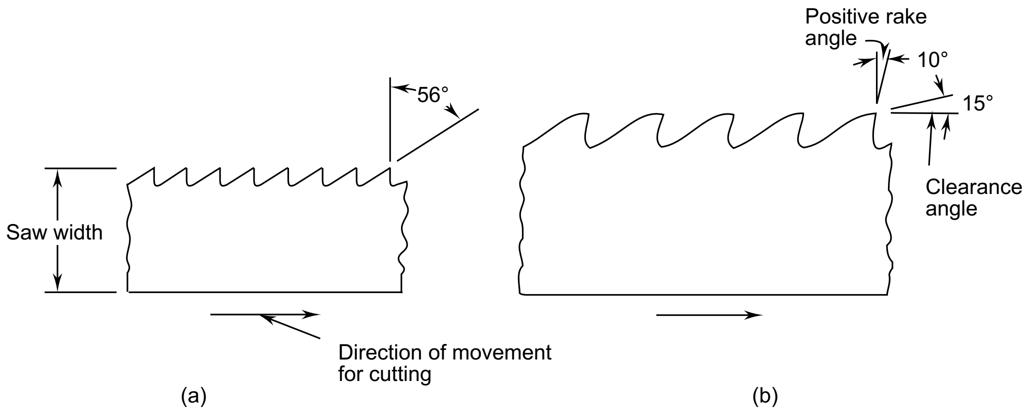


Fig. 10.2 *Commonly used saw tooth forms*

Tooth spacing It is very important since it determines the size of teeth as well as the amount of chip space available. Large tooth spacing gives more space for the chips as well as strong teeth. Tooth spacing also determines how many teeth are in contact with the work piece at any given time. Ideally, at least 4 teeth should be in contact with the work piece during sawing. This problem becomes acute when dealing with the sawing of thin stock such as pipes. If only one tooth is in contact with the work piece in such cases, there is a possibility that the teeth may be stripped from the saw blade. Generally coarse pitch is used for wider materials and fine pitch for narrow materials as shown in Fig. 10.3. Also for sawing softer materials coarse pitch blades are used since they produce long chips, while fine pitch blades are used for sawing harder materials.

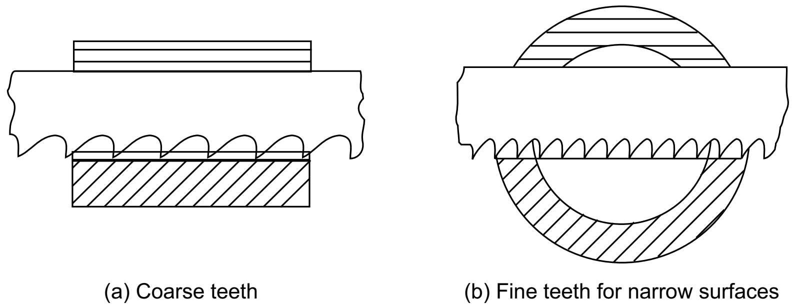


Fig. 10.3 *Effect of saw tooth pitch while sawing materials of different widths*

Tooth set This refers to the way the teeth are offset with respect to the centre line of the blade width as shown in Fig. 10.4. The tooth set makes the saw cut, termed as kerf, wider to facilitate the easy movement of the saw blade for cutting. The straight set as shown in (a) is used for non-ferrous materials and plastics. The raker set is used for general purpose machining of ferrous materials. The wavy set is used in the fine pitch tooth blades for cutting thin sheets and tubes.

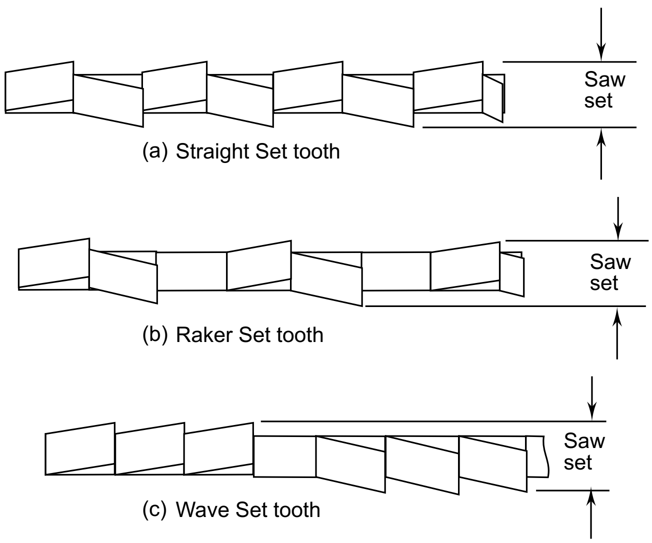


Fig. 10.4 *Different types of tooth sets used in sawing*

Typical circular saw blade teeth are shown in Fig. 10.5. These consist of large disks with the teeth cut at the periphery. Small sizes will have the teeth directly cut into the disk while the larger sizes have segmented or inserted teeth as shown in Fig. 10.5. The teeth are made of high speed steel or tungsten carbide while the disk is made of low alloy steel. Their metal removal rate is high and also produce smooth surface comparable to that of slitting saws used in milling.

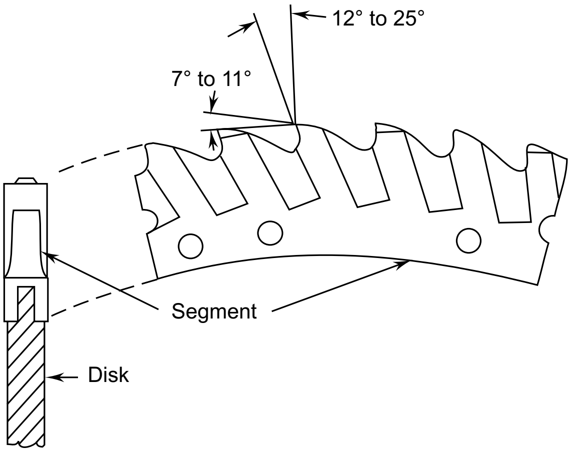


Fig. 10.5 *Typical circular saw blade construction*

10.1.2 Sawing Machines

The various types of sawing machines used are:

- Hack saw
  - Manual
  - Power
- Band saw
  - Vertical
  - Horizontal
  - Contour
- Circular saw

Power Hack Saw

The power hack saw uses the hack saw blade. The blade is mounted in the hacksaw frame and reciprocated for the sawing operation as shown in Fig. 10.6. These are very simple machines with a tool frame for holding the saw and some work holding devices similar to a vice. The reciprocating motion is inherently inefficient because no cutting takes place during the return stroke.

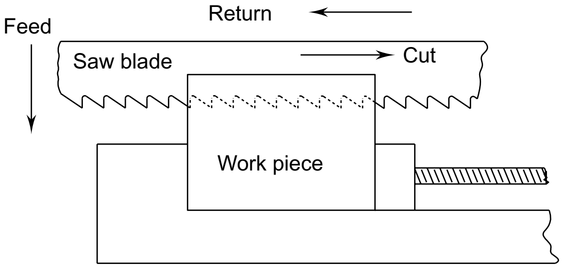


Fig. 10.6 *Power hack saw in action*

Band Saw

These basically have a continuous band of saw blade rotated between two disks such that the cutting action will be continuous unlike the power hack saw. Typical construction of a vertical band saw machine is shown in Fig. 10.7. These are generally used for cutting off single stationary work pieces that can be held on to the table of the band saw. The saw blade can be tilted up to 45° to permit cutting at any angle.

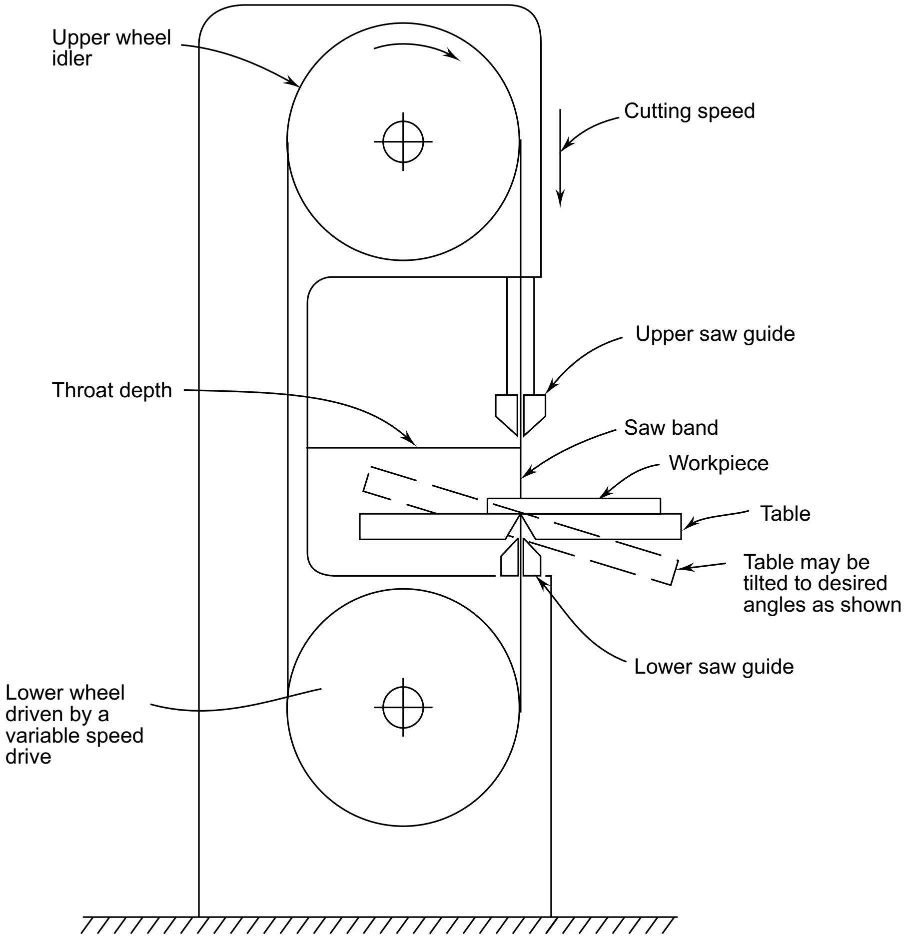


Fig. 10.7 *Vertical band saw*

The band saw operates continuously such that the cutting force is always directed against the table. This is relatively safer compared to the hack saw and can cut work pieces without even clamping them to the table.

Contour band saw machines are similar to band sawing machines and are used for sawing of any predefined contours in the work piece. The contour need not start from the edge of the work piece, but can start anywhere inside the work piece. In these machines the saw band can be broken and then inserted through a predrilled hole in the work piece. After that the blade is butt welded into a continuous circle and sawing is completed. After completing the sawing operation, the band is again broken to remove the work piece and the cycle continued. Also, these have swivel tables which permits angular cutting required for die blocks. These machines are equipped with flash butt welder, annealing and grinding units for the purpose of welding the saw blade.

Circular Saw

A typical circular sawing operation is shown in Fig. 10.8. These have the ability to run the saw at very high cutting speeds up to about 130 m/s and large feed rates. The stock can be cut very quickly and therefore care has to be taken for the selection of the parameters to maximise the productivity.

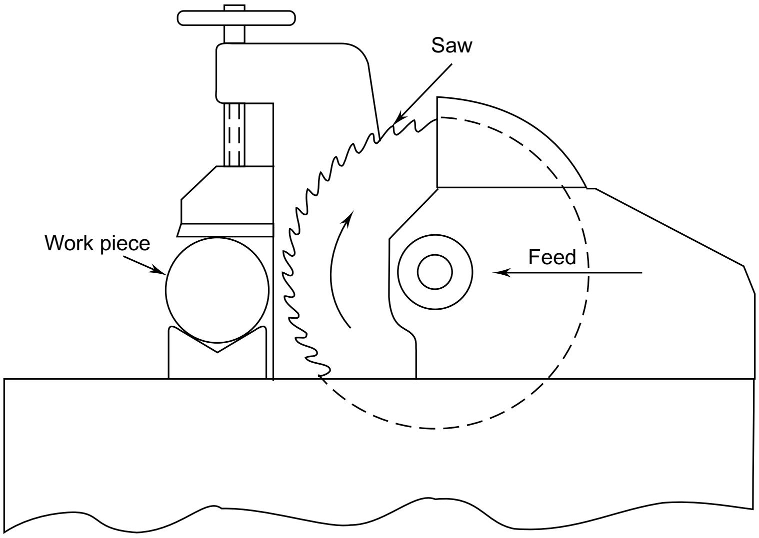


Fig. 10.8 *A circular sawing machine*

10.2 BROACHING

Broaching is a multiple-tooth cutting operation with the tool similar to the sawing operation described earlier. The similarities end there, since in broaching the machining operation is completed in a single stroke as the teeth on the cutting tool called broach, are at a gradually increasing height corresponding to the feed per tooth of a milling cutter. The material removal using the broach teeth is shown schematically in Fig. 10.9. Here the dotted line indicates the amount of material removed by the successive individual teeth. Using broaching, Tolerances of the order of ±0.013 mm can be easily obtained. Very high surface finish is not the aim of broaching, and as a result it is possible to get 0.80 μm with normal broaching. A better finish can be obtained by sacrificing tool life and employing expensive fixtures.

Since each successive tooth removes only a small material at a time, the tool life of a broach is very high. It is possible to produce 2,000 to 10,000 accurate parts with a broach, since sharpening depends on the work piece and tool materials used.

Some typical examples of internal shapes that can be very conveniently machined by using the broaching process are shown in Fig. 10.10.

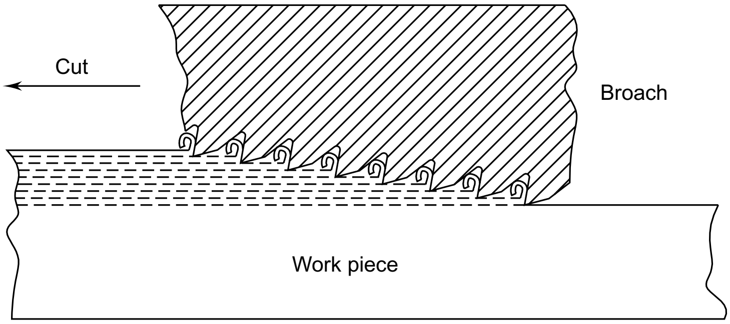


Fig. 10.9 *Cutting action of individual teeth in a broach*

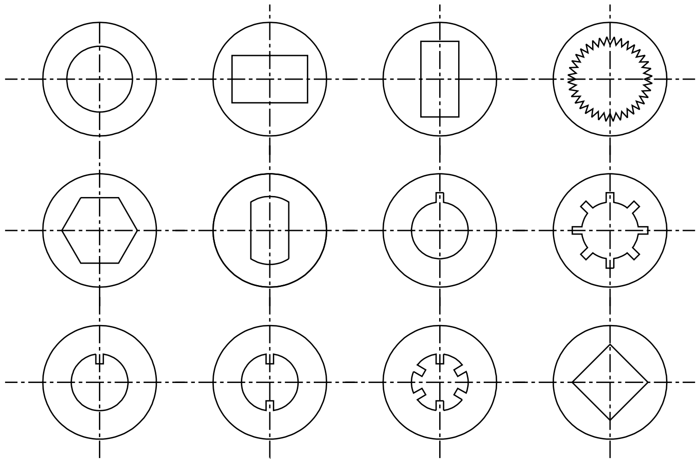


Fig. 10.10 *Some typical internal profiles that can be broached*

10.2.1 Broach Construction

Though in the above figure the material being removed is shown in an exaggerated fashion for clarity, in actual broach, there are a large number of teeth each of which will removes a small amount of material such that by the time the broach completes the operation the component is completely machined. A typical broach is shown in Fig. 10.11. Broaching was originally developed for machining internal keyways, but looking at the advantages, has been extensively used in the mass production of automobile component manufacture and for various other surfaces as well.

The broach is composed of a series of teeth, in which each tooth can be considered as a single tool. Each tooth of a broach is slightly higher than the previous one. This rise per tooth determines the depth of cut and also the material removed by the tooth. There are basically three sets of teeth present in a broach as shown in Fig. 10.10. The roughing teeth that have the highest rise per tooth will remove bulk of the material. The semi-finishing teeth whose rise per tooth is smaller will follow this. Hence, they remove relatively smaller amounts of material compared to the roughing teeth. The depth of cut can vary from approximately 0.15 mm for roughing teeth in machining free-cutting steel to a minimum of 0.025 mm for finishing teeth.

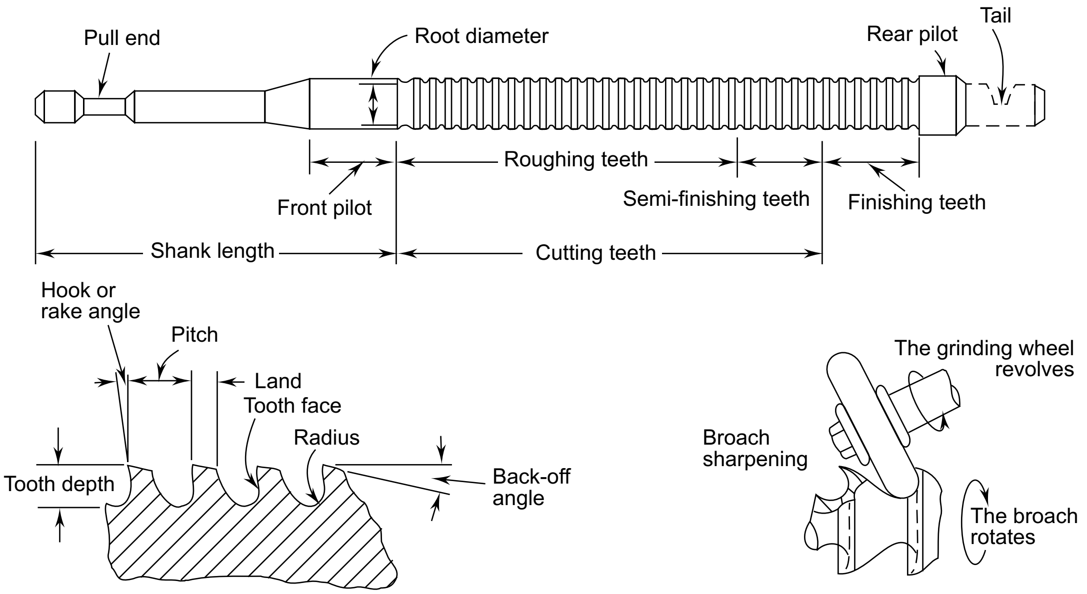


Fig. 10.11 *Typical construction of a pull broach*

The last set of teeth is called the finishing or sizing teeth. Very little material will be removed by these teeth. The necessary size will be achieved by these teeth and hence all the teeth will be of the same size as that required finally. With the progress of time, when the first teeth wears out, then the next teeth and so on, will be able to provide the sizing function.

The face of the tooth is ground with a certain hook (rake) angle depending upon the work piece material. Soft steel work pieces usually require greater hook angles; hard or brittle materials, smaller hook angles. The land is required to support the cutting edge against stresses. A slight back-off angle is ground onto the land to reduce friction (Fig 10.11).

If the broach tooth engagement is irregular, vibrations may be caused. The depth of cut taken by each tooth called the tooth rise is calculated so that it does not impose too great a strain on individual teeth. A large tooth rise increases power requirements while if it is too small causes glazed or galled finish.

The pitch as shown in Fig 10.11 determines the length of cut and chip thickness that a particular broach can handle.

$$\text{Pitch, } P = 0.35\sqrt{\text{Length of cut}}$$

The total stock, D_s to be removed is distributed among the teeth uniformly as depth of cut per tooth, D_T, and then the length of broach, L_B can be given as

$$\text{Length of broach, } L_B = \left(\frac{D_S}{D_T} + Z_f\right)P$$

Where Z_f = the number of teeth required for finishing the operation and is assumed to be 4 or 5 teeth.

The broaching time, T_B in min = $\dfrac{L}{1000 \times V_f} + \dfrac{L}{1000 \times V_r}$

where L = Length of stroke, mm;

V_f = Cutting speed in the forward stroke, m/min;

V_r = Return speed, m/min.

The pull end of the broach (Fig. 10.11) is attached to the pulling mechanism of the broaching machine with the front pilot aligning the broach properly with respect to the work piece axis before the actual cutting starts. The rear pilot keeps the broach in a square position, as it leaves the work piece after broaching.

Broaching speeds are relatively low of the order of 6 to 15 m/min. However the production rate is high with the cycle times about 5 to 30 seconds including the work piece and tool handling times. The low cutting speeds are conducive to very high tool lives with very small tool wear rates.

Broaches are generally made of high speed steel in view of its high impact strength. Sometimes the titanium nitride coating helps to improve the tool life further. Also the carbide insert type broaches are used for surface broaching of cast iron for very large volume production to reduce the frequent re-sharpening of the broach, which is a very difficult operation.

Standard broaches are available for common and more often used forms such as round and square holes, keyways, etc. These are relatively inexpensive compared to the custom designed broaches, which will have to be used for other specific designs.

The very high cutting forces in broaching require the use of proper fixtures to support the cutting force. Also since broaching is used for mass production, high cost of fixturing can be easily justified. The general principles followed for broaching fixture is similar to those of other fixtures as given in Chapter 14.

Advantages of broaching

1. It is the fastest way of finishing an operation with a single stroke.
2. Since all the machining parameters are built into the broach, very little skill is required from the operator.
3. Broaching machine is simple since only a single reciprocating motion is required for cutting.
4. Final cost of the machining operation is one of the lowest for mass production.
5. Any type of surface, internal or external, can be generated with broaching.
6. Many surfaces which are very difficult or impossible by other means can be done by broaching. For example, square hole and internal splines can be easily produced by broaching.
7. Good surface finish and fine dimensional tolerances can be achieved by broaching, often better than boring or reaming.

Limitations of broaching

1. Custom made broaches are very expensive and can therefore be justified only for very large volume production.
2. A broach has to be designed for a specific application and can be used only for that application. Hence the lead time for manufacture is more for custom designed broaches.
3. Broaching being a very heavy metal removal operation requires that the work piece is rigid and capable of withstanding the large forces.

4. Broaching can only be carried out on the work piece whose geometry is such that there is no interference for the broach movement for the cutting.

10.2.2 Broaching Machines

There are basically four different types of broaching machines as follows:

- Push broaching machines,
- Pull broaching machines,
- Surface broaching machines, and
- Continuous surface broaching machines.

Broaching machines are specified mainly using the main parameters:

- The force applied on the cutting tool by the machines in tonnes or kN, e.g. 1000 kN
- The maximum length of stroke of the ram, e.g. 450 mm which provides an indication of the maximum length of cut that can be taken.
- Various ranges of cutting speeds available
- Various ranges of feeds available
- The type of power source utilized
- Physical dimensions of the machine

Push Broaching Machines

These are generally used for internal surfaces where the broach movement is guided by a ram. These machines are simple, since the broach only needs to be pushed through the component for cutting and then retracted. The work piece is fixed into a boring fixture on the table. Even simple arbour presses can be used for push broaching.

Pull Broaching Machines

These are a little more complex in terms of operation. These consist of a work holding mechanism, broach pulling mechanism along with a broach elevator to help in the removal and threading of the broach through the work piece. The work piece is mounted in the broaching fixture and the broach is inserted through the hole present in the work piece. Then the broach is pulled through the work piece completely. Then the work piece is removed from the table. Afterwards the broach is brought back to the starting point before a new work piece is located on the table. The same cycle will then be repeated.

The power for the pulling mechanism and the movement of the broach are provided generally by hydraulic fluid. Because of the inner surface broaching, it is necessary to remove the broach from the pulling mechanism to thread it through the work piece hole in every cycle.

Surface Broaching Machines

Surface broaching is relatively simple since the broach can be continuously held and then it will carry out only a reciprocating action. A typical surface broaching machine is shown in Fig. 10.12. The work piece is held in the fixture while the surface broach is reciprocated with the ram on the vertical guideways on the column. In Fig. 10.12 is shown the progressive action surface broach with the teeth segments distributed into three areas A, B and C. The progressive action reduces the maximum broaching force, but results in a longer broach.

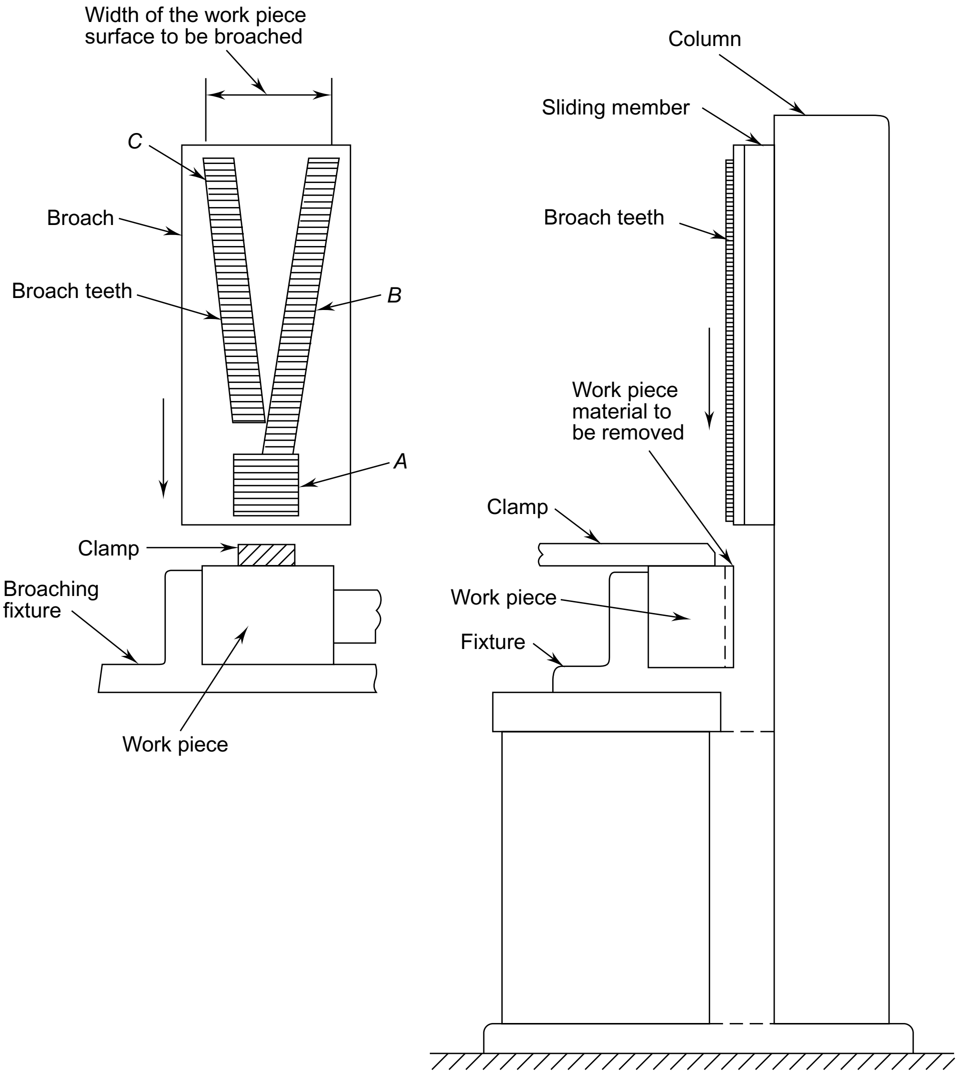


Fig. 10.12 *Surface broaching machine*

Continuous Surface Broaching Machines

The reciprocation of the broach always involves an unproductive return stroke, which is eliminated in a continuous surface broaching machine. In this the small work pieces are mounted on the broaching fixtures, which are in turn fixed to a continuously moving conveyor as shown in Fig. 10.13. Broaches, which are normally stationary, are kept above the work pieces. The work pieces are pushed past the stationary broaches by means of the conveyor for cutting. The work pieces can be loaded and unloaded onto the conveyor manually or automatically. These machines are used for mass production.

Typical examples of jobs that can be done by surface broaching are shown in Fig. 10.14.

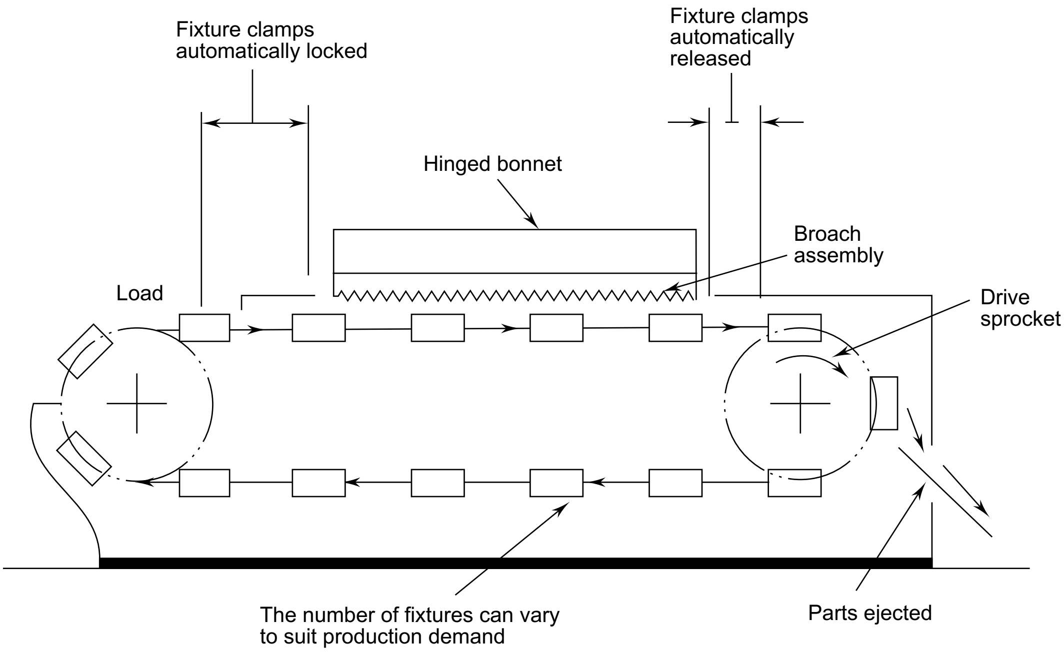


Fig. 10.13 *Continuous surface broaching machine*

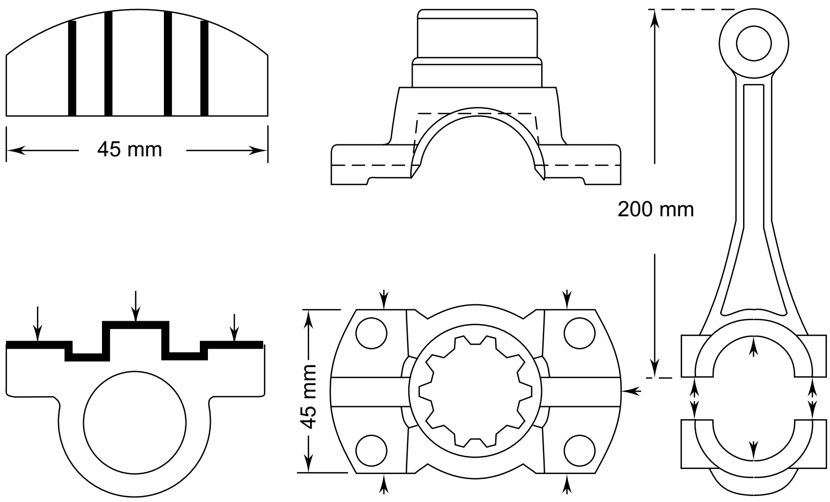


Fig. 10.14 *Examples of parts made by surface broaching*

10.3 GEAR CUTTING

Gears are important machine elements that transmit power and motion. There is a large variety of gears used in industrial equipment as well as a variety of other applications. The gear teeth have the contact surface as an involute curve along which they roll and slide while transmitting the motion. An involute curve is generated by unwinding a tautly held string from a base circle. The involute curve is chosen because it is simple, easy to manufacture and allows the centre lines to be varied between the mating gears.

Pitch circle, which is an imaginary circle on the gear, corresponds to the diameter of the wheel. For proper operation, the two pitch circles must be tangential at the point where the centre line connecting the two centres of rotation intersects the pitch circle. To reduce the friction, gears are designed such that the teeth have rolling motion rather than the sliding motion. Gear tooth size is identified as module. Some important parameters of gears that are relevant for gear manufacture are given below. The rest of the details will be found in gear design books.

$$\text{Pitch diameter} = \text{No. of teeth} \times \text{module}$$
$$\text{Tooth thickness} = 0.5 \times \pi \times \text{module}$$
$$\text{Total depth} = 2.25 \times \text{module}$$

Spur gear is the most common and easy to make. It has straight teeth on the periphery of a cylinder. It transmits motion between two parallel shafts. A rack is a gear with infinite radius, having the teeth that lie on a straight line. These are used for converting rotary motion to a straight line or vice versa.

10.3.1 Gear Forming

This is a process of machining gears using a form milling cutter in a milling machine. Gears can be cut using a form milling cutter, which has the shape of the gear teeth. The form milling cutter called 'dp' (diametral pitch, used in inch systems, which is equivalent to inverse of module) cutters have the shape of the teeth similar to the tooth space with the involute form of the corresponding size gear. The commercial gear milling cutters are available as a set for a given module or diametral pitch. These can be used on either horizontal axis or vertical axis milling machines, though horizontal axis is more common. The vertical axis cutters will be similar to end mills with tooth profile on the cutting edge.

The cutting tool as shown in Fig. 10.15 is fed radially into the work piece till the full depth is reached. Then the work piece is fed past the cutter to complete machining of one tooth space.

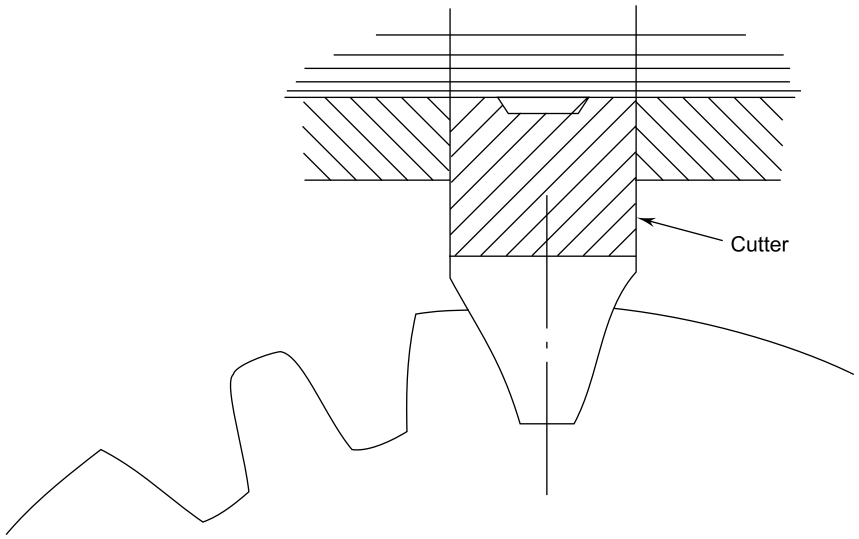


Fig. 10.15 *Form milling for spur gears*

The work piece is actually mounted on the dividing head. The machining process is repeated after indexing gear blanks by one tooth. Sometimes it may be necessary to index by more than one tooth since the work material gets heated by the heavy material removal, which may affect the accuracy of the surface generated.

The tooth space actually depends upon the number of teeth. Accordingly a range of cutters with numbers 1 to 8 are available to cover the entire range of gears to be machined as shown in Table 10.1. This is made possible by approximating the tooth profile by considering the fact that profile variation is relatively small between the gears with close range teeth. For achieving closer accuracy, addition of 7 cutters to account for the intermediate range of gears are also available, as shown in Table 10.1 in the third and fourth columns.

Table 10.1 Dp cutters used for form milling

Cutter Number	Range of Teeth that can be Cut	Cutter Number	Range of Teeth that can be Cut
1	135 to Rack	1.5	80 to 134
2	55 to 134	2.5	42 to 54
3	35 to 54	3.5	30 to 34
4	26 to 34	4.5	23 to 25
5	21 to 25	5.5	19 to 20
6	17 to 20	6.5	15 to 16
7	14 to 16	7.5	13
8	12 to 13		

Though form milling of gears is relatively common process in machine shops, it is suitable for small volume production, particular for one off quantities. With a smaller number of cutters, it is possible to produce a large range of gears. The process is also suitable for producing spur, helical and worm gears. For small volumes the process is economical.

However, the accuracy of the gear is dependent upon the accuracy of the dividing head as well as the profile accuracy of the cutter for the given gear. Thus it is not to be used for very high accuracy requirements. Further this is a slow process. Internal gears cannot be produced by this process.

10.3.2 Gear Generation

To obtain more accurate gears, the gear is generated using a cutter, which is similar to the gear with which it will be meshed, by following the general gear theory. As a result, one cutter can theoretically generate all gears regardless of the number of teeth in the manufactured gear. All the teeth will have the correct profile as well as mesh correctly. The gears produced by generation are more accurate and the manufacturing process is also fast. These are used for large volume production.

There are two types of gear generation methods that are commonly used:

- Gear shaping, and
- Gear hobbing

10.3.3 Gear Shaping and Planing

To understand the concept of gear shaping, imagine a gear blank, which has a periphery that is very soft and easily deformable. An ordinary involute gear is pressed into the rim of the gear blank until the two pitch circles are in contact. Then the gear blank is rolled together with the gear such that the pitch circles roll on each other without slipping. Since the rim of the gear blank is soft the gear teeth will be pressed and theoretically correct teeth will be formed on the gear blank. The teeth so formed will mesh with any other involute gear of the same module regardless of the number of teeth.

In actual gear shaping operations, the gear cutter is not actually pressed but removes the material by reciprocation similar to a vertical shaper. The gear shaper cutter, which is very similar to the gear, but the teeth are form relieved to act as the cutting edges as shown in Fig. 10.16.

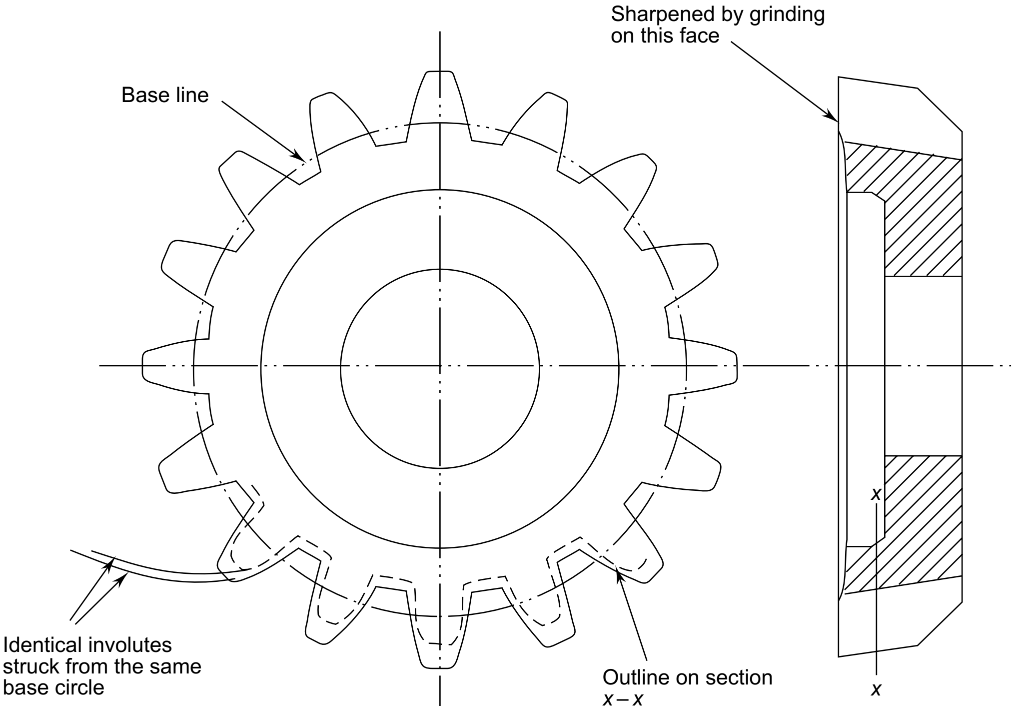


Fig. 10.16 *Schematic representation of gear shaping operation*

The gear shaper cutter is mounted on a vertical ram and is rotated about its axis as it performs the reciprocating action. The work piece is also mounted on a vertical spindle as shown in Fig. 10.16 and rotates in mesh with the shaping cutter during the cutting operation. The relative rotary motions of the shaping cutter and the gear blank are calculated as per the requirement and incorporated with the change gears.

The cutter will slowly move into the gear blank surface with incremental depths of cut, till it reaches the full depth. The cutter and gear blank are separated during the return (up) stroke and comes to the correct position during the cutting (down) stroke.

Gear shaping can cut internal gears, splines and continuous herringbone gears that cannot be cut by the other processes. This process can also cut gears close to a shoulder with very small clearance. The length of stroke of a gear shaper limits the maximum width of gear that can be produced. However since the approach and over travel are small, this process is fast for narrow gears. Otherwise, the production rate of a gear shaper is lower compared to the other generating process, i.e., gear hobbing. For producing helical gears special oscillating motion has to be provided on a gear shaper for the reciprocating of the cutter.

The process described above is called the 'Fellows gear shaping' process since it is invented by E. R. Fellows. There is another variation of the same process, termed as gear planing. In the 'Sunderland gear planing' process a rack cutter is used in place of a pinion cutter, as in Fellows gear shaping. The rack will move tangential to the gear blank while reciprocating. Since the length of the rack is limited, it needs to be periodically indexed to bring the rack back to its starting position. This has a facility for the vertical axis to be swivelled in the vertical plane such that single helical gears can be produced without any additional mechanisms.

10.3.4 Gear Hobbing

Gear hobbing is a continuous process eliminating the unproductive return motion of the gear shaping operation. To understand the concept of gear hobbing, an analogy similar to that used in gear shaping is given. Imagine an involute rack being pressed into the gear blank with a very soft rim up to the point when the pitch circles of the rack and the gear blank meet. During this process the rack is moved length-wise while the gear blank is rotated such that it rolls with the rack without slipping. Theoretically correct tooth profiles would be formed on the gear blank rim as it is very soft. The number of teeth formed depends upon the size of the gear blank used.

The above procedure may be difficult to implement practically. It therefore is modified in the form of a gear cutter called hob, where the rack is wound round the cylinder in a helix like a worm as shown in Fig. 10.17. Since the involute rack teeth are straight lines, the hob threads are easier to produce and inspect. Gashes will be cut across the threads as shown in Fig. 10.17 to provide the cutting edges. These cutting edges are relieved behind for the clearance similar to a form milling cutter. Rotation of the hob causes the theoretical rack to move along a straight line.

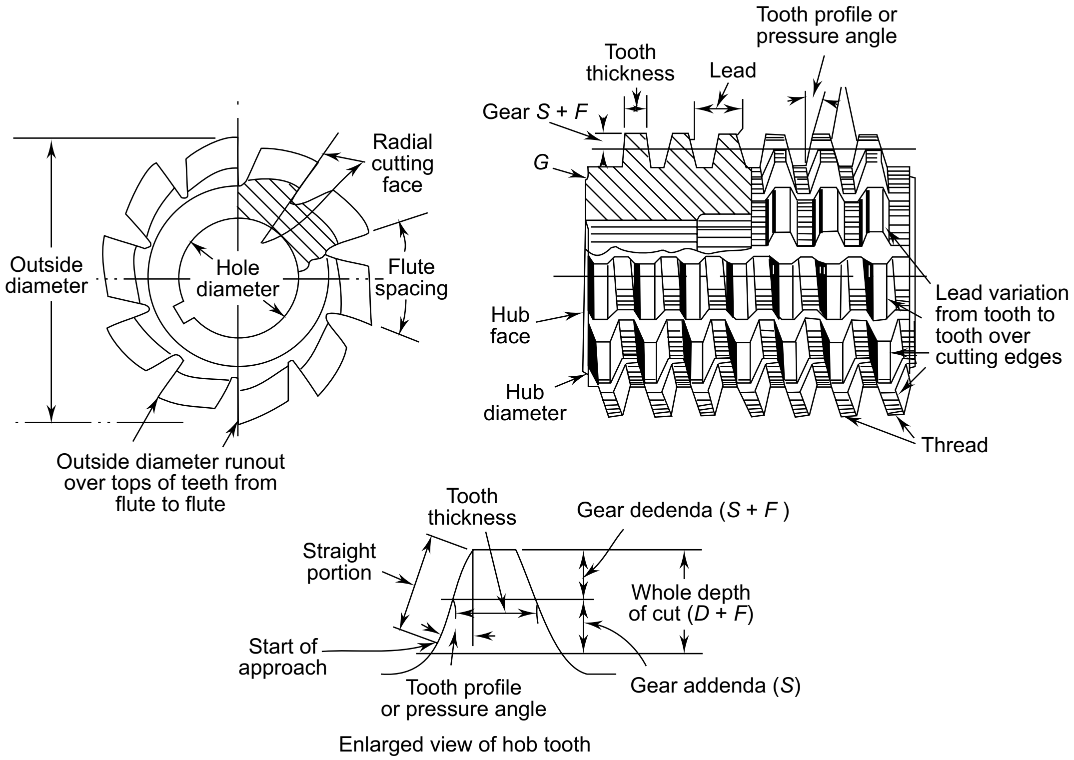


Fig. 10.17 *A typical gear hob with its elements*

A hob may have one, two or more threads of cutting edges. When a gear blank of *N* teeth is cut with a single thread hob, the blank turns a complete revolution when the hob completes N revolutions. For a multi-threaded hob, the gear blank rotates as many times more as the number of starts. Thus a multiple threaded hob is much faster in production compared to a single threaded hob. However, the single threaded hobs are more accurate compared to the multi-threaded ones.

The work piece is mounted on a vertical axis and rotates about its axis. The hob is mounted on an inclined axis whose inclination is equal to the helix angle of the hob as shown in Fig. 10.18. This brings the gear blank teeth in the same plane as that of the gear hob teeth, which is termed as the generating plane. The hob is rotated in synchronisation with the rotation of the blank and the hob is slowly moved into the gear blank till the required tooth depth is reached in a plane above the gear blank. Then the hob is fed slowly in the axial direction of the gear blank till the complete tooth face width is achieved.

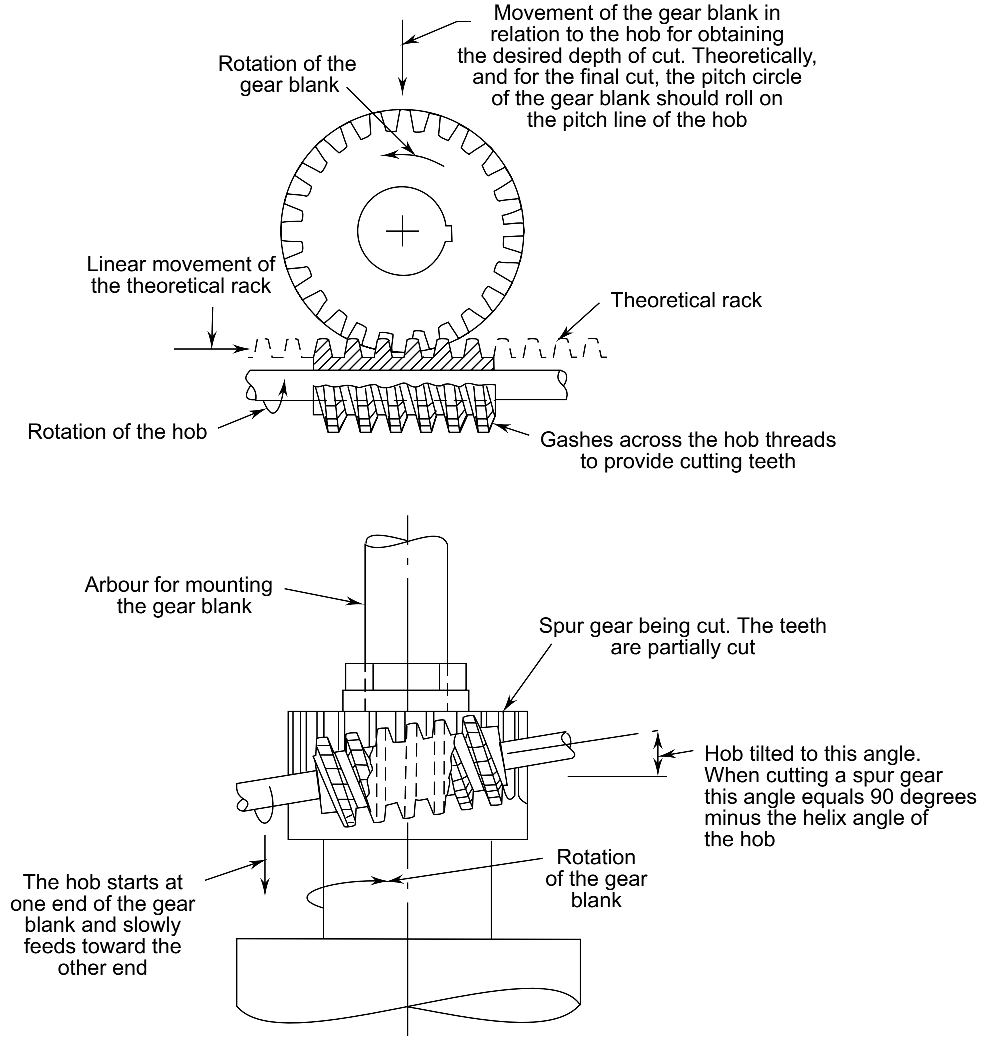


Fig. 10.18 *Schematic representation of gear hobbing operation*

For hobbing helical gears, the hob is swivelled by an additional angle to the helix angle of the gear to be made. Hobbing process though used for gears and splines more frequently, is also suitable for other shapes such as ratchets and sprockets.

Since hobbing is a continuous process, it is fast, economical, and the most productive gear machining process. It is also possible to mount more than one gear blank in the work axis to increase production rate. However, this process cannot be used for machining internal gears or gears with shoulders and flanges, because of the clearance needed for the hob.

SUMMARY

These are some special machine tools that are used for specific applications and are not as common as the other machine tools that were discussed so far.

- Sawing is an operation performed with a saw blade having evenly spaced teeth on the periphery. This is the most economical operation for cutting-off of parts in view of the simplicity of the machine.
- Broaching is also similar to sawing with multi-tooth tool that reciprocates the cutting. However broaching is used to make specific profiles of interest rather than cutting. This is mostly used for internal details which are difficult to produce economically by other methods. This process is suitable only for mass production in view of the high cost of the tool.
- Gear forming is used for cutting gears in small quantity. It is an approximate method of making gears.
- Gear hobbing and shaping are the generating methods to cut accurate gears. These processes are used for large volume production.

Questions

10.1 Why is sawing the most economical cutting operation?

10.2 Describe the important features of saw teeth.

10.3 What are the various types of sawing machines used in the industry? Give their applications.

10.4 Give the applications of the following from the application point of view in sawing:
(a) Tooth set, (b) Tooth spacing

10.5 What is the importance of saw tooth form with respect to the application of sawing?

10.6 What is the difference between form cutting and generation of gears with respect to the principle?

10.7 List the motions the gear milling cutter has with respect to the work piece.

10.8 What is a DP cutter? Why is it used for gear milling?

10.9 What are the characteristics of DP cutters used for involute gear cutting?

10.10 In form cutting of gear, the cutter is changed (different number for the cutter is used) whenever the number of teeth on the gear to be produced falls in a different range. Is it true? Give the reasons for your answer.

10.11 Give a comparison of the various gear machining methods in terms of their application, accuracy and process.

10.12 What are the various gear manufacturing methods you are familiar with? Give brief explanation about them.

10.13 Sketch a gear shaping cutter and label its elements.

10.14 Sketch a gear hob and label its elements.

10.15 Explain the principle of gear shaping.

10.16 Explain the principle of gear hobbing.

10.17 Compare the gear shaping and gear hobbing giving the process and product requirements.

10.18 Explain briefly the cutting action of a broach. What are the factors that contribute to the increased production rate in broaching?

10.19 Describe the various broaching machines used in industry. List their typical applications, advantages and limitations.

10.20 Describe about the continuous broaching machine and explain the type of component surfaces for which it is useful.

10.21 Give a simple sketch of a broaching tool and explain various elements.

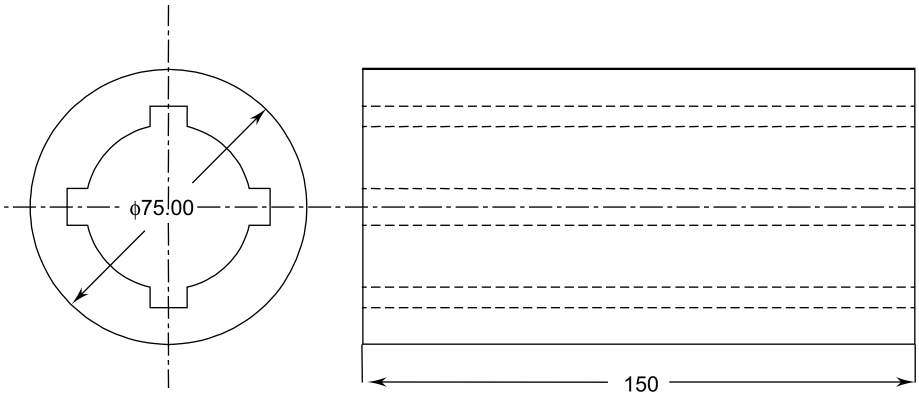
10.22 Give the reasons for robust fixtures in broaching.

10.23 What are the parameters related to a broach that are important? Explain how the length of a broach is determined.

10.24 How is the milling cutter used for gear cutting re-sharpened?

10.25 Compare broaching operation with that of any other metal machining operation for the purpose of generating constant inside contours. Show sketches of some example jobs done using broaching.

10.26 A hollow steel tube with 4 slots along the axis is to be machined. Specify all the possible processes to be used for machining the internal details. Your answer should clearly indicate the application where each of the process is applicable, with reasons.



All dimensions in mm
C35 steel
All tolerances +0.050

Fig. 10.19

Multiple Choice Questions

10.1 In sawing operation, the saw blade large (coarse) tooth spacing is used
(a) For sawing soft materials
(b) For sawing hard materials
(c) For thin materials
(d) For non-ferrous materials

10.2 You need to cut a piece of 20 mm diameter C20 CRS steel into four equal length pieces. The original stock is 550 mm long. If the saw blade produces a 1.5 mm kerf, how long will each piece of material be after cutting?
(a) 137.500 mm
(b) 136.375 mm
(c) 136.000 mm
(d) 135.000 mm

10.3 In sawing operation, the saw blade with straight tooth set is used for
(a) Ferrous materials
(b) General purpose work
(c) Non-ferrous materials
(d) None of the above

10.4 Which of the following machining operations does **not** utilise reciprocating action for cutting metal
(a) Broaching
(b) Sawing
(c) Turning
(d) Shaping

10.5 Which of the following machining operations utilises reciprocating action for cutting metal
(a) Broaching
(b) Turning
(c) Gear hobbing
(d) Milling

10.6 Which of the following machining operations can be used only with very large volume manufacture because of the high cost of the tooling used
(a) Broaching
(b) Sawing
(c) Gear hobbing
(d) Slotting

10.7 Advantage of broaching operation is
(a) It is the fastest way of finishing an operation with a single stroke.
(b) Final cost of the machining operation is one of the lowest for mass production.
(c) Broaching can do many surfaces that are very difficult or impossible by other methods. For example, square holes and internal splines.
(d) All of the above

10.8 Disadvantage of broaching operation is
(a) It is the slowest way of finishing an operation.
(b) Overall cost of machining with a broach is very high compared to other machining operations
(c) Broaching can only be carried out on the work piece whose geometry is such that there is no interference for the broach movement for the cutting.
(d) Final dimensional tolerances are poor compared to other machining operations

10.9 Which of the following operations produces a gear which is not very accurate
(a) Gear forming
(b) Gear shaping
(c) Gear hobbing
(d) Gear planing

10.10 Internal gears cannot be produced by which process.
(a) Gear shaping
(b) Gear hobbing
(c) Gear forming
(d) Gear planing

10.11 Which of the following is **not** a gear generating process
(a) Gear shaping
(b) Gear hobbing
(c) Gear forming
(d) Gear planing

10.12 The process used for cutting a gear on a milling machine is called
(a) Gear forming
(b) Gear shaping
(c) Gear hobbing
(d) Gear planing

Answers to MCQs

10.1 (b)	10.2 (b)	10.3 (c)	10.4 (c)	10.5 (a)
10.6 (a)	10.7 (d)	10.8 (c)	10.9 (a)	10.10 (c)
10.11 (c)	10.12 (a)			

CHAPTER 11

Unconventional Machining Processes

Objectives

Unconventional machining processes were developed initially to machine very hard materials that are almost impossible to be machined economically by the conventional methods. After completing this chapter, the reader will be able to

- Understand the need for inventing the unconventional processes, and their range of applications
- Understand the principles and applications of electric discharge machining
- Understand the principles and applications of electrochemical machining
- Understand the principles and applications of ultrasonic machining
- Understand the principles and applications of chemical machining
- Understand the principles and applications of laser beam machining
- Understand the principles and applications of abrasive water jet machining
- Understand the principles and applications of electron beam machining
- Understand the principles and applications of ion beam machining
- Understand the principles and applications of plasma arc machining

11.1 NEED FOR UNCONVENTIONAL PROCESSES

Conventional machining processes utilise the ability of the cutting tool to stress the material beyond the yield point to start the material removal process. This requires that the cutting tool material is harder than the work piece material. New materials which have high strength to weight ratio, heat resistance and hardness such as nimonic alloys, alloys with alloying elements such as tungsten, molybdenum, and columbium are difficult to machine by the traditional methods. Machining of these materials by the conventional methods is very difficult as well as time consuming since the material removal rate reduces with an increase in the work material hardness. Hence the need for development of non-traditional machining processes which utilise other methods such as electro-chemical processes for the material removal. As a result, these processes are termed as unconventional or non-traditional machining methods.

Besides these the complex shapes in these materials are either difficult to machine or time consuming by the traditional methods. In such cases, the application of the non-traditional machining processes finds extensive use. Further in some applications a very high accuracy is desired besides the complexity of the surface to be machined. These processes are not meant for replacing the conventional processes, but are in fact supplements to them.

There a number of processes available in this category. They are:

- Electric Discharge Machining (EDM)
- Electro Chemical Machining (ECM)
- Electro Chemical Grinding (ECG)
- Ultrasonic Machining (USM)
- Laser Beam Machining (LBM)
- Chemical machining (CHM)
- Abrasive Water Jet Machining (AWJM)
- Water Jet Machining (WJM)
- Plasma Arc Machining (PAM)

The conventional metal cutting processes makes use of the shearing process as the basis for material removal. However, the non-traditional processes depend on a number of other factors such as the vaporisation of the metal, electrolytic displacement, chemical reaction and mechanical erosion.

A comparison of the various processes is given in Table 11.1 for comparison purpose.

TABLE 11.1 **Comparison of the non-traditional machining processes**

Process	Material Removal Rate, mm^3/min	Tolerance, mm	Surface Finish, CLA, μm	Surface Damage, Depth μm	Corner Radius mm	Power Watts
USM	300	0.0075	0.2–0.5	25	0.025	2 400
AJM	0.8	0.0500	0.5–1.2	2.5	0.100	
ECM	1500	0.0500	0.1–2.5	5	0.025	100 000
CHM	15	0.0500	0.4–2.5	50	0.125	
EDM	800	0.0150	0.2–12.5	125	0.025	2 700
EBM	1.6	0.0250	0.4–2.5	250	2.500	150
LBM	0.1	0.0250	0.4–1.25	125	2.500	2
PAM	75 000	0.1250	Rough	500	—	50 000
Milling	50 000	0.0500	0.4–5.0	25	0.050	3 000

The main reasons for using the non-traditional machining processes are:

- High strength alloys: The hardness of the work material is often higher than the cutting tool material or sometimes it becomes necessary to use the machining process on hardened material. In such cases the electro-chemical processes described would be required.
- Complex surfaces: At times very complex surfaces in three dimensions need to be produced, such as those in moulds and dies where the work piece surface being hardened tool steel, it would be difficult to be processed by the conventional means.
- Higher accuracies and surface finish: The accuracy and surface finish desired in hard work piece materials require the conventional machining to be done very slowly, as well as adding a number of finishing processes, making the process very slow and uneconomical.
- Difficult geometries: In addition to the complex geometries, sometimes it is required to produce difficult geometries such as long holes with length to diameter ratio approaching 100, or very small

size holes such as those with less than 0.1 mm in diameter, which are almost impossible to be produced by the conventional methods.

- Automation

The relative economic comparison of these processes is given in Table 11.2.

TABLE 11.2 **Economic comparison of unconventional processes**

Process	Capital Investment	Tooling	Power Required	Efficiency	Tool Consumption
USM	Low	Low	Low	High	Med.
AJM	V. Low	Low	Low	High	Low
ECM	V. High	Med.	Med.	Low	V. Low
CHM	Med.	Low	High	Med.	V. Low
EDM	Med.	High	Low	High	High
EBM	High	Low	Low	V. High	V. Low
LBM	Med.	Low	V. Low	V. High	V. Low
PAM	V. Low	Low	V. Low	V. Low	Low
Milling	Low	Low	Low	V. Low	Low

11.2 ELECTRIC DISCHARGE MACHINING

11.2.1 Introduction

Erosion of craters left by electric discharges on the cathode surface was first discovered by Joseph Priestley, an English theologian and chemist in 1766. His observations are:

"June the 13th, 1766. After discharging a battery, of about forty square feet, with a smooth brass knob, I accidentally observed upon it a pretty large circular spot, the centre of which seemed to be superficially melted. (...) After an interruption of melted places, there was an intrie and exact circle of shining dots, consisting of places superficially melted, like those at the centre."

"June the 14th, 1766. (...) Examining the spots with a microscope, both the shining dots that formed the central spot, and those which formed the external circle, appeared evidently to consist of cavities, resembling those on the moon, as they appear through a telescope, the edges projecting shadows into them, when they were held in the sun."

Since then arcs have been used for a variety of purposes and not necessarily for the removal of metal. The first use of arc for removing metal was attempted by the Russian scientists Boris and Natalya Lazarenko at the Moscow University in 1943. They were investigating the wear caused by sparking between tungsten electrical contacts which was critical for maintenance of automotive engines during the Second World War, under an assignment from the Soviet government. They had noted during the experimentation that the sparks were more uniform and predictable in oil than in air. They then came up with the idea of using this controlled sparking as a method of removing metal. They developed the "Lazarenko circuit" which is a relaxation circuit based on the resistance-capacitance circuit that remained the standard EDM generator for years (Fig. 11.1). The tool and work piece are immersed in a dielectric fluid and connected to a DC power supply. The capacitor gets charged from the DC power supply until it is greater than the breakdown potential of the gap between the tool and the work piece, at which time it will be discharged as an arc removing material from the work piece and the tool. They have found this process to be extremely useful for machining hard metals such as tungsten or tungsten carbide.

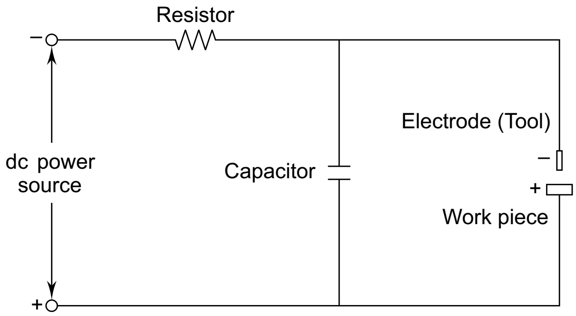


Fig. 11.1 *Relaxation circuit as used by Lazarenko*

Interest in the spark machining process was ignited and a number of studies in the process were initiated during the 1950s. Machine tool industries from Switzerland were involved very early in this market, with Agie that was founded in 1954, while les Ateliers des Charmilles produced their first machine in 1955. They are still the major players in the EDM technology.

The wider adoption of the process could not materialize during this period due to the poor quality of electronic components. However in the 1960's, the development of the semi-conductor industry permitted considerable improvements in EDM machines. Die-sinking machines became reliable and produced surfaces with controlled quality. Also with a large amount of research that went into the process, it became possible to know the process capability with reasonable accuracy. Then with the availability of numerical control in the 1960's and early 1970's, the movements of electrodes became much more precise. With the development of the microprocessor in the 1970's and the development of Computer Numerical Controlled systems (CNC) further improved the performance of EDM. Thus though EDM is generally considered as a non-traditional method because of the nature of the material removal process, because of its wide adoption particularly by the tool rooms, it has become a common place in all manufacturing shops. Excellent historical study of the EDM process is given by SME.

11.2.2 Principle

It has been recognised for many years that a powerful spark will cause pitting or erosion of the metal at both the anode (+) and cathode (–), e.g. automobile battery terminals, loose plug points, etc. This process is utilised in Electric Discharge Machining (EDM). This process is also called spark machining or spark erosion machining. The EDM process involves a controlled erosion of electrically conductive materials by the initiation of rapid and repetitive spark discharges between the tool and work piece separated by a small gap of about 0.01 to 0.50 mm. This spark gap is either flooded or immersed in a dielectric fluid. The controlled pulsing of the direct current between the tool and the work produces the spark discharge.

Initially the gap between the tool and the work piece, which consists of the dielectric fluid, is not conductive. However, the dielectric fluid in the gap is ionised under pulsed application of DC as shown in Fig. 11.2, thus enabling the spark discharge to pass between the tool and the work. Heat transfer from the spark to both tool and the work piece melts, partially vaporises and partially ionises the metal in a thin surface layer. Due to the inertia of the surrounding fluid, the pressure within the spark becomes quite large and may possibly assist in 'blasting' the molten material from the surface leaving a fairly flat and shallow crater. The amount of metal removed per spark depends upon the electrical energy expended per spark and the period over which it is expended.

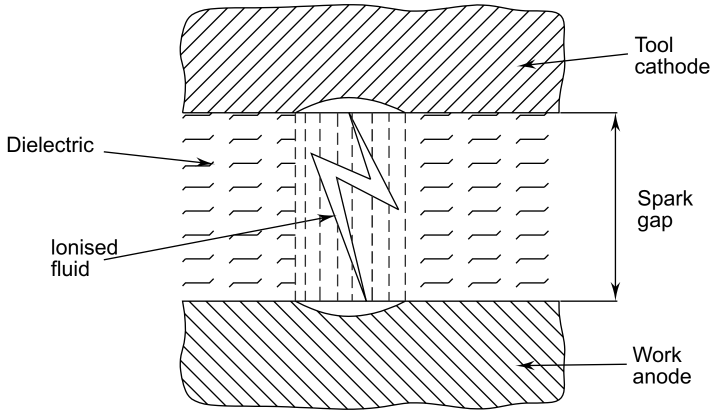


Fig. 11.2 *Schematic of the arc formation in EDM process*

Thus, the sequence of events in EDM can be summarized as

"1. With the application of voltage, an electric field builds up between the two electrodes at the position of least resistance. The ionization leads to the breakdown of the dielectric which results in the drop of the voltage and the beginning of flow of current.

2. Electrons and ions migrate to anode and cathode respectively at very high current density. A column of vapour begins to form and the localized melting of work commences. The discharge channel continues to expand along with a substantial increase of temperature and pressure.

3. When the power is switched off, the current drops; no further heat is generated, and the discharge column collapses. A portion of molten metal evaporates explosively and/or is ejected away from the electrode surface. With the sudden drop in temperature the remaining molten and vaporized metal solidifies. A tiny crater is thus generated at the surface.

4. The residual debris is flushed away along with products of decomposition of dielectric fluid. The application of voltage initiates the next pulse and the cycle of events."

Also due to the inertia of the surrounding fluid, the pressure within the spark becomes quite large and may possibly assist in 'blasting' the molten material from the surface leaving a fairly flat and shallow crater. The amount of metal removed per spark depends upon the electrical energy expended per spark and the period over which it is expended.

At any given time only one spark will be made between the tool and work piece at the shortest path as shown in Fig. 11.3. As a result of this spark some volume of metal is removed from both the tool and the work piece. Then the spark will move to the next closest distance as shown in Fig. 11.4. This process continues till the required material is removed from the work piece.

The temperature of the arc may reach about 10 000°C. The vapour of the metal would be quenched by the dielectric medium when the arc is terminated by the electric pulse and thus the wear debris is always spherical in nature. The wear debris would be carried away by the dielectric fluid, which is in continuous circulation. The same process as described above would be continued a number of times per second with each pulse removing a small wear particle from the work piece thereby causing the material to take the shape of the electrode. The arc will always be struck at a point between the work piece which is closest from the tool (electrode) thereby the complimentary tool surface will be reproduced in the work piece as shown in Fig. 11.5.

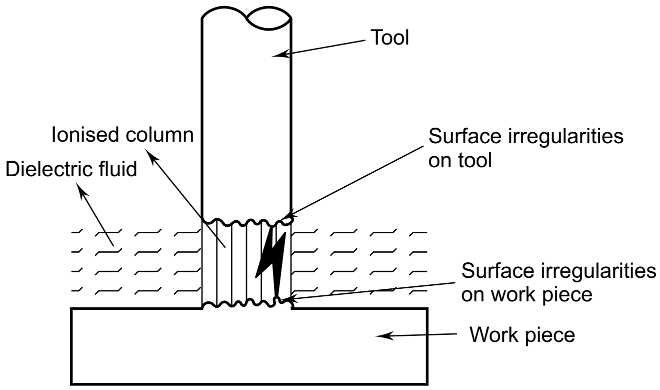


Fig. 11.3 *Schematic of the arc forming at the smallest distance between the tool and the work piece in the EDM process*

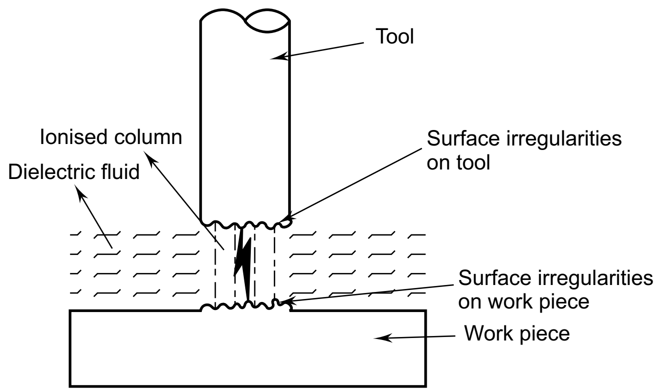


Fig. 11.4 *Schematic of the arc moving to the next smallest distance between the tool and the work piece in the EDM process*

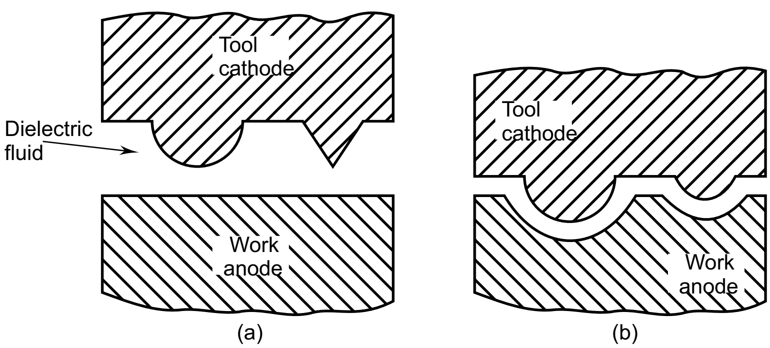


Fig. 11.5 *Typical surface generation in EDM process, (a) Initial shape of electrode and work piece, (b) Final complimentary shapes of electrode and work pieces after machining*

11.2.3 Schematic

A typical schematic of the various elements present in a commercial EDM machine is shown in Fig. 11.6. The main power unit consists of the required controlled pulse generator with the DC power to supply to the power pulses. The pulse frequency as well as the on and off time of the pulses can be very accurately controlled using electronic controllers.

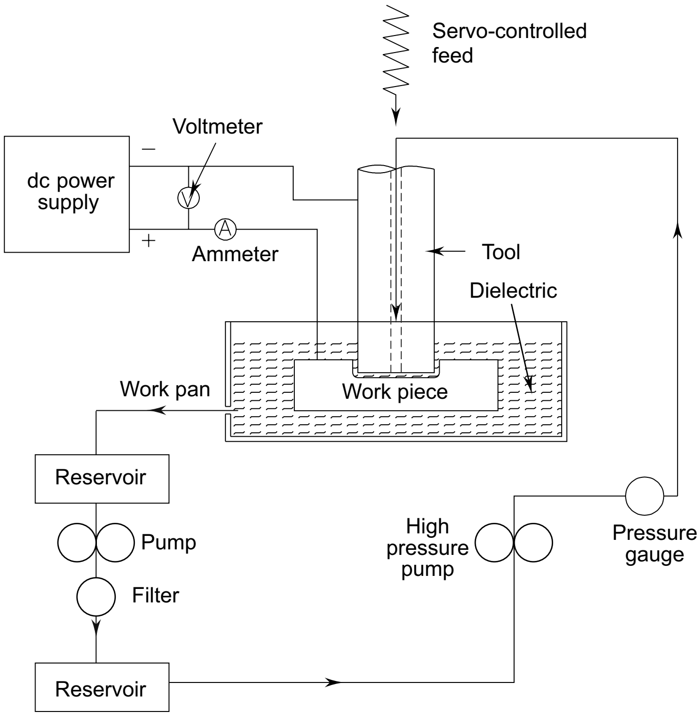


Fig. 11.6 *Typical schematic of the components that form a typical EDM machine*

The gap between the electrode and the work piece increases with the removal of metal. The EDM power supply voltage is dependent upon the spark gap, which needs to be maintained constant. A servo-controlled electrode feeding arrangement would be available which continuously senses the spark gap and moves the tool (electrode) to maintain this gap.

Typical parameters used in EDM process are:

Spark gap	0.0125 to 0.125 mm
Current	0.5 to 400 A
Voltage (DC)	40 to 300 V
Pulse duration	2 to 2000 µs
Dielectric pressure	< 0.2 MPa

The metal removal rates are about 16.4 cm^3/hour per 20 A of current. This can go as high as 250 cm^3/hour per 20 A of current.

Surface finish	3 to 10 µm Rough
	0.8 to 3 µm Finish

11.2.4 Dielectric Fluid

The dielectric fluid is a spark conductor, coolant and also a flushing medium. The requirements are:

1. The dielectric fluid should have sufficient and stable dielectric strength to serve as insulation between the tool and work till the breakdown voltage is reached.
2. It should de-ionise rapidly after the spark discharge has taken place.
3. It should have low viscosity and a good wetting capacity to provide effective cooling mechanism and remove the swarf particles from the machining gap.
4. It should flush out the particles produced during the spark out of the gap. This is the most important function of the dielectric fluid. Inadequate flushing can result in arcing decreasing the life of the electrode and increasing the machining time.
5. It should be chemically neutral so as not to attack the electrode, the work piece, the table or the tank.
6. Its flash point should be high so that there are no fire hazards.
7. It should not emit any toxic vapours or have unpleasant odours.
8. It should maintain these properties with temperature variation, contamination by working residuals and products of decomposition.
9. It should be economical and easily available.

A large number of fluids satisfy the requirement and are used as dielectric fluids. Most popular are the hydrocarbon fluids, silicone-based oils, and de-ionised water. Kerosene and water with Glycol are generally used. Typical values of material removal rates for various dielectric fluids are shown in Table 11.3.

Table 11.3 Performance of various dielectric fluids

Dielectric Fluid	Material Removal Rate, Work Material Removed $cm^3/A\ min \times 10^4$	Wear Ratio
Hydrocarbon oil	39.0	2.8
Distilled water	54.6	2.7
Tap water	57.7	4.1
Tetra ethylene glycol	102.9	6.8

$$\text{Wear ratio} = \frac{\text{Volume of work material removed}}{\text{Volume of electrode material worn out}}$$

A number of properties of dielectric fluids need to be considered while selecting them for the operation.

Flash point It the lowest temperature at which a dielectric gives off sufficient vapours to produce an inflammable mixture of air and gases in a standardized apparatus. The higher the flash point, the better it is for use in EDM. Generally any temperature higher than 80°C should be considered good for the operation.

Dielectric strength This is the ability of the fluid to maintain high resistivity before spark discharge and in turn the ability to recover rapidly with a minimal amount of OFF time. Fluid with a high dielectric strength offers finer degree of control. This will ensure better cutting efficiency.

Viscosity Fluids with lower viscosity are better for producing accurate and finer surfaces. Dielectrics of 2 to 3.5 cSt at a temperature of 20° C are suitable for polishing work while 4 to 6. 5 cSt at 20° C is suitable for rough cut operations. Thinner (lower viscous) fluids are easier to flush through smaller gaps between the tool and work in the case of fine machining. However for high MRR and high current values, heavier oils can be used.

Specific gravity Dielectrics normally used today have a specific gravity of 0.750-0.820. The shorter the chain of hydrocarbon molecules lower is its specific gravity. The heavier chips settle down in the lighter dielectric reducing the gap contamination and possibilities of secondary discharge and/or arcing.

Odour The unused dielectric should normally be odourless and should not begin to smell, even when heated.

Effect on health The effect of the dielectric on the health of the operator is of paramount importance. Of particular importance are its effects on skin irritation, toxicity, and smoke generation.

Typical properties of the dielectric fluids is given in Table 11.4.

Table 11.4 **Typical Properties of Dielectric Fluids from British Petroleum**

	180	200	200T
Gravity, °API	46.0	47.0	53.2
Density	0.797	0.793	0.766
Flash Point COC, °C (°F)	82	90	108
Viscosity, cSt 40° C	1.65	2.25	1.92
Aromatic Content, %	<0.01	<0.01	<0.001

Flushing refers to the method in which the dielectric fluid flows between the tool and the work gap. The efficiency of machining depends to a greater extent on the efficiency of the flushing. The wear debris present in the spark gap should be removed as quickly as possible. With poor flushing there is a possibility of build-up of the machined particles in the gap resulting in the short-circuiting and lower material removal rates. Problems with improper flushing are: uneven and significant tool wear affecting accuracy and surface finish; reduced removal rates due to unstable machining conditions and arcing around regions with high concentration of debris. It is noted during an experimental study that there is an optimum dielectric flushing rate of about 13 ml/s while machining AISI O1 tool steel, where the crack density and average thickness of the recast layer are at a minimum.

The flushing method that can be used in EDM depends upon the work piece geometry and can be classified as:

- Normal flow
- Reverse flow
- Jet flushing
- Immersion flushing

Normal flow In this method, fluid is introduced, under pressure, through one or more passages in the tool and then it is forced to flow through the gap between the tool and work piece as shown in Fig. 11.7. The location of the flushing holes should generally be in areas where the deepest material is to be removed. It is generally good to have a single hole rather than a number of holes. This will help in improving the flow of the dielectric fluid. However the choice will also depend upon the work piece geometry and sometimes it may become necessary to have a number of holes to facilitate the flow of the dielectric fluid to the different areas in the work piece that need to be cut.

Normal flow flushing gets facilitated if there is a hole present in the work piece or a hole can be made without harming the work piece geometry. In this case there will be no need to make a hole in the tool as shown in Fig. 11.8. In this method electrolyte is pumped into the manifold under pressure, which will then flows through the hole in the work piece. This arrangement is normally used in situations where it becomes difficult to drill a hole in the tool because of the tool's length or because the cross-section area is too small.

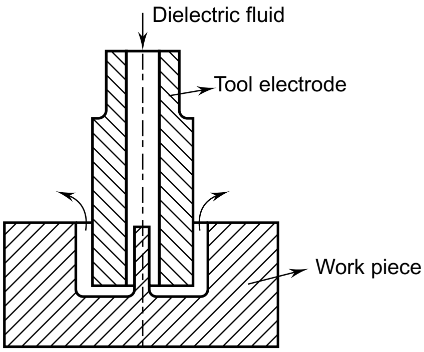


Fig. 11.7 *Normal flow flushing techniques used with the dielectric flow through the tool in the EDM process*

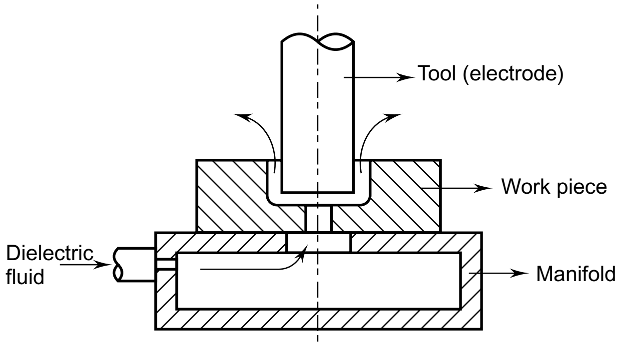


Fig. 11.8 *Normal flow flushing techniques used with the dielectric flow through the work piece in the EDM process*

Under certain circumstances there is a possibility of tapered surfaces being produced by normal flow as shown in Fig. 11.9. Chips produced by the tool at the end face such as those shown in A and B need to pass through the gap as shown in figure 11.9. As they move, their conductivity will reduce the gap between the tool and work piece thereby prompting for additional side arcs, which will remove material from the work piece. This will cause the vertical walls to become tapered as shown. As the mouth widens as shown at C, more chips need to be aligned in order produce a side arc, which is less probable. As a result this part of the wall will remain vertical without any taper.

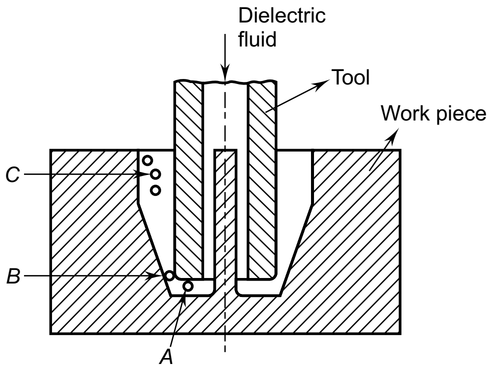


Fig. 11.9 *Production of tapered surfaces during normal flow flushing technique in the EDM process*

Reverse flow To reduce the taper produced the reverse flow can be used. In this case the gap between the tool and work piece is completely submerged in the dielectric fluid and then a vacuum is applied to the manifold as shown in Fig. 11.10. Since the chips will be flowing through the work piece there is no possibility of side arc. However the flow rate that can be achieved is limited depending upon the vacuum applied. Typical pressure differentials that could be achieved are about 65 to 90 kPa. This is particularly suitable for deep-cavity dies.

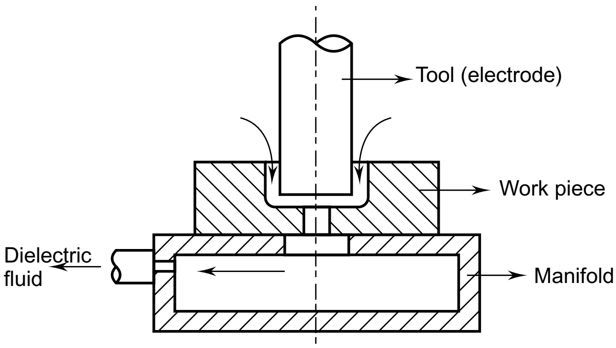


Fig. 11.10 *Reverse flow flushing techniques used in the EDM process*

Instead of vacuum, a positive pressure can be applied to achieve reverse flow as shown in Fig. 11.11. Dielectric fluid is introduced into the gap through an expendable cover placed on top of the work piece as shown in Fig. 11.11. Tool will first machine the cover with a clearance at C. As the tool advances to machine the work piece some fluid may be lost through the gap, but it will be possible to provide higher pressures and consequently higher dielectric flow rates.

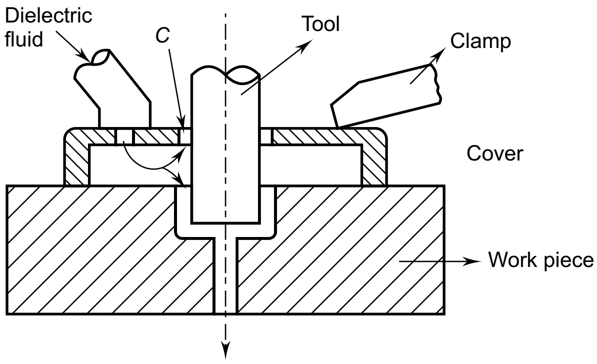


Fig. 11.11 *Reverse flow flushing techniques used in the EDM process*

Jet flushing A simpler flushing technique is the use of direct spray or jet of the fluid at the machining zone as shown in Fig. 11.12. This is sometimes used for machining of a long narrow slot or cavity in a work piece. However machining times are longer with this technique. For jet flushing of an array of shallow cavities, important considerations are distribution of the nozzles, flow rates, angles at which the nozzles are directed at the gap, and layout of the cavities.

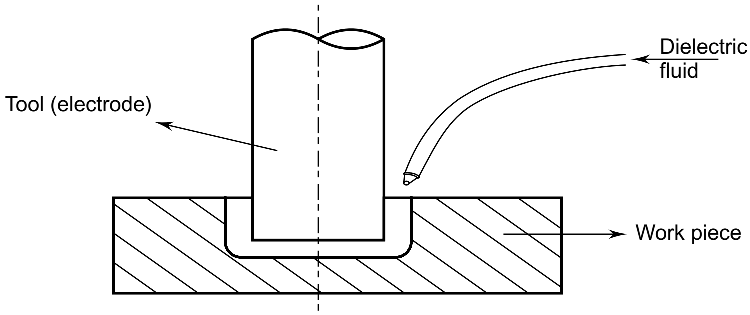


Fig. 11.12 *Jet flushing technique used in the EDM process*

Immersion flushing It is also possible to allow the cutting to takes place without any pumping of the fluid and by simple immersion. A little agitation may be provided by the relative movement of the tool and adding a vibration. This is called immersion flushing.

A closed-form mathematical expression for the fluid pressure and fluid velocity under a "no-sparking" condition at all points in the electrode-to-work piece gap of a typical EDM set up was derived by Earnest Y. Seborg. The volume flow rate of the fluid and the force developed on the tool electrode by the fluid flow are derived directly from the expressions for velocity and pressure. The EDM setup is a round electrode with a

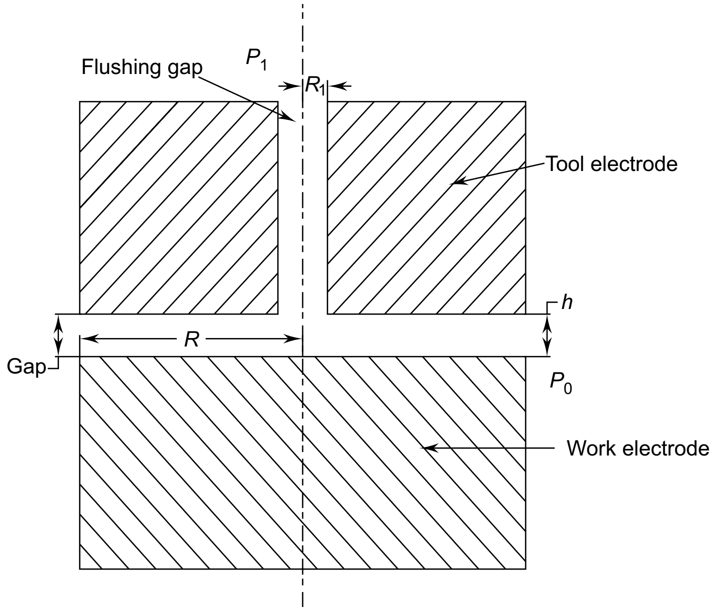


Fig. 11.13 *Fluid flow passage through the electrode in the EDM process*

central hole for electrolyte flow as shown in Fig. 11.13. The following assumptions were made in deriving the equations:

- Electrode and work piece surfaces with which the electrolyte is coming into contact are perfectly smooth
- The electrolyte flow is laminar (Reynolds number < 2000)
- Inside radius of the flushing line (R_1) is much larger than the flushing gap.

Pressure distribution in the flushing gap, $P(r) = P_0 + \dfrac{(P_1 - P_0)\ln(R_0/r)}{\ln\left(\dfrac{R_0}{R_1}\right)}$

Flow rate of the electrolyte, $Q = \dfrac{\pi}{6}\dfrac{(P_1 - P_0)h^3}{\mu \ln\left(\dfrac{R_0}{R_1}\right)}$

Net force acting on the electrode, $F_{net} = \dfrac{\pi(P_1 - P_0)(R_0^2 - R_1^2)}{2\ln\left(\dfrac{R_0}{R_1}\right)}$

Where

R_0 = radius of the electrodes
R_1 = radius of flushing hole
h = gap spacing
P_0 = atmospheric pressure
P_1 = pressure in flushing hole

11.2.5 Electrodes

As explained earlier, in the EDM process the shape of the electrode is impressed on the work piece in its complimentary form. The shape and accuracy of the electrode plays a very important role in the final accuracy of the work piece machined.

The electrode material should have the following characteristics to serve as a good tool.

1. It should be a good conductor of electricity and heat.
2. It should be easily machinable to any shape at a reasonable cost.
3. It should produce efficient material removal rates from the work pieces.
4. It should resist the deformation during the erosion process.
5. It should exhibit low electrode (tool) wear rates.
6. It should be available in a variety of shapes.

Various electrode materials used: Graphite, Copper, Copper graphite, Brass, Zinc alloys, Steel, Copper tungsten, Silver tungsten, Tungsten, etc. Some of the physical properties of the common electrode materials are given in Table 11.5. The detailed considerations for the selection of the individual electrode materials are given below:

Table 11.5 Physical properties of some EDM electrode materials

Property	Copper	Graphite	Tungsten	Iron
Melting point, °C	1 083	—	3 395	1 535
Boiling point, °C	2 580	> 4 000	5 930	2 800
Heat to vaporise 1 cm^3 from room temperature, cal/cm^3	12 740	20 000	22 680	16 900
Thermal conductivity, Ag = 100	94.3	30.0	29.6	16.2
Electrical conductivity, Ag = 100	96.5	0.1	48.1	16.2
Thermal expansion, per °C $\times 10^{-6}$	16.0	4.5	4.6	15.0
Strength, MPa	241	34	4137	276
Modulus of elasticity, MPa $\times 10^3$	124	5.9	352	186

Copper

Pure copper or electrolyte grade copper is extensively used as an electrode material. It is most often used when fine finishes are required in the work piece. It is also excellent for no wear EDM. It exhibits a very small wear ratio. Machining of copper is a major problem because of its poor machinability.

Tellurium Copper

The main problem with copper is during the grinding for fine finish, wheel loading takes place. In such cases the tellurium-copper electrodes may replace copper electrodes. Tellurium-copper has machinability rating compared to 100% for free machining brass. EDM characteristics of tellurium-copper are same as that for copper. The only disadvantage of tellurium-copper is its scarcity.

Brass

Free machining brass is often used as an electrode material. The main advantage is that it is easily available and can be readily machined. The wear ratio is more of the order of 1 to 6 for small electrodes. It is often used for tubular electrodes in specialised small hole EDM drilling machines where high wear is acceptable. Brass is a good electrode material for some alloys of titanium under poor chip removal conditions. However it is not used on tungsten carbide because of high wear. Its main advantage is that it is available in small tubing and shim stock.

Graphite

Graphite is perhaps the most widely used EDM electrode material because of good machinability and wear characteristics. Graphite permits fast material removal rates as well as best wear ratio. It offers excellent stability.

Grain size is the most important property of graphite electrodes. Coarse particle graphite is normally used for large volume EDM work, where there is little or no fine detail to be produced. Alternatively the fine particle high-density graphite is used where fine details and accuracy are required, since its strength makes it capable of producing thin cross-sections. Since graphite is a sintered material, it contains porosity. This must be taken into account while machining three-dimensional cavities, which require fine surface finish. Coarse particles have more porosity and produce rougher finishes. Medium grade graphite usually works well when machining through holes in steels.

Care should be exercised when machining tungsten carbide. During the machining of tungsten carbide by graphite electrodes, if the chip removal conditions are not good, cutting surface will become carbonised and uncontrolled arcing may result. This uncontrolled arcing is also known as DC arcing and sometimes is quite dangerous.

The DC arcing is caused by the carbon deposited on the work surface being heated to the point that de-ionisation of the dielectric fluid does not takes place. Without de-ionisation, current flows across the same point between electrode and work piece causing excessive heating.

While machining tungsten carbide with graphite, it is generally recommended that only fine particles and high density graphite be used. Its main advantages are:

1. It is comparatively inexpensive.
2. It has wear ratio equivalent to that of copper-tungsten alloys.
3. Large electrodes can be made by adhesive bonding of laminations.
4. It produces high material removal rates.

Copper - Graphite

Copper-graphite is graphite that has been infiltrated with copper. Its flexural rigidity is higher than the comparable grade of graphite and has the best characteristics of both copper and graphite. However, it is 1.5 to 2 times more expensive than graphite. It has more conductivity than graphite but corner wear is not as good as that of the pure graphite. It works well under poor flushing conditions. It works well for machining tungsten carbides.

Steel

Steel is not a satisfactory electrode material. However, its use is limited to match the parting planes of the moulds in which half of the mould is used as the electrode and the other half is used as the work piece. Material removal rate is very slow and the wear ratio is suitable only for certain combination of steels.

Copper - Tungsten

Copper-tungsten is also a sintered electrode where copper infiltrates the tungsten powder compact. The machinability is fair and has good wear and finish characteristics. It is used for close tolerances, fine detail and low wear. It has good strength and is less prone to breakage or fracture when machined into thin sections and fine details. It has high density, strength and good thermal and electrical conductivity.

Tungsten

Tungsten has high rigidity as well as good ratio. It is used for making small holes that are less than 0.2 mm for which electrodes with small flush holes are not available. The disadvantage is its high cost.

Zinc Alloys

The main application of zinc alloys is for high production where a large number of identical electrodes are required which can be mass produces by pressure die casting or coined. Complex shapes of electrodes can be easily die cast compared to machining used with other electrodes. Cost of the electrodes is less while high currents up to 150 amperes can be used with them. The main disadvantage is the rapid corner wear and very poor wear ratio.

Comparative capabilities of the individual electrode materials is given in Table 11.6.

Table 11.6 Electrode-material selection

Electrode Material	Form	Corner Wear Ratio in Finishing	End-wear Ratio in Roughing	Relative Cost	Machin-ability Rating	Uses	
						Recommended	Not Recommended
Graphite	Blocks, rod, tube, bar	5:1	to 100:1	Low	Excellent	Tooling	—
Copper	Bar, rod, sheet, wire, tube, frogings, stampings	1:1	2:1	Medium	Good	Holes, slots	High accuracy and detail
Copper–Graphite	Blocks, rods	2:1	4:1	Medium	Fine	General purpose	—
Brass	Same as copper	0.7:1	1:1	Low	Good	Holes and cavity sinking	High accuracy
Zinc alloys	Cast, die casting	0.7:1	2:1	Low	Good	Forging die cavities	Holes
Steel	All forms	1:1	2:1	Low	Excellent	Through holes	Carbides
Copper–Tungsten	Bar, flats, shim stocks, rod, wire tube	3:1	8:1	Medium	Fair	Slots, carbides	Large areas
Silver–Tungsten	Sintered	8:1	12:1	High	Fair	Small slots, holes and intricate details	Large areas
Tungsten	Wire, rod, ribbon	5:1	10:1	High	Poor	Small holes	Irregular holes

11.2.6 System Analysis

The exact amount of material removed per spark cannot be calculated. The major work piece parameter is the energy required to raise a unit volume to its melting point. The power supplies are 50 to 80% efficient and a typical value is of the order of 2.3 mm^3/s/kW. However, steel can be raised to its melting point using about 120 mm^3/s/kW. Thus the heat required for melting accounts only for about 2% of the energy input.

A typical circuit used for supplying the power to an EDM machine is shown in Fig. 11.14 named as the relaxation circuit. The circuit consists of a DC power source, which charges the capacitor 'C' across a resistance 'R_c'. Initially when the capacitor is in the uncharged condition, when the power supply is on with a voltage of V_o, a heavy current, i_c, will flow in the circuit as shown to charge the capacitor. The voltage at any time, t, is given by

$$V_c = V_o\left(1 - e^{-\frac{t}{R_c C}}\right)$$

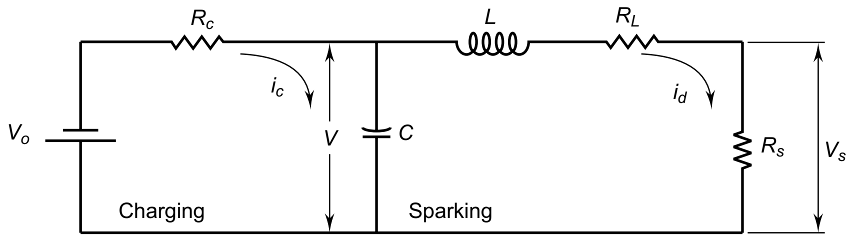


Fig. 11.14 *Relaxation circuit used for generating the pulses in EDM process*

The time constant, τ of the circuit is given by

$$\tau = R_c \times C$$

Charging current can then be specified by

$$i_c = \frac{V_o}{R_c} e^{-\frac{t}{\tau}}$$

The energy delivered per spark, P is given by

$$P = \frac{V_o^2 \tau}{R_c}\left[\frac{1}{2} - e^{-\frac{t}{\tau}} + \frac{1}{2} e^{-\frac{2t}{\tau}}\right]$$

If τ_c = time for charging the capacitor up to the breakdown voltage, then the average power, P_{av} is given by

$$P_{\text{av}} = \frac{V_o^2 \tau}{R_c \tau_c}\left[\frac{1}{2} - e^{-\frac{\tau_c}{\tau}} + \frac{1}{2} e^{-\frac{2\tau_c}{\tau}}\right]$$

For maximum power

$$\frac{dP_{\text{av}}}{d\left(\dfrac{\tau}{\tau_c}\right)} = 0$$

which gives rise to

$$V_c = 0.72 \times V_o$$

Discharge circuit:

Neglecting the inductance.

$$V_s = V_c\, e^{-\frac{t}{R_{LS} C}}$$

Where $R_{LS} = R_L + R_S$

The discharge current, i_d is given by

$$i_d = \frac{V_s}{R_{LS}}$$

Since the R_L and R_S are so arranged that the frequency of discharge oscillation, f_d is not affected. The frequency of discharge is given by

$$f_d = \frac{1}{2\pi}\sqrt{\frac{1}{LC}}$$

This means that spark reversal will take place. However, the energy available for the reverse spark is so small because of the resistance that there is no reverse spark.

For satisfactory metal removal the spark must be completely quenched between the charging cycles. This will allow the spark gap to be de-ionised. This will be the limiting condition for the maximum frequency of sparks to be provided for material removal.

An approximate criterion for arcing is that the charging circuit maintains the maximum spark current, i_{dmax} indefinitely, i.e.,

$$V_o - i_{\text{d max}} R = \overline{V_C}$$

Where $\overline{V_C}$ is the spark discharge voltage at maximum spark current.

It can be shown that

$$i_{\text{d max}} = \frac{V_c}{\sqrt{\dfrac{L}{C}}}$$

Hence,

$$V_o - \frac{V_c R}{\sqrt{\dfrac{L}{C}}} = \overline{V_c}$$

Assuming $V_o - \overline{V_c} \approx V_c$

The value of the minimum resistance in the circuit, $R_{\min}$ is given by

$$R_{\min} \approx \sqrt{\frac{L}{C}}$$

However, experimental evidence shows that

$$R_{\min} \approx 30\sqrt{\frac{L}{C}}$$

Charging time, τ_c is given by

$$\tau_c = R_{\min}\, C = 30\sqrt{LC}$$

Sparking time, τ_s is given by

$$\tau_s = \frac{0.5}{f_d} = \pi\sqrt{LC}$$

$$\text{Frequency of cutting} = \frac{1}{\tau_s + \tau_c} = \frac{0.03}{\sqrt{LC}}$$

The material removal rate, MRR is proportional to the product of frequency of charging and the energy delivered per spark.

Energy delivered per spark = $\frac{1}{2}CV_c^2$

Thus,

$$MRR \propto \frac{1}{2}\, f_c\, V_c^2\, C$$

Since
$$V_c = V_o\left[1 - e^{-\left(\frac{t}{R_c C}\right)}\right]$$

or,
$$t = R_c\, C \log_e \frac{1}{\left(1 - \frac{V_c}{V_o}\right)}$$

Frequency of charging, f_c is given by

$$f_c = \frac{1}{t}$$

Hence, the material removal rate, MRR is given by

$$MRR = \frac{K_M V_c^2}{2 R_c} \frac{1}{\log_e \frac{1}{\left(1 - \frac{V_c}{V_o}\right)}}$$

Where K_M is the proportionality constant.

The Fig. 11.15 shows the charging curve for the capacitor. The capacitor voltage will increase until it reaches the breakdown voltage at which point the discharge takes place. The same cycle will continue.

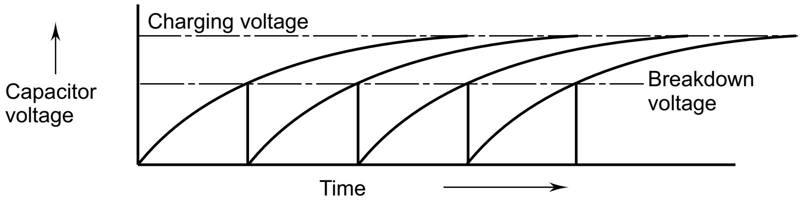


Fig. 11.15 *Variation of capacitor voltage with time in a relaxation circuit*

11.2.7 Pulse Generation

The relaxation circuit as explained above was used in the early EDM machines. They are limited to the low material removal rates for fine finish, which limits its application. This can be explained from the fact that the time spent on charging the capacitor is quite large during which time, no machining can actually take place. Thus the material removal rates are low. The peak current experienced by the electrodes is high for very short duration that raises the temperature and consequently the tool wear. Hence the ideal would be to increase the spark duration and reduce the peak current (consequently the maximum temperature in the spark) thereby improving the electrode life.

The power supply circuits used for this purpose are with pulse generators that provide the necessary spark energy at the required intervals without the use of any capacitors. The spark will have all the required energy and is made available as soon as the gap conditions are right for sparking. In these systems the spark current flows from a source whose voltage is not less than the limiting voltage of the spark gap. Typical variation of voltage and current in a pulse generator type power supply is given schematically in Fig. 11.16. As can be noted the pulse on time (t_c) is increased considerably while the pulse off time (t_i) is reduced which would result in the machine removing material most of the time. Further the peak current is only little above the average current thereby reducing the peak temperatures, reducing the tool wear.

Typical pulse generator power supply circuits are shown in Fig. 11.17. As can be noted in Fig. 11.17(a) the switching is driven by an oscillator that can be set at any frequency required. Gap conditions controls the oscillator such that the transistors can be turned off in case of a d-c short circuit at the gap. Solid-state power supplies have greater reliability, precise control of parameters, and compactness. Many circuits are available for controlling current as well as voltage with any wave shape as a single or repetitive pulse.

With developments in microprocessors, EDM power supplies are benefited as shown in Fig. 11.17(b). All the spark parameters, as well as machine tool functions, can now be controlled by a microcomputer thereby allowing for optimum performance.

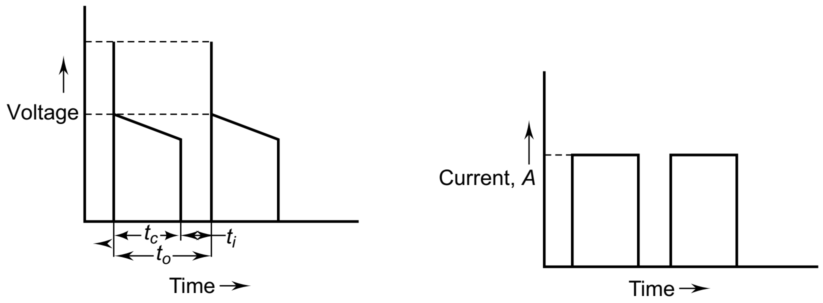


Fig. 11.16 *Variation of pulse voltage and current with time*

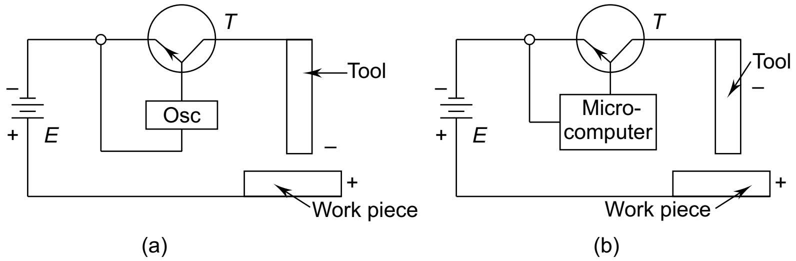


Fig. 11.17 *Pulse generator type power supplies for EDM*

11.2.8 Process Characteristics

The metal removal rates in EDM depend upon the following parameters, which are discussed in greater details below:

- Current in each spark
- Frequency of the discharge
- Electrode material
- Work piece material
- Dielectric flushing condition

The amount of material removed and the surface finish produced is dependent upon the current in the spark. The material removed by the spark can be assumed to be a crater as shown in Fig. 11.18. The amount removed therefore will depend upon the crater depth, which is directly proportional to the current. Thus as shown schematically in Fig. 11.18, the material removed increases and at the same time the surface finish also decreases.

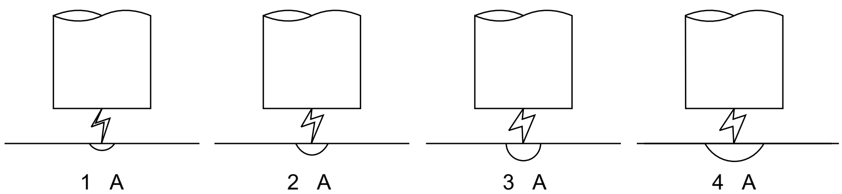


Fig. 11.18 *Schematic representation of the effect of the current in each spark, which determines the material removed*

However, decreasing the current in the spark, but increasing its frequency will improve the surface finish in view of the small crater size, but at the same time the material removal rate can be maintained by increasing the frequency. The same is shown schematically in Fig. 11.19.

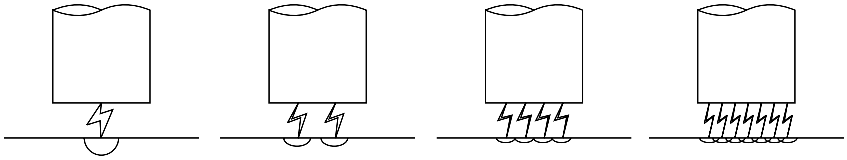
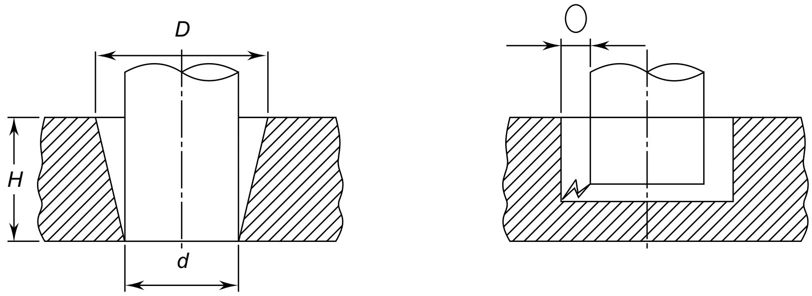


Fig. 11.19 *Schematic representation of the effect of spark frequency on the material removed and the surface finish*

The side sparks affects the accuracy of the surface produced. Two possible inaccuracies are shown in Fig. 11.20.



(a) Taper (b) Overcut

Fig. 11.20 *The possible errors in the EDM surfaces*

Taper The side sparks between the tool and the machined surface produces taper. As a result, the taper increases as the depth of the machined surface increases. It is empirically noted that the taper is proportional to the square of the diameter as given below:

Taper is given by

$$\frac{D - d}{2H} = K_T\, d^2$$

Where K_T is an empirical constant.

Over cut Similarly the over cut produced by the side sparks is shown in Fig. 11.20(b). Lazarenko has obtained the following relationship for the over cut in the case of EDM using a relaxation circuit.

$$\text{Overcut, } O = A\, C^{0.333} + B$$

Where A and B are constants given in Table 11.7.

The dependence of over cut on the capacitance of the power supply is shown in Fig. 11.21.

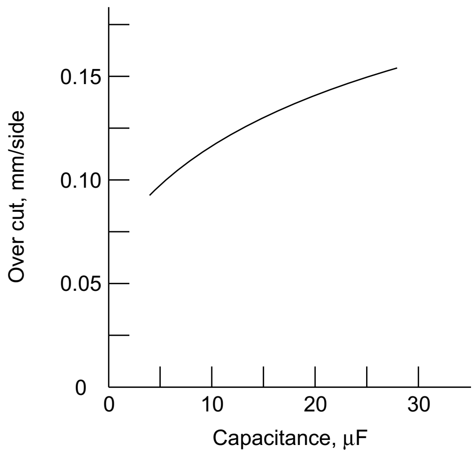


Fig. 11.21 *Variation of over cut with the capacitance in a relaxation type EDM machine; Tool: Copper, Work: Mild steel, Voltage: 150 V*

Table 11.7 Constants for the over cut equation

Work	Tool	*A*		*B*	
		V_c		V_c	
		100 V	150 V	100 V	150 V
C30	Copper	0.035	0.0426	0.015	0.025
T15K60	Copper	0.030	0.044	0.015	0.020
Copper	Copper	0.032	0.045	0.015	0.210

Metallurgical modification The surface of the work piece melts and gets quickly solidified by the cooling action of the dielectric fluid. The layer of the re-solidified metal is very small with low powers, but will become thicker with the EDM higher powers used. Next to the re-solidified material is the heat-affected material, which is usually less than 0.25 mm in thickness. It is necessary to remove the heat-affected layer for higher fatigue strength of the component.

11.2.9 Applications

The EDM process is extensively used because of its many advantages. A few of them are discussed below:

There is no physical contact between the tool and the work piece and hence no cutting forces acting on the work piece. Even fragile work pieces can be machined using this process.

- Any complex shape required in dies and mould can be easily produced to the required degree of accuracy and finish. Since the process copies the tool shape in a complimentary form, making a male electrode is easier than the complimentary female form required.
- The process is not affected by the hardness of the work material. Hence even the hardened material can be machined thus avoiding the possible distortions due to heat treatment on the final geometry.
- The material removal rates are almost comparable with that of the conventional machining processes.
- Since there are no cutting forces acting on the tool, high aspect ratio surfaces can be machined using EDM process.
- Though the material is removed by the heat produced by the spark, there is thermal damage to the work piece material.
- The process is generally highly automated with very little operator skill required.
- The actual surface produced by EDM consists of small craters, which may help in the retention of the lubricants.

Disadvantages

- The wear rate on the electrode is considerably higher. Sometimes it may be necessary to use more than one electrode to finish the job.
- The work piece should be electrically conductive to be machined using the EDM process.
- The energy required for the operation is more than that of the conventional process and hence will be more expensive.

The process generally described so far is termed as EDM die sinking. The other processes that are possible are described below:

EDM Drilling Drilling of very small and high aspect ratio of the order of 30 is one common example of EDM application. A typical example to demonstrate the capability of the process is the drilling of a hole of 0.3 mm diameter through a 20 mm hardened ball bearing. The aspect ratio is 67 and the process is completed in 90 seconds.

EDM Milling Milling EDM (Fig. 11.22) can be used to machine complex shapes with simple cylindrical electrodes. This is very similar to the conventional milling process with the cylindrical electrode acting like a milling cutter to produce complex surface by eroding the surface. It is possible to use simple electrodes and produce complex jobs using the Computer Numerical Control technology to provide the necessary complex motions.

Fig. 11.22 *Typical example of EDM milling producing a complex work piece with simple cylindrical electrode*

11.2.10 Wire EDM

Electric discharge wire cutting or more commonly called wire EDM is a process of producing complex 2 and 3 dimensional shapes using a simple wire eroding the material from an electrically conducting material. A typical schematic of a wire EDM operation is shown in Fig. 11.23.

The electrode wire is typically of a diameter of 0.05 to 0.25 mm copper or brass, which is wound between the two spools as shown in Fig. 11.23. The wire moves past the work piece at fast rates up to 3 m/min. The spark is struck between the moving electrode wire and the work piece thereby removing the material. The dielectric most commonly used is the de-ionised water applied as a localised stream rather than submerging the whole work piece.

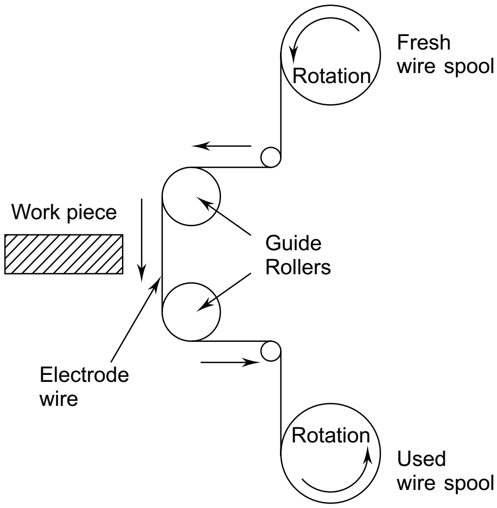


Fig. 11.23 *Principle of wire EDM process*

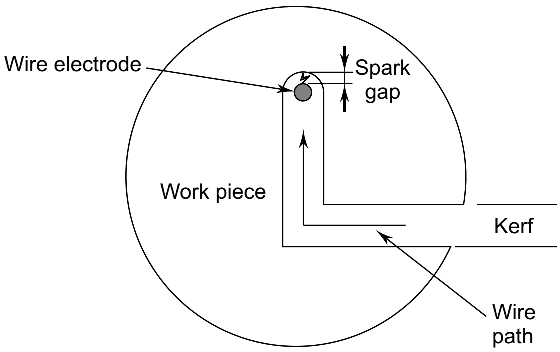


Fig. 11.24 *Close-up of wire EDM cutting process*

The close-up view of the cutting process is shown in Fig. 11.24. Kerf is the width of the cut produced by the wire.

Wire EDM is used for machining the sheet metal dies, extrusion dies and prototype parts. It is relatively a very slow process (linear travels of the order of 100 mm/hour 25 mm thick steel) utilising computer controlled machines.

11.3 ELECTRO CHEMICAL MACHINING

Electro chemical machining (ECM) is another process using electrical current to remove the metal, but unlike EDM relies on the principle of electrolysis for material removal. Michael Faraday (1791–1867) discovered that if two electrodes are placed in a bath containing a conducting liquid and a DC potential is applied across them, then metal can be depleted from the anode and plated on the cathode. This process is universally used in electroplating by making the work piece as cathode. However, in ECM material is to be removed and hence electroplating is reversed, i.e., making the work piece as anode.

11.3.1 Principle of ECM

ECM is based on electrolysis. A typical arrangement of an electrolytic cell is shown in Fig. 11.25. When a potential difference is applied across the cathode and anode, there are a number of possible reactions that can takes place. The following are some of those that are relevant for ECM.

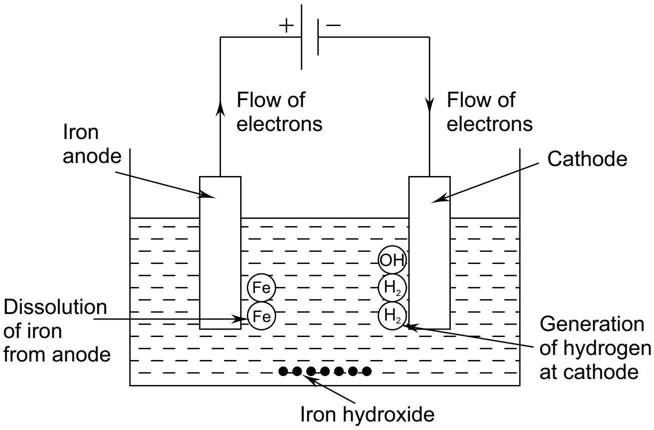


Fig. 11.25 *Principle of electrolysis*

The reaction taking place at the anode is the dissolution of anode by the electrolyte.

$$Fe \rightarrow Fe^{++} + 2e$$

Similarly at the cathode, hydrogen gas is released from the water contained in the electrolyte.

$$H_2O + 2e \rightarrow H_2 + 2OH^-$$

Combining the above two reactions, the iron and hydroxyl ions would combine to form the iron hydroxide as follows:

$$Fe^{++} + 2OH^- \rightarrow Fe\,(OH)_2$$

The net reaction of all the above three reactions can be shown as

$$Fe + 2H_2O \rightarrow Fe\,(OH)_2 + H_2$$

It is further possible that the iron (ferrous) hydroxide may further react with water and oxygen forming the ferric hydroxide as shown below:

$$4Fe\,(OH)_2 + 2H_2O + O_2 \rightarrow 4Fe\,(OH)_3 + H_2$$

What is interesting to note at this stage is that the net result of all this is that iron gets dissolved from the anode and forms the residue consuming the electricity and water and nothing else. The reaction products are ferric hydroxide and hydrogen gas. Based on this it is possible to make the following observations:

- The metal from the anode is dissolved electrochemically and hence the metal removal rate based on the Faraday's laws will depend upon atomic weight, valency, the current passed and the time for which the current is passed, and on no other parameter.
- At the cathode only hydrogen gas is evolved and no other reaction takes place, so the shape of the cathode is unaffected.

Based on the above observations, the ECM process can now be conceived as a process involving a tool cathode which has the complimentary shape of the part to be produced and the work to be done as anode, as shown in Fig. 11.26. In the small gap between the work piece and the tool a suitable electrolyte is pumped at high pressure. Note that in the final position the shape of the tool remains the same as when it started, which is one of the major advantages of the ECM process.

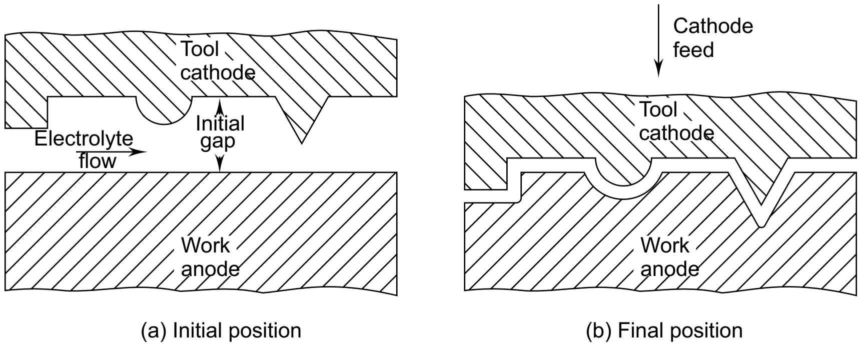


Fig. 11.26 *Complimentary shape produced by ECM*

11.3.2 ECM Equipment

A schematic view of the various elements present in an ECM machine is shown in Fig. 11.27. Generally the commercial ECM machines are characterised by the large sizes in view of the high power supplies involved. Typical power supplies used may range from 500 A to 40 000 A.

The tool to work the gap needs to be maintained at a very small value of the order of 0.25 mm for satisfactory metal removal rates. The electrolyte needs to be pumped through this gap at high pressures ranging from 0.70 to 3.00 MPa. This introduces a large amount of load on the machine, because of the large working areas involved. For example, if the working area is 800 cm^2, with an electrolyte pressure of 1.0 MPa, the resulting load on the tool will be 80 000 N. Hence the machine structure will have to be made rigid to withstand such forces.

The electrolyte consists of the metal debris removed from the anode, which will have to be filtered before it is re-pumped into the system. Also a large amount of heat is generated during the electrolysis, which heats up the electrolyte, and hence it needs to be cooled. The electrical conductivity of the electrolyte changes with temperature. A constant equilibrium gap needs to be maintained during the ECM operation. A servo drive is provided on the tool axis for this purpose.

The electrolytes used in the ECM process are corrosive in nature and hence proper care needs to be taken to see that the all the materials that come in contact with the electrolyte be made of stainless steel, plastic or other materials to withstand the corrosion. Similarly provision needs to be made to safely exhaust the hydrogen gas generated during the process with explosion-proof blowers continuously.

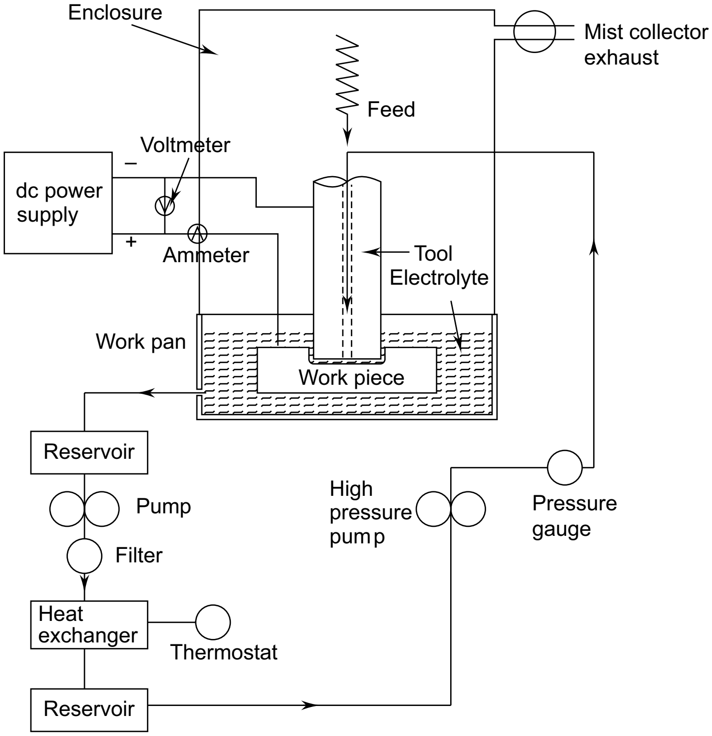


Fig. 11.27 *Schematic of the various elements present in a commercial ECM machine*

11.3.3 Electrolyte

Electrolytes used in ECM should be carefully selected such that they provide the necessary reactions without plating the cathode. The typical functions expected to be served by an electrolyte in ECM are:

- Completes the electrical circuit between the tool and the work piece,
- Allow desirable machining reactions to take place,
- Carry away heat generated during the operation,
- Carry away products of reaction from the zone of machining.

The properties that should be carefully looked into during the selection of the electrolyte to serve the function are:

- High electrical conductivity
- Low viscosity and high specific heat
- Chemical stability
- Resistance to formation of passivating film on work piece surface
- Non corrosive and non-toxic
- Inexpensive and readily available

The salt solutions with water forming a large proportion satisfy many of the above conditions and therefore are generally used. Some general electrolytes used are:

- Sodium chloride or potassium chloride up to 0.25 kg/litre. Most widely used because of low cost and stable conductivity over a broad range of pH values. However it is corrosive and produces large amount of sludge. It cannot be used on tungsten carbide or molybdenum.

- Sodium nitrate up to 0.50 kg/litre. Less corrosive but forms a passive film on the work piece surface. Hence not used as a general purpose electrolyte. It is used for machining aluminium and copper.

The electrolyte, when drilling a hole, is allowed to flow through the tool under high pressure and exit through the work piece. Since the electrolysis takes place as long as the current is flowing through the tool, there is a steep taper produced by an uninsulated tool as shown in Fig. 11.28. To reduce this taper, it is necessary to insulate the sides of the tool as shown in Fig. 11.28(b), which will produce straight walls.

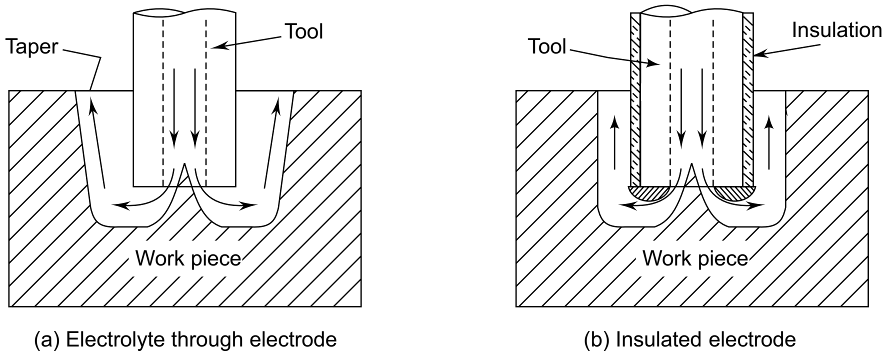


Fig. 11.28 *Electrolyte flow methods in ECM*

Insulated electrode as shown in Fig. 11.18(b) produces straight walls, but the over cut of the hole produced gets reduced in the process since material is only being removed from the bottom surface. Also the sharp corners experienced by the electrolyte flow causes turbulence which is not good for the process. Hence streamlining can be done by providing an uninsulated tool tip at the bottom as shown in Fig. 11.29. The tool tip provides the dual purpose of increasing the over cut slightly and at the same time promoting the smooth flow of the electrolyte thereby reducing the pressure drops across the machining gap.

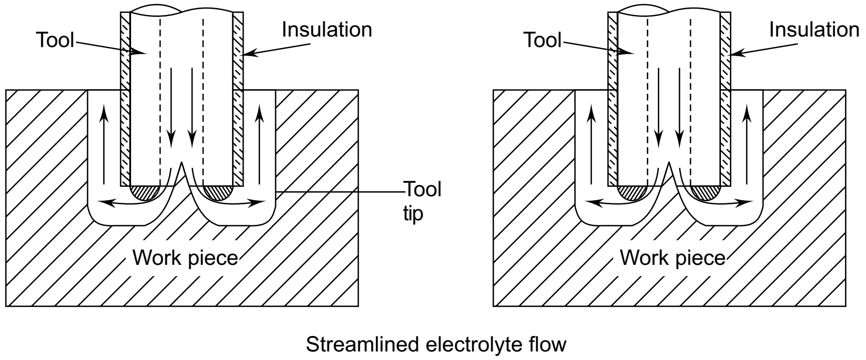


Fig. 11.29 *Electrolyte flow methods in ECM*

Sometimes the reverse flow of electrolyte through the tool as shown in Fig. 11.30 would be useful since it decreases the metal removed, by leaving a large slug at the centre of the hole produced. Also this is the best arrangement for the electrolyte flow since the finished surface is not affected by the electrolyte containing the metal debris.

11.3.4 ECM System Analysis

The first law of Faraday's electrolysis states that the chemical change produced during electrolysis is proportional to the current passed and the electrochemical equivalence of the anode material.

Hence, the material removal rate (MRR) in ECM (assuming 100% current efficiency) is given by

$$MRR = \frac{AI}{ZF}\ \text{kg/s}$$

The volumetric material removal rate is given by dividing the above with the density of the work piece material,

$$MRR = \frac{AI}{ZF\rho_a}\ \text{mm}^3\text{/s}$$

Where A = Atomic weight of the work material,
I = Current, Amperes
Z = Valency of the work material,
F = Faraday's constant = 96 540 Coulombs
ρ_a = Density of work material

The gap resistance, R is given by

$$R = \frac{\rho h}{A_{\text{Gap}}}$$

Where ρ = specific resistance of the electrolyte,
h = equilibrium gap
A_{Gap} = Cross-sectional area of the gap

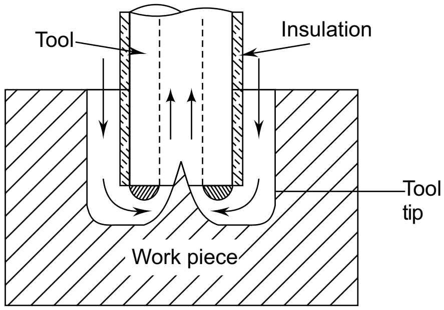


Fig. 11.30 *Electrolyte flow method in ECM*

Example 11.1

Calculate the material removal rate and the electrode feed rate in the electrochemical machining of an iron surface that is 25 mm × 25 mm in cross-section using NaCl in water as electrolyte. The gap between the tool and the work piece is 0.25 mm. The supply voltage is 12 V DC. The specific resistance of the electrolyte is 3 Ω cm.

Solution Given, $A_{\text{Gap}} = 25 \times 25 = 625\ \text{mm}^2$
$H = 0.25$ mm
$V = 12$ V
$\rho = 3\ \Omega$ cm.

For iron, Valency, $Z = 2$
Atomic weight, $A = 55.85$
Density, $\rho_a = 7860\ \text{kg/m}^3$

The gap resistance, R is given by

$$R = \frac{3 \times 0.25}{625} = 0.0012\ \Omega$$

Current, $$I = \frac{V}{R} = \frac{12}{0.0012} = 1000\ \text{A}$$

The material removal rate (MRR) in ECM (taking 100% current efficiency) is

$$\text{MRR} = \frac{AI}{ZF} = \frac{55.85 \times 1000}{2 \times 96540} = 0.2893\ \text{g/s} = 289.3 \times 10^{-6}\ \text{kg/s}$$
$$= 0.03677 \times 10^{-6}\ \text{m}^3/\text{s}$$

The feed rate of the electrode is

$$\text{Feed rate} = \frac{\text{MRR}}{\text{Surface area}} = \frac{0.03677 \times 10^{-6} \times 60}{625 \times 10^{-9}} = 3.53\ \text{mm/min}$$

Flow analysis:
To calculate the fluid flow required, match the heat generated to the heat absorbed by the electrolyte.
The heat generated in the gap, H is given by

$$H = I^2 \times R$$

Heat absorbed by the electrolyte, H_e is

$$H_e = q\rho_e c_e (\theta_B - \theta_o)$$

Where q = Flow rate of the electrolyte
ρ_e = Density of the electrolyte
c_e = Specific heat of the electrolyte
θ_B = Boiling point of the electrolyte
θ_o = Ambient temperature

Neglecting all the heat losses

$$I^2 R = q\ \rho_e\ c_e\ (\theta_B - \theta_o)$$

or, the flow rate is given by

$$q = \frac{I^2 R}{\rho_e\ c_e\ (\theta_B - \theta_o)}$$

Example 11.2

For the above example, estimate the electrolyte flow rate. Specific heat of the electrolyte is given as 0.997 cal/g°C. The ambient temperature is 35°C and the electrolyte boiling temperature is 95°C.

Solution Given,
Density of the electrolyte, $\rho_e = 1.0$ g/cm^3
Specific heat of the electrolyte, $c_e = 0.997$ cal/g °C

Boiling point of the electrolyte, $\theta_B = 95°C$
Ambient temperature, $\theta_o = 35°C$
Calculated earlier, $I = 1000$ A
$R = 0.0012\ \Omega$

The flow rate, q is given by

$$q = \frac{I^2 R}{\rho_e\, c_e\, (\theta_B - \theta_o)} = \frac{1000^2 \times 0.0012}{4.187 \times 1 \times 0.997 \times (95 - 35)}$$
$$= 4.79 \text{ cm}^3/\text{s}$$

11.3.5 ECM Tools

The properties expected of the tool materials are:

- High electrical and thermal conductivity
- Good stiffness
- Easy machinability
- High corrosion resistance

Generally aluminium, copper, brass, titanium, cupro-nickel and stainless steel are used as tool materials. Tool design requires careful considerations to maintain a constant gap over the entire work piece surface. Also the flow characteristics of the electrolyte need to be considered during the tool design. The modification of the tool profile to get the required final surface is relatively complex. It is generally done using empirical methods. Alternatively the modern complex analysis methods such as finite element can also be used to get the final tool design.

11.3.6 Process Characteristics

The material removal rates with ECM are sufficiently large and comparable with that of the conventional methods. The theoretical metal removal rates are given in Table 11.8.

Excellent surface finish of the order of 0.4 μm can be obtained with tolerances of the order of ± 0.02 mm or less. The repeatability is also good. This is possible because as noted earlier, the tool wear is almost non-existent. The process parameters that have a control on the performance of the ECM process are:

Feed rate High feed rate results in higher material removal rate. It also decreases the equilibrium-machining gap resulting in improvement of surface finish and tolerance control.

Voltage Low voltage decreases the equilibrium-machining gap and results in better surface and finer tolerance control.

Current Increased current leads to electrolyte heating, the limiting condition being the boiling point of the electrolyte. More metallic ions react with the electrolyte causing higher hydrogen evolution. Also it leads to polarised ionic layers forming at the electrodes causing voltage drops.

Electrolyte concentration Low concentration of the electrolyte decreases the machining gap and results in better surface and finer tolerance control.

Electrolyte temperature Low temperature of the electrolyte is conducive to better surface finish and tolerances.

TABLE 11.8 Theoretical metal removal rates of some metals.

Material	Atomic Weight	Valency	Density, kg/m^3	Metal Removal Rate Per 1000 A $\times$ 10^3 kg/min
Aluminium	26.97	3	2670	5.7
Chromium	51.99	2 3	7190	15.0 12.0
Copper	63.57	1 2	8960	39.6 19.8
Iron	55.85	2 3	7860	17.4 11.7
Magnesium	24.31	2	1740	7.5
Molybdenum	95.94	3 4 6	10 220	19.8 15.0 9.9
Nickel	58.71	2 3	8900	18.3 12.3
Titanium	47.90	3 4	4510	9.9 7.5
Tungsten	183.85	6 8	1930	18.9 14.4

Typical ECM parameters used are:

Current	50 to 40 000 Amperes
Current density	8 to 233 Amperes/cm^2
Voltage	4 to 30 V DC
Gap	0.025 to 0.75 mm
Electrolyte velocity	15 to 60 m/s
Electrolyte pressure	0.069 to 2.700 MPa
Electrolyte temperature	24 to 65°C
Feed rate	0.5 to 19.0 mm/min

11.3.7 Applications

ECM is used for regular production of components in a number of industries because of its many advantages as shown below:

Advantages

- Complex 3 dimensional surfaces can be machined accurately.
- Since there are no cutter marks, surface finish will be higher.
- The tool wear is practically nil which results in a large number of components produced per tool.
- The ECM process does not thermally affect the work piece.

Limitations

- Use of corrosive media as electrolytes makes it difficult to handle.
- Sharp interior edges and corners (< 0.2 mm radius) are difficult to produce.
- Very expensive machine.

Because of the nature of the ECM process, the machining surface can be at any inaccessible location as well. The tool design and tool motion can take care of reaching such surfaces as blind cavities and pockets in the walls. It is used for the machining of the gas turbine blades.

11.3.8 Electro Chemical Grinding

Electrochemical grinding (ECG) is a process that combines the electrochemical machining with the mechanical grinding operation to remove material. It uses a grinding wheel with electrically conductive abrasive bonding agent. The electrolyte is introduced into the gap between the wheel and the work piece in a manner similar to the application of grinding fluid in the conventional grinding operation as shown in Fig. 11.31. The ECG wheel is negatively charged while the work piece acts as anode.

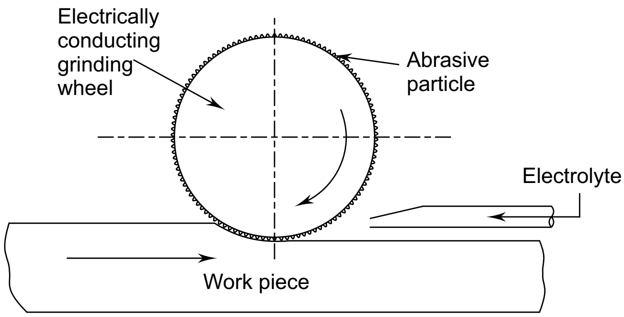


Fig. 11.31 *Schematic of an electrochemical grinding operation*

The material is removed by a combination of electrochemical action as well as grinding. The rotation of the grinding wheel draws the electrolyte into the gap. Within the contact area of the wheel and the work piece, the material is removed by the electrochemical action in the beginning. But as the wheel advances the electrolyte becomes weak and the electrochemical action reduces where the abrasive grains will be able to remove the material by mechanical action.

The material removal rates are high compared to conventional grinding processes by as much as 10 times to an average of about 1.6 cm^3/min/1000 amperes. Surface finish range from 0.15 to 0.40 µm. Tolerances are not as good as conventional grinding and range between ±0.012 mm to ±0.025 mm.

The most common application of ECG is the grinding of tungsten carbide tool inserts.

11.4 ULTRASONIC MACHINING

Ultrasonic machining (USM) is a mechanical metal removal process for brittle materials by using high frequency oscillations of a shaped tool using abrasive slurry. The term ultrasonic refers to the frequency range above the audible range which is above 16 kHz.

A schematic of the ultrasonic machining set-up is shown in Fig. 11.32. The transducer generates the high frequency vibrations of the order of 20 to 30 kHz with an amplitude of the order of 0.02 mm. This vibration is transmitted to the tool made of soft material through a mechanical coupler known as tool holder. The tool shape is a close complimentary shape of the final surface to be generated.

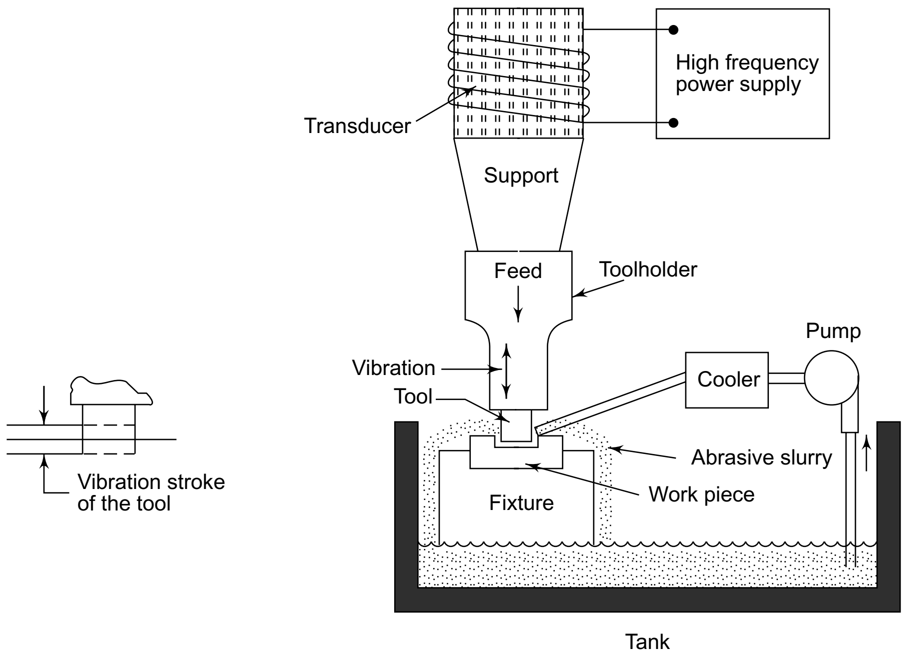


Fig. 11.32 *Schematic of an ultrasonic machining operation*

The tool while oscillating would be pressed against the work piece and fed continuously. A slurry of abrasive grains suspended in a liquid is fed into the cutting zone under pressure as shown in Fig. 11.33. The slurry is about 30% concentration. Abrasive particles are driven into the work surface by the oscillating tool. The force is typically about 150 000 times the weight of the individual grains. A small crater will be formed at the impact site of the grain, if the work piece is brittle. A very large number of such small craters remove sufficiently large material from the work piece.

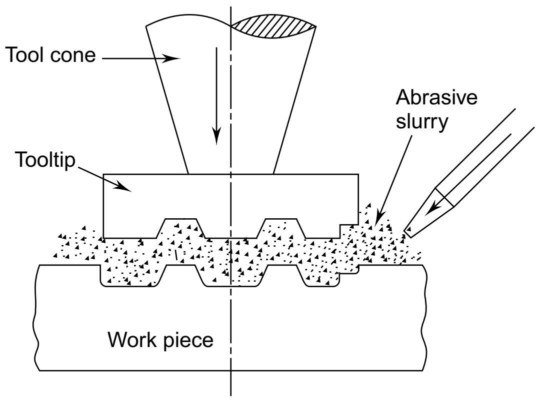


Fig. 11.33 *Schematic of the material removal process in ultrasonic machining*

As the material is removed, the tool is gradually advanced into the work piece by a servo mechanism such that a constant gap is maintained between the tool and the work piece. Finally the shape of the tool is impressed into the work piece as shown in Fig. 11.33. The material removal rates in USM are relatively small, but in materials which are brittle, this is the only way to produce economically complex cavities without breaking the work piece. Since there is no direct contact between the work piece and the tool, fragile work pieces can be conveniently used in USM.

Transducer

The transducer in USM is utilised to convert the electrical energy to vibratory motion utilising either the piezoelectric or magnetostrictive principles. Piezoelectric materials such as quartz or lead zirconate titanate increase in size when an electric current is applied to them and return to normal size when the current is removed. Such piezoelectric transducers can be used up to a power of 900 W. The magnetostrictive transducers are constructed from nickel, permalloy (Ni 45% and Fe 55%) or permedur (Co 49%, Fe 49% and V 2%) plates when exposed to strong magnetic field will change the length. These are more rugged than the piezoelectric transducers and can be used for higher power up to 2400 W. However, their conversion efficiency is low of the order of 20 to 35% and hence get heated when in use. So a separate cooling arrangement has to be made to remove this waste heat.

Tool Cone (Horn)

Tool cone amplifies the mechanical energy produced by the transducer. Horn mechanically amplifies the vibratory energy to give the required force-amplitude ratio. The horn must be tuned to the required frequency. It acts as a resonator to amplify the signal. It should have adequate strength. Titanium, monel and stainless steels are generally used as tool cone materials. Stainless steel is used only for low amplitude applications.

Tool tip is attached to the cone by means of silver brazing or by screws. Shape of the tool and its dimensions are governed by the size of abrasive used. For example a 11.98 mm diameter tool tip may produce a 12 ± 0.005 mm hole when a 600 abrasive grit is used. Length of the tool should be short, since massive tools absorb the vibration energy reducing the efficiency of machining. Also long tools cause over stressing of the tool and the brazed point. Typically they are about 25 mm long.

Slenderness (length to diameter) ratio of the tool should not be greater than 20. The tool material should be tough and ductile, but should not be too soft. Generally, low carbon steel and stainless steel are good tool materials.

Abrasive Slurry

A large variety of abrasives are available for using in USM. The abrasive selected should be harder than the material being machined. Typical abrasives used are aluminium oxide, silicon carbide and boron carbide. Aluminium oxide wears fast and is good for glass and ceramics. Boron carbide is the most popularly used abrasive, harder than silicon carbide and more expensive. It has faster material removal rate and can withstand high vibrational forces. It is best for tungsten carbide, tool steel and precious stones. Diamond dust is sometimes used for good accuracy, surface finish and cutting rate. It is used for diamonds and rubies.

Abrasive grain sizes used are from 200 to 2000. The choice of grain size depends upon the finish desired. Generally sizes 20 to 400 are used for roughing while 800 to 1000 grit sizes are used for finishing.

The abrasive is suspended in a liquid with about 30 to 60% by volume of abrasive. The liquid serves several functions. It acts as an acoustic bond between the vibrating tool and the work piece, to give efficient transfer of energy between the two. It acts as a coolant on the tool face. It also provides a medium to carry the abrasive to the cutting zone and carry the spent abrasive and swarf away.

The liquid requires the following properties.

- A density approximately equal to that of the abrasive.
- Good wetting properties, to wet the tool, the work and the abrasive.
- A low viscosity to carry the abrasive down the sides of the hole between tool and work piece.
- A high thermal conductivity and high specific heat for efficient removal of heat from the cutting zone.

Water is most commonly used while benzene and glycerol are also used.

Typical process capability of the USM process for a variety of materials is given in Table 11.9. It can be noted from the table that the wear ratio increases directly for softer work piece materials. For example in the case of tool steel the wear ratio is 1:1 which means that for every mm^3 of material removed, equal amount of material is also removed from the tool.

TABLE 11.9 **Typical process characteristics of USM process (Tool material: Low carbon steel, Slurry –30 to 40% of 180–240 grit boron carbide, Amplitude of vibration –0.025 to 0.035 mm, frequency –25 kHz)**

Work Material	Material Removal Rate		Max. Practical Tool Area, mm^2	Wear Ratio
	Volume, mm^3/min	Penetration Rate mm/min		
Glass	425	3.8	2580	100:1
Ceramic	185	1.5	1935	75:1
Ferrite	390	3.2	2260	100:1
Quartz	200	1.7	1935	50:1
Tungsten carbide	40	0.4	775	1.5:1
Tool steel	30	0.3	775	1:1

Advantages of USM:

- USM is used for machining hard and brittle materials to complex shapes with good accuracy and reasonable surface finish.
- It is not affected by the electrical or chemical characteristics of the work material.
- Holes of any shape can be produced.
- It has no high speed moving parts. Working is not hazardous.
- Power consumption is about 0.1 Watt Hour/mm^3 for glass and about 5.0 Watt Hour/mm^3.

Limitations of USM:

- Metal removal rates are low.
- Depth of hole produced is limited.
- Tool wear is high and sharp corners cannot be produced.
- Flat surfaces cannot be produced at the bottom of the cavity because of the ineffective slurry distribution.

11.5 CHEMICAL MACHINING

Chemical machining or chemical milling is a process used to dissolve the work piece material in chemical solutions. Since the chemical solutions used have the ability to dissolve all of the work piece material, the parts which are not to be dissolved would have to be covered with a mask which resists the chemical action of the solution, so that only the unmasked portion gets removed by the chemical solutions.

The steps involved in a typical chemical machining operation are:

- Clean the work piece thoroughly. This is necessary to ensure that the masking material will adhere to the work piece well to reduce any possibility of stray etching due to maskant de-bonding.
- Apply a chemical resistant mask on the work piece surface where no material is to be removed.
- Dip the work piece into the chemical solution called etchant and leave it for sufficient time to get the necessary depth of etching. The etchant is either continuously sprayed onto the work piece surface or the part is immersed in a tank of constantly agitating etchant solution. This helps in removing the material uniformly from all the exposed surfaces of the part. The strength of the etchant is maintained since it becomes weak by absorbing the work piece material with time.
- Remove the mask and clean the work piece.

During the etching process, the removal of material takes place in the depth wise unexposed portions as well as in the inward direction under the mask as shown in Fig. 11.34. The distance etched under the mask is termed as undercut, while the distance etched in the exposed portion is termed as the depth of cut.

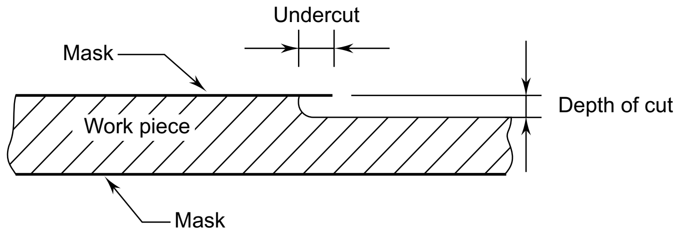


Fig. 11.34 *Chemical machining process*

The undercut is dependent upon the depth of cut, the strength of the etchant solution and the work piece material. It is necessary during the design of the maskant to take the undercut into account to get the actual size required. The relationship between the undercut and the depth of cut is termed as the etch factor and defined as:

$$\text{Etch factor} = \frac{\text{Undercut}}{\text{Depth of cut}}$$

Masks

The selection of maskants for a given application depends upon a number of important considerations. They are:

Chemical resistance Thicker maskants can resist the erosive action of the etchant for a longer period and hence larger depth of cut is possible with them.

Quantity of parts For large volume production, the masking process should be as simple as possible to reduce the cost.

Ease of removal Delicate parts require that the maskant be easier to remove since the mask should be removed before the part is used.

Required resolution Thicker masks are generally less accurate, though the method of making the mask also contributes to it.

There are three generally followed methods for making the masks.

Cut and peel This is generally a neoprene, butyl or vinyl based thick material, which is applied by dip, spraying or flow coating. Afterwards the pattern is scribed on the mask using a pattern and peeled away exposing the areas to be etched. The thickness of the mask can range from 0.025 to 0.125 mm, which allows it to be used for large depths of cut. It can also be used for step etching application.

Screen-printing The maskant is transferred to the work piece using a fine mesh screen such as those used for silk screen-printing. This is fast and economical for large volume production with relatively less accuracy. The thickness of the mask is relatively small (less than 0.05 mm) and hence used for shallower etching depths.

Photo resist masks This is the most versatile compared to the other mask making technologies. Due to the use of these masks the process is called 'Photochemical machining (PCM)', though the rest of the processing remains the same as chemical machining.

The first step in preparing the mask is developing the engineering drawing of the pattern in a scale 2 to 20 times larger. The larger drawing helps in improving the accuracy when it is reduced photographically to the actual size. The etching factor has to be taken into account while making the pattern.

In the next step the pattern is photographed and reduced to the more accurate actual size master transparency pattern. This master pattern will be used to transfer the mask onto the work piece using a photoresist coating. The steps involved in preparing the mask are shown in Fig. 11.35.

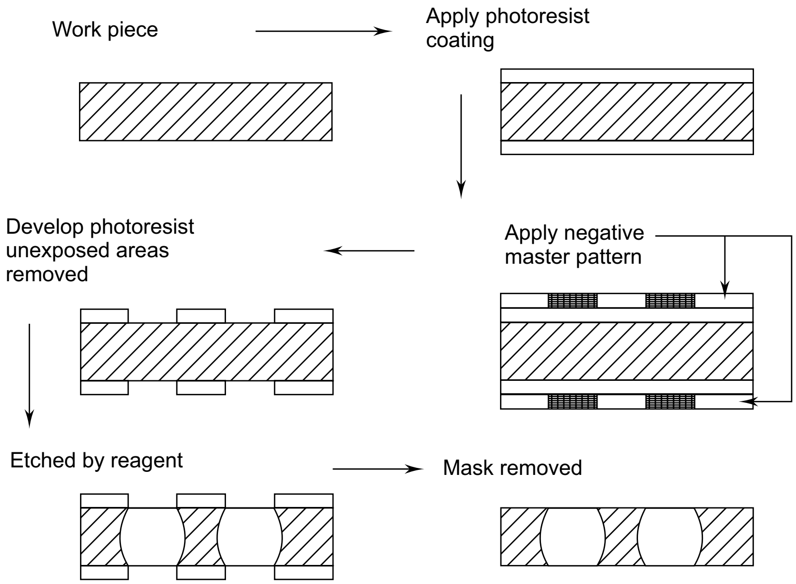


Fig. 11.35 *Photo Chemical machining process*

The work piece is carefully cleaned of the entire dirt and oxides. Then it is coated with a light activated, etchant resistant material called photoresist and baked dry. After this the work piece is exposed to a strong ultra-violet light to expose the photoresist. After the exposure the photoresist is developed to remove the coating from all the areas where etching is desired. After the photoresist is developed, the component is chemically etched to complete the operation.

Etchants

The function served by the etchant is to dissolve the metal from the part by converting it to a salt, which then goes into the solution. The choice of an etchant depends upon a number of factors. They are:

Surface finish Some etchants promote the formation of surface oxides, which is detrimental to the surface finish.

Removal rate Active etchants remove material faster reducing the machining time, but also are likely to attack the maskant, gives poor surface finish and are also likely to generate more heat.

Material type The etchant while removing the material should not cause inter-granular attacks, hydrogen embrittlement or stress corrosion cracking.

Some etchants generally used are iron chloride, chromic acid, hydro flouric acid and nitric acid.

Applications

Chemical machining is generally used when very small amounts of material are to be removed from the surface in any application. In the aerospace application, (Fig. 11. 36) removal of a large volume of unwanted material from the surface to reduce the weight, thereby increasing the stiffness to weight ratio, can be conveniently done with chemical machining.

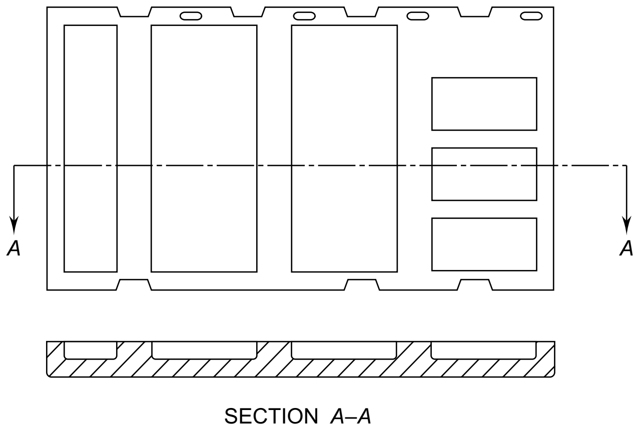


Fig. 11.36 *Example of Chemical machining process to remove the bulk of material form an aircraft part to increase the stiffness to weight ratio*

The components of complex profiles in very thin metals used in instrumentation and sensors are machined by photochemical machining. Chemical machining can also do engraving of any type on the metal.

The main advantage of the process is that a large number of work pieces can be simultaneously machined, thereby improving the productivity. It requires very little capital investment for the basic equipment. Tooling costs are also low.

11.6 LASER BEAM MACHINING

Laser (light amplification by stimulated emission of radiation) beam machining (LBM) utilises the energy from the coherent light beams called laser (light amplification by stimulated emission of radiation). The basic principle utilised in LBM is that under proper conditions light energy of a particular frequency is used to stimulate the electrons in an atom to emit additional light with exactly the same characteristics of the original light source.

The first laser was invented by Maiman in May, 1960 which is a solid ruby laser. Since then a number of lasers were invented there after - Uranium Laser by IBM labs in 1960, Helium-Neon Laser by Bell Laboratories in 1961, semiconductor laser by Robert Hall at General Electric Labs in 1962, Nd:YAG (Neodymium-doped Yttrium Aluminium Garnet) laser and CO_2 (carbon dioxide) laser by Bell Laboratories in 1964, argon ion laser in 1964, and a number of others.

One of the main important properties of laser is the coherency, with a diverging angle of less than 1 to 2 mrad. This low divergence helps in achieving high intensity of energy. In addition, if a focusing lens is used then it is possible to melt the work piece material. Typical power densities used for cutting are of the order of 1.5×10^3 to 1.5×10^5 kW/cm^2. Because of the large power requirement, the eligible lasers for this are - CO_2 lasers, Nd-YAG or Nd-glass lasers and Excimer lasers, etc. At the higher end of this power scale, the material changes directly from solid state to vapour state without going through the molten state and is used in some laser markers.

There are a number of advantages claimed for the use of laser beam machining because of which it has become the more common material processing method in many industries. Some of these are:

1. Laser beam machining does not apply any direct force because it is a non-contact machining. This allows for the use of very little clamping force on the work piece to be applied. This is a big advantage particularly when machining delicate materials.
2. Laser beam machining can be localized to a small area thereby removing a very small amount of material scale. This results in a very small kerf in LBM while maintaining a low depth, which gives it a great flexibility.
3. Since the heat is localized, Heat Affected Zone (HAZ) in laser beam machining is small. Also the distortion in laser beam machining is negligible.
4. Laser beam machining can be applied to any material that is not reflective and can properly absorb the laser irradiation.
5. There is no need to have multiple passes to complete machining in laser beam machining. Also high aspect ratio holes with very small diameters can be done using lasers.
6. Laser light can be transmitted and reflected using simple mechanisms at very high speeds. This gives the laser beam machining a lot more flexibility.

The Nd:YAG system is a solid-state laser. The Neodymium atoms are responsible for the laser action and are held in suspension in a low-percentage mixture within the YAG synthetic crystal. This type of laser develops about 40 watts of laser energy per cubic centimetre of crystal volume, with efficiency between 2 to 4 percent. The Nd:YAG systems emit radiation at 1.06 μm wavelength. The YAG laser is especially suited for drilling and cutting metals. For typical drilling operations, certain YAG systems can penetrate up to 25 mm thickness of material. Hole sizes range from 0.1 mm up to a maximum of about 1 mm without beam manipulation. With beam or part manipulation, almost any size or shape of hole can be cut. Also, the laser does well at cutting holes at an angle to the surface, down to 20 degrees from the surface.

CO_2 (carbon dioxide) gas lasers use a mixture of CO_2, helium and nitrogen gases in a laser tube to create a laser beam. The laser action is created by the CO_2 molecule. The helium is utilized to cool the gas mixture by transferring the energy to the water-cooled walls of the flow system. The nitrogen acts as a catalyst to enhance the CO_2 laser action. CO_2 laser systems are excellent for cutting organic materials, such as plastics, rubbers, cloth and paper. They are also excellent for cutting, welding and heat-treating iron and its alloys. Clear plastic and glass can be cut with a CO_2 laser.

11.6.1 Laser Drilling

A number of processes are possible using the laser beam such as laser drilling, laser cutting and laser grooving, marking or scribing. A schematic of the laser drilling setup is shown in Fig. 11.37. The laser beam is focussed with the help of the lens and the work piece is placed near the focal point of the lens. A short pulse of laser melts and vaporises the material. The explosive escape of the vaporised metal helps in removing most of the molten metal from the hole as tiny droplets. Any of the molten metal not removed will be re-solidified along the walls of the hole. This process can be used for machining small holes of 0.125 to 1.25 mm with a length to diameter ratio up to 100. Hole size depends largely on set-up and spot size of the laser beam. Laser drilling of metals is used to produce very small orifices for nozzles, cooling channels in air turbine blades, drilling of circuit board, etc.

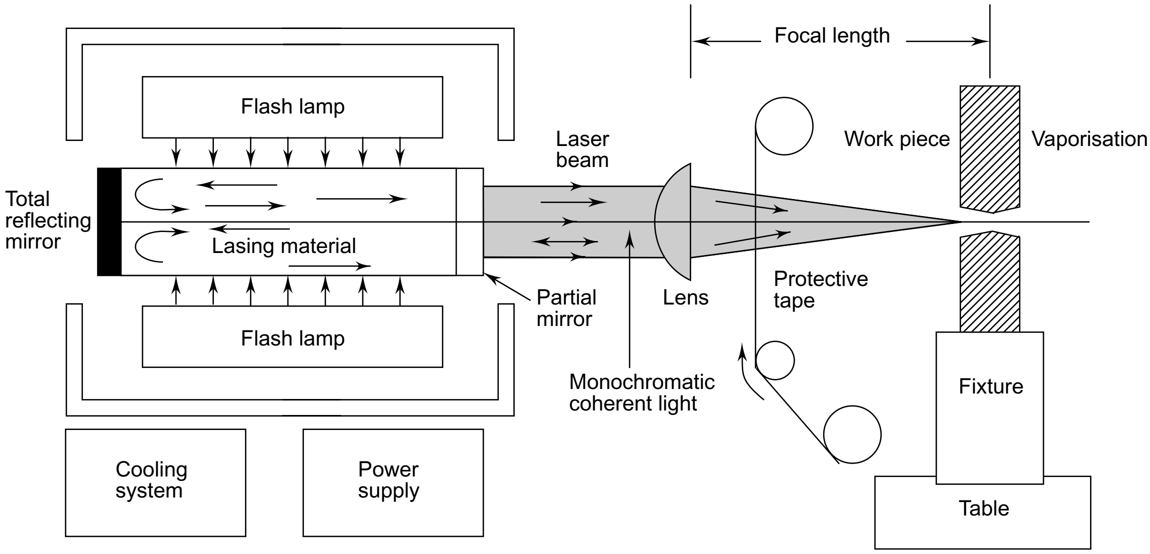


Fig. 11.37 *Schematic setup of a laser drilling operation*

The laser-drilled holes exhibit a taper and also lack a high degree of roundness. Hole size can be controlled to within 0.025 mm, with some taper being evident in thick materials. Holes larger than 1.25 mm cannot be drilled because the power density will decrease. Hence laser cutting is used rather than laser drilling. In the laser cutting operation, a high velocity gas jet is used in conjunction with the laser beam. The gas jet helps to rapidly remove the metal from the hole.

11.6.2 Laser Cutting

Laser cutting is similar to flame cutting where work pieces can be cut along lines or curves. Since the laser is not contacting the work piece directly, it is possible to cut thin work pieces . CO_2 laser and Nd:YAG laser are the most popular lasers because of their ability to provide high power, such as above 1 kW.

Based on the method used, laser cutting can be basically divided into two types. Evaporative laser cutting is the process in which laser vaporizes the material directly for some organic materials such as paper, cloth or polymers. For cutting metals laser directly melts the target material and the gas jet blows the molten material away thus forming a hole. Then the nozzle continues to move in the direction of the profile to be cut. The energy required for cutting in this process is much less compared to the evaporative laser cutting. In this way the requirement on laser energy is lower compared with vaporization cutting.

Thicknesses of up to 25 mm can be cut with a laser, with a kerf down to a tenth of a millimetre. The biggest advantage of laser cutting is that very complex shapes, with corners that have radii of a tenth of a millimetre and good, almost burr-free edge quality, can be cut at high speeds.

Generally, laser cutting is good for cutting thinner materials. For example, maximum work piece thickness that can be cut in carbon steel with a 1200 W CO_2 laser is about 10 mm using oxygen assist. However the thickness becomes much smaller for those materials that do not exothermically react with oxygen. The kerf is narrow in laser cutting, as little as 0.1 mm for thin materials. Also the resultant heat affected zones are negligible, particularly for mild and low carbon steel. Cut edges are smooth, clean, and square.

11.7 ABRASIVE WATER JET MACHINING (AWJM)

11.7.1 Principle

Abrasive Water Jet Machining is a process that uses a very high speed (supersonic about 2.5 Mach number) water jet mixed with abrasives to cut any type of material without, in any way, affecting the work material or the environment. If the process is used without any abrasives, it is called as water jet machining. The inlet water is typically pressurized between 1300 to 4000 bar which is forced through a tiny hole in the jewel, which is typically 0.18–0.4 mm in diameter creating a very high velocity beam of water. The process is generally used for cutting operations, but can also be used for selective removal of the material, which are discussed later.

Water jet cutting has started in the late 1960's for cutting space age alloys as the conventional processes were adding thermal stresses due to the heat produced. High pressure water jets that were used for cleaning in the mining industry were used to cut composites and other space age materials. The commercial machines were available since 1971. In the early 1980's abrasives were introduced into the water jet, thereby the AWJM process was born. Originally garnet abrasive was introduced into the jet stream to cut harder materials such as steel, glass and concrete.

The AWJM machines aim a highly focused, supersonic stream of water at the material such that it can cut composites smoothly by eroding them without generating any heat. Thus the AWJM process eliminates all the thermal and mechanical distortion caused by conventional cutting methods. Also the water jet nozzle can be directed at any angle to the material thereby allowing for angled cuts. For cutting soft materials such as textiles and food stuffs, pure water without any abrasives is used.

11.7.2 Equipment

A typical AWJM system consists of the following major sub systems:

- An intensifier pump to provide high-pressure water,
- The abrasive delivery system and a cutting head for producing the abrasive water jet,
- Computer controlled manipulator to provide the desired motion of the cutting head, and
- A catcher that dissipates the remaining jet energy after cutting.

A typical system is schematically is shown in Fig. 11.38.

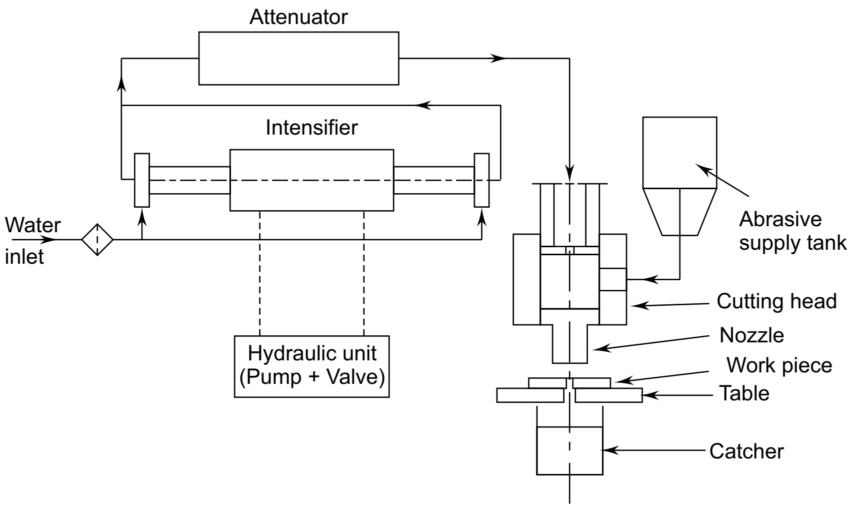


Fig. 11.38 *Schematic of AWJ Machine*

For the proper functioning of the system, pure water without any contaminants is to be used. For low pressure applications up to 280 MPa direct drive pumps are used. However for very high pressures, these become unreliable so intensifiers upto 4000 MPa are used to boost the pressure. In the intensifiers there are two circuits, a hydraulic circuit and a water circuit. The intensifier acts as an amplifier as it converts the energy from the low-pressure hydraulic fluid into ultra-high-pressure water as shown in Fig. 11.39.

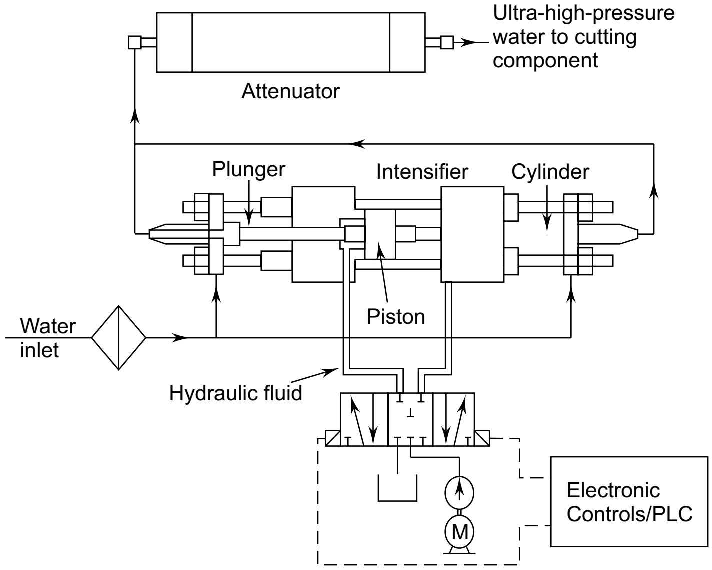


Fig. 11.39 *The hydraulic circuit of the intensifier arrangement in AWJM*

The hydraulic pump pressurises the oil to about 200 MPa. This pressurized oil is sent to the manifold where manifold's valves create the stroking action of the intensifier by sending hydraulic oil to one side of the piston. The intensifier is a reciprocating pump, with the piston reciprocating back and forth, delivering high-pressure water out from one side of the intensifier while low-pressure water fills in the other side. The hydraulic oil pushes against the piston. A plunger with a face area of 20 times less than the piston pushes against the water. Therefore, the 200 MPa oil pressure is "intensified" twenty times, yielding 4000 MPa water pressure. The attenuator damps pressure fluctuations from the intensifier and delivers a constant and steady stream of ultra-high-pressure water to the cutting tool.

The cutting head converts the pressurised water into a cutting instrument. Abrasive particles enter the water jet in the mixing chamber as shown in Fig. 11.40. The abrasive particles generally used are crushed garnet, olivine sand, or aluminium oxide, with particle sizes ranging from 0.2 to 0.5 mm. The typical hardnesses of the abrasive materials is given in Table 11.10. Garnet is widely used because of its relatively low cost and high cutting speed. Abrasive particles enter the water jet with negligible velocity. They are accelerated by the incoming water jet in such a way that the velocity direction of the abrasive particles is nearly parallel to the direction of the water jet. The smaller the inner diameter of the focusing tube, the more concentrated the total energy. The minimum diameter is related to the abrasive particle size, which is about five times or more of the particle diameter. Longer tubes produce a more coherent jet compared with shorter tubes, but the wall friction results in lower abrasive jet velocities. New materials have increased the focusing tube life from 3 or 4 h to 100 h. Some typical lives of mixing tube materials are given in Table 11.11.

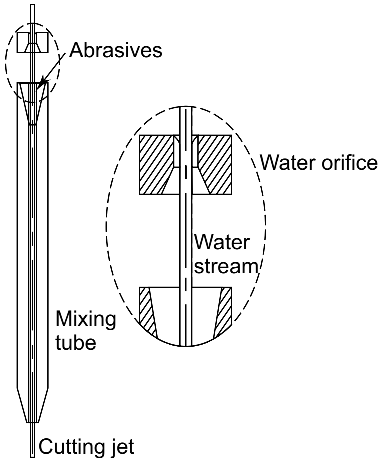


Fig. 11.40 *Close-up view of the nozzle in the AWJM process*

Table 11.10 **Hardness values of abrasive materials**

Material	Hardness	
	Mohs	Knoop
Aluminium oxide	8–9	2100
Garnet	7.5	1350
Olivine	5.5	1100
Silicon carbide	9.15	2500

Table 11.11 **Mixing tube materials and their lives**

Tube Material	Life
Tungsten carbide	4 to 6 hours
Low cost Composite carbide	35 to 60 hours
Mid-life Composite carbide	80 to 90 hours
Premium Composite carbide	100 to 150 hours

A tank kept below the nozzle catches the water and abrasives after they have completed the cutting action. A lot of energy is still left in the jet stream and therefore the catcher should be robust, reliable and have a long life. It is designed as an energy absorbing bed, filled with water, steel pellets or alternatively ceramic pellets, as the absorbing media. If filled with water, it should have a depth of at least 0.6 m to be able to dissipate the jet energy.

11.7.3 Process Parameters

The quality of the part in AWJM depends upon a number of process parameters that need to be properly taken care of. Some of the important parameters are:

- Jet velocity
- Feed rate
- Abrasive used and their size
- Work material and its thickness

Jet velocity

The jet has to impinge on the work material with a minimum velocity for the material removal to take place, since that indicates the kinetic energy present in the abrasive grains. As the velocity increases the material removal rate increases which will have an effect on the quality of the surface produced.

The velocity, V for a given quality is given by the following empirical equation [Olsen]

$$V = \left(\frac{fa \times M \times P^{1.594} d^{1.374} \ Ma^{0.343}}{163 \ Q \ H \ Dm^{0.618}} \right)^{1.15}$$

Where P = Stagnation pressure of the water jet in Ksi (50,000 psi = 50Ksi)
d = Orifice Diameter in inch typically 0.014 inches
Ma = Abrasive Flow Rate in lb/min typically 0.8 lb/min
fa = Abrasive Factor (1.0 for garnet)

Q = Quality. Set to 1.0 For Calculating Separation Speed
H = Material Thickness in inch
Dm = Mixing Tube Diameter in inch typically 0.030 inches
V = Traverse Speed in inch/min
M = Machinability of Material as given in Table 11.12

Table 11.12 Machinability (M) of work materials

Work Material	Machinability
Hardened Tool Steel	80
Mild Steel	87
Copper	110
Titanium	115
Aluminium	213
Granite	322
Glass	327
Plexiglass	690
Pine Wood	2637

Feed Rate

Feed rate is the speed with which the jet moves past the work piece surface to generate the cut. As the feed rate of the jet increases the jet has lower time to erode the surface and consequently the surface finish of the cut edge will be rough. It also affects the part quality by increasing the drag as discussed later. It is important that the feed rate be controlled such that it is increased to the maximum possible for straight cuts for the given surface quality, and decrease at the corners where the drag is likely to be a problem. Generally the operating software of the machine should achieve this function.

Part Accuracy

Some of the accuracy problems that need to be given proper attention during the operation of AWJM are as follows:

Jet Drag

As the abrasive jet moves through the material, the lower portion of the jet lags behind the upper section as shown in Fig. 11.41. The amount of drag depends upon a number of parameters such as the flow rate, material thickness and jet speed. If the cut is straight then this may not constitute a problem. However, when the jet has to make a corner or bend, the jet should be slowed to control the amount of lag. This is normally accomplished in the software that controls the jet velocity to some extent. However it is always not possible and as a result the tolerances on the part due to the drag need to be considered particularly as the part thickness increases.

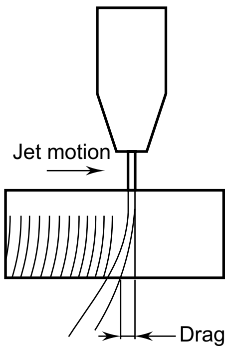


Fig. 11.41 *Water jet lags the leading edge during the cutting process in the AWJM process*

Jet Kerf

Kerf is the thickness of the cut made by the jet. This is controlled by the rate at which the nozzle moves across the work piece surface. The slower a jet nozzle moves across the material being cut, the wider is the kerf width. This effect is generally negligible for harder or thinner material, but is considerable in thick material or soft material.

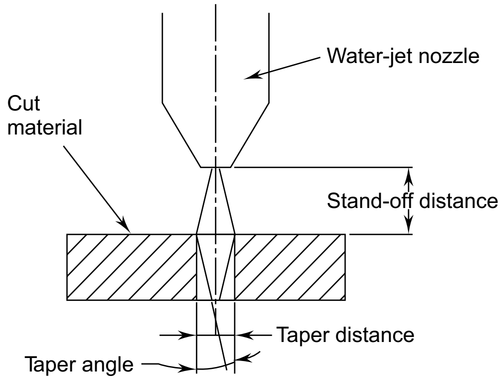


Fig. 11.42 *Water jet lags the leading edge during the cutting process in the AWJM process*

Taper

As the jet moves along the thickness of the material it loses its kinetic energy, and thereby its cutting ability. As a result as the cut thickness increases, the size of the cut decreases as shown in Fig. 11.42. This is termed as taper. Taper becomes larger as the material thickness and traverse speed increase. In very thick materials the taper becomes barrel shaped, where the top and bottom of the part may measure out accurately, but the middle part may be significantly off. Also it is noticed that the softer materials exhibit larger taper.

Standoff

Standoff is the distance between the face of the nozzle and the surface of the work piece as shown in Fig. 11.42. As the standoff increases, the depth of cut decreases and the taper increases. Too small a standoff will not have enough room for the jet to start the hole. A typical standoff used is about 1 mm. If the upper surface of the part being cut is having a very rough surface then the standoff is likely to vary and this will have an effect on the taper produced.

11.7.4 Applications

Some of the advantages that can be expected from the use of abrasive water jet machining are:

- Cuts through any material irrespective of the hardness or any other material characteristic.
- Costs with AWJM are less than other machining methods.
- The quality of the cut achieved by AWJ machine virtually eliminates additional finishing. In most of the cases no additional finishing is required while in some cases, secondary processes are greatly reduced.
- AWJM is faster than other methods, especially for materials that would otherwise require a lot of additional finishing.
- AWJM does not generate a lot of heat during the cutting process. As a result, there is no heat-affected zone. Most of the heat generated during the AWJM process dissipates through the removed material.
- Scrap material left after using the AWJM remains unaltered and hence can be reused.
- Since the side forces are low, it is possible to cut thin (as low as 0.5 mm) walled parts. Also it is possible to have very close nesting thereby saving the scrap material.
- Very little fixturing is required for most parts.
- In many cases, material can be stacked to decrease production costs.
- Since the jet does not affect the left over material scrap material can be minimized. In fact, parts can be nested very close to each other to maximize the material utilization. It is even possible to share the cutting line between parts.

- No start hole is required such as for wire EDM. The water jet nozzle can be placed where the actual cutting is to begin.
- Since the water jet cuts with very little force, the amount of burr generated is extremely small, if at all present.

A comparison of the different processes that can be used for cutting is given in Table 11.13.

Table 11.13 Comparison of different processes for cutting

	Process		
	Wire EDM	Laser	Water Jet
Material	Metals	Non reflective metals	Any
Thickness (mm)	25 and above	Below 6	Any size
Accuracy (mm)	0.0025	0.025	0.025
Feed rate	Extremely slow	Fast	Fast
Heat affected zone	Yes	Yes	None
Cut edge quality	High	Low	Medium
Pre processing	Pre drill to thread the wire	None	None
Post processing	Remove HAZ	Remove HAZ	None
Machine cost	Low	High	Medium

11.8 ELECTRON BEAM MACHINING (EBM)

The heat source in electron beam machining (EBM) is a focused pulsating beam of high velocity electrons. The electron beam upon impinging the work piece releases the necessary heat by converting its kinetic energy. A schematic sketch of the electron beam machining is shown in Fig. 11.43.

The cathode (filament generally made of tungsten or tantalum and heated to 2500°C) within the electron gun is the source of a stream of electrons. These electrons are accelerated towards the anode because of the large potential difference that exists between them. The potential differences that are used are of the order of 30 kV to 175 kV. The higher the potential difference, higher would be the acceleration. The current levels are low ranging between 50 mA to 1000 mA. Depending on the accelerating voltage, the electrons would travel at the speed of 50 000 to 200 000 km/s. The material removal rate depends on this electron speed which in turn is dependent upon the accelerating voltage.

Unlike in electron beam welding, the gun in EBM is used in pulsed mode. The electron beam is focused by means of an electromagnetic lens so that the energy is released in a small area. The high-energy electron beam can be focussed on the work piece with a spot size of 10 to 100µm. When the high velocity electron beam strikes the work piece all the kinetic energy is converted to heat. As these electrons penetrate the metal, the material that is directly in the path is heated, melted or vaporised depending upon the process parameters and the material characteristics. More often in EBM a combination of melting and evaporation is used in such a way that the generated vapour pressure ensures ejection of the molten metal from the spot.

The time interval of contact between the beam and the metal is maintained between 10 µs and 10 ms to ensure that the liquid metal in the spot is completely ejected. The process parameters are selected so as to ensure that the liquid metal is completely ejected and the thickness of the heat affected zone around the machined region is of minimum thickness.

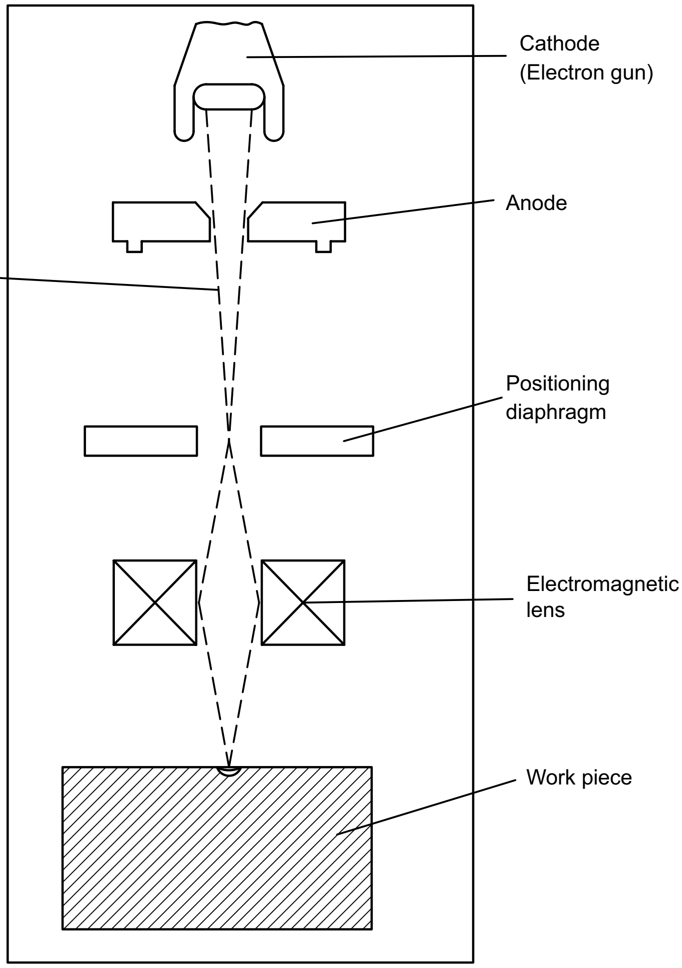


Fig. 11.43 *Schematic of electron beam machining set up with work piece in a vacuum chamber*

It is necessary that electron beam has to remain in a vacuum chamber with a level of vacuum in the order of 10^{-1} to 10^{-3} mTorr. Maintenance of suitable vacuum is essential so that electrons do not lose their energy colliding with the air molecules and a significant life of the cathode cartridge is obtained. Such vacuum is achieved and maintained using a combination of rotary pump and diffusion pump. The process parameters, which directly affect the machining characteristics in electron beam machining are:

- The accelerating voltage, 30 kV to 175 kV
- The beam current, 200 μA to 1 A
- Pulse duration, 50 μs to 15 ms
- Energy per pulse, > 100 J/pulse
- Spot size, 10 to 100μm

11.8.1 Applications of EBM

A wide range of metals and non-metals kept in vacuum can be cut using EBM. EBM can drill holes with a very high aspect ratio up to 100:1. Also holes as small as 100 μm can be drilled using EBM. EBM will not form burr and provides a rounding edge for the hole. Though this process utilizes heat to remove the metal similar to EDM, the heat affected zone is relatively small of the order of 20 to 30 μm. Since EBM does not apply any force on the work piece, simple work holding will be sufficient.

Typical applications of EBM are gas orifices for pressure-differential devices, wire-drawing dies, light-ray orifices, round or profile shaped holes on sleeve valves, rocket-fuel injectors, or injection nozzles on diesel engines.

11.8.2 Advantages and Limitations of EBM

Advantages

- EBM can be used to cut very small holes of the order of 100 μm.
- EBM can be used to cut high aspect ratio holes of the order of 100:1.
- EBM is a fast process. Since the beam of electrons move at a very high velocity it is possible to complete the drilling process in a short time.
- Very high accuracies of the order of ±0.03 mm can be achieved in nearly all materials. Occasionally a high accuracy of ±0.005 mm can also be achieved when needed.
- Since EBM will not have any cutting forces acting on the work piece thin and fragile work pieces can be machined without any distortion.
- Utilizing the CNC table for the machine it can accommodate small batches to achieve on-demand machining.
- It is versatile equipment. The same equipment can be used for annealing and/or welding since it will not use any other tool.

Limitations

- Since the process requires a vacuum chamber EBM is limited to certain part sizes. Also the time required to achieve the desired vacuum is significant.
- EBM equipment is expensive and can only be justified for the type of part dimensions and accuracy requirements.
- The hole shape is affected by the depth of the work piece. Because of the divergence of the beam away from the focal point the hole will have an hour glass shape.

11.9 ION BEAM MACHINING (IBM)

Ion beam machining process appears similar to EBM, but the actual material removal is not a thermal process. It is done by displacing individual atoms from the surface by a process called sputtering. When an ion strikes the surface of a material it usually collides with an atom there. Material removal in IBM takes place by the transfer of momentum from the incident ions to the atoms that are present in the surface of the material. This transfer of momentum results in an atom to be removed from the surface, while the ion is deflected away from the material.

Typical ion beam machining equipment (Fig. 11.44) consists of:

- A plasma source that generates the ions
- Extraction grids for removing the ions from the plasma, and accelerating them towards the substrate (or work piece)
- A table for holding the work piece.

Heated filament, usually tungsten acts as the cathode, from which electrons are accelerated by means of high voltage towards the anode. During the passage of these electrons from the cathode toward the anode,

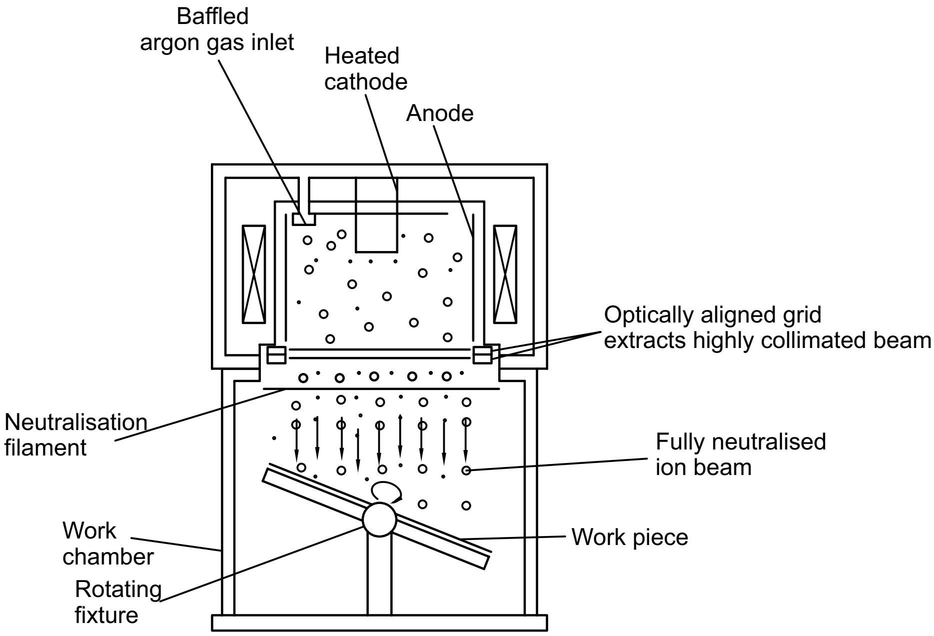


Fig. 11.44 *Schematic of ion beam machining set up with work piece in a vacuum chamber*

they interact with argon atoms in the plasma source, to produce argon ions. The produced ions are then extracted from the plasma towards the work piece, which is mounted on a water-cooled table having a tilting angle of 0° to 80°. The ions are removed from the plasma by means of extraction grids. The grids are normally made of two or three arrays of perforated sheets of carbon or molybdenum that can withstand erosion by ion bombardment. The perforations in each of the sheets are aligned above one another.

Ion beam machining can be utilized to etch selectively materials typically the silicon or gallium arsenide wafers by utilizing the masking tapes similar to chemical machining without the use of dangerous chemicals. IBM has been used in smoothing of laser mirrors as well as reducing the thickness of thin films without affecting their surface finish [McGeough, 1988]. Also polishing and shaping of optical surfaces by direct sputtering of pre-forms in glass, silica, and diamond can be done by using patterning masks [El-Hofy, 2005].

11.10 PLASMA ARC MACHINING (PAM)

When a gas is raised to high temperatures of the order of 30,000°C, the atoms get ionized. The phase of the ionised gas is termed as plasma. In view of the high temperature available in plasma arc it has been used for sheet and plate cutting operations in place of the traditional oxy-fuel cutting.

In plasma arc machining an arc is generated between the hot cathode and the work piece acting as the anode. A gas is introduced around the cathode and is allowed to flow through a narrow path towards the anode (Fig. 11.45). In a typical plasma torch a device to swirl the gas is provided. The "swirler" which may be ceramic, encircles the lower portion of the electrode that stabilizes the gas flow and prevents gas turbulence [SME, 1998]. In the narrow path the temperature of the gas rises close to 28,000°C and the plasma transfers the heat rapidly to the work piece and the material gets melted and vaporised. The plasma torch is made of copper and water cooled. The primary gas such as nitrogen or argon is forced through the nozzle to get it ionized while the secondary gases or water flow are often used to help clean the kerf of molten metal during cutting.

The secondary gases used also called as shielding gases depend upon the material being machined. Hydrogen is often used as a shielding gas for stainless steel, aluminium, and other nonferrous metals. Carbon dioxide is also used for ferrous and nonferrous materials, while air or oxygen can be used with mild steel. In place of the shielding gas if water is used, it is noticed that the quality of the cut is improved in addition to the beneficial effect of cooling the torch.

Only a small portion of the energy is actually utilised in PAM for material removal. In a typical PAM cutting or machining operation, up to 45% of the electrical power delivered to the torch is used to remove metal from the work piece [SME, 1998]. Rest of the power is wasted in the form of heating the cool water in the plasma generator, the work piece and the gas.

Typical parameters that are used with plasma arc machining are [El-Hofy, 2005]:

- Velocity of plasma jet – 500 m/s
- Material removal rate – 150 cm^3/min
- Specific energy – 100 W/cm^3.min
- Power range – 2 to 200 kW
- Voltage – 30 to 250 V
- Current – up to 600 A
- Cutting speed – 0.1 to 7.5 m/min
- Maximum plate thickness – 200 mm

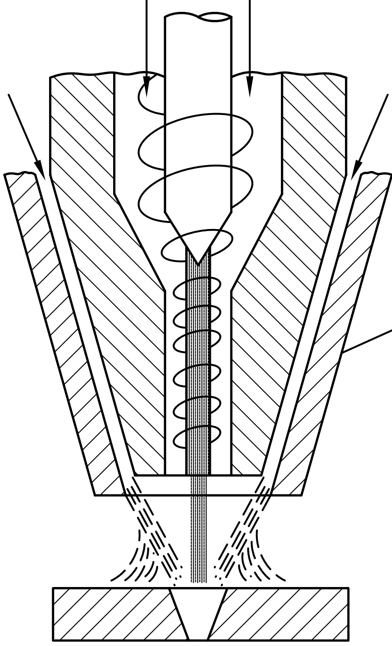


Fig. 11.45 *Schematic of plasma arc machining set up*

Applications

The major application of PAM is in cutting. It can cut any electrically conductive metals. Coupled with a CNC table, PAM can be used to cut any shapes such as straight line, circle, or complex profiles. Often PAM is combined with a CNC turret punching press (Amada Coma 567 turret press with plasma cutting) for wide ranging fabrication jobs.

It can also be used in an operation similar to turning where the plasma torch is applied to a rotating cylindrical surface. The torch is kept tangentially in contact with a rotating cylindrical surface. This is normally used with difficult to turn machines since PAM will not have any advantage in easy-to-machine materials such as steel, aluminium or copper alloys. So it is normally used for difficult-to-machine materials such as Inconel, stainless steel and hastelloys.

Advantages

- It is equally effective on any electrically conductive material regardless of its hardness or refractoriness.
- It does not directly contact the work piece thereby simplifying the setup process.

- It does not require any special surface preparations or use any dangerous chemicals for cleaning after machining.

Disadvantage

- The process because of the intense heat used allows for a large heat affected zone which needs to be removed later using other operations.

SUMMARY

Unconventional machining processes were developed to machine hard and heat resistant materials. These processes are based on various principles that are different from the conventional processes, which rely on the hardness of the cutting tool material.

- There are a number of processes such as EDM, ECM, USM, etc. each of which rely on different principle to displace material from the work piece. These processes have a lot of differences that make each of them applicable only for specific applications.
- Electric discharge machining (EDM) removes metal by the heat generated by the spark generated between the electrode and the work piece. Electrode materials include copper, graphite and tungsten, though graphite is the most commonly used electrode material. Dielectric fluid needs to be applied to continuously flush out the debris generated during the material removal process. Most widely used process to machine complex dies and moulds used for die casting and injection moulding.
- Electrochemical machining (ECM) utilises the principle of anodic dissolution of material to remove the material from the work piece. The accuracy of the work piece achieved is excellent with very little wear of the electrode. Electrolyte flow between the electrode and work piece is very critical since it completes the electrical circuit while removing the heat generated and products of reaction away from the machining zone.
- Ultrasonic machining (USM) removes material by abrading a brittle material by the abrasive in a slurry using the ultrasonic vibrations. It is a simple equipment however the tool wear is high and the material removal rates are low.
- Chemical Machining (CHM) is essentially the removal of material by dissolving in a chemical solution.
- Laser beam machining (LBM) uses the focused energy of the light beam to partially melt and blow away the material to cut at very precise locations. Used predominantly to make holes and cut complex contours.
- Abrasive water jet machining (AWJM) utilises very high speed jet of water mixed with abrasive to cut any type of material. This is being increasingly utilised in the industry since it can cut any material and also will not affect the work piece material in any way.
- Electron beam machining (EBM) uses the high-energy electron beam to melt and vaporize the metal in its path to remove metal. This is used for small holes with extremely high aspect ratio.
- Ion beam machining (IBM) removes metal by the sputtering process where the atoms are displaced by the collision of ions. This is used specifically for small parts without using any harmful chemicals.
- Plasma arc machining (PAM) uses the ionised gas plasma to melt the material at a very fast rate. This is generally used for cutting of very thick metal plates.

Questions

11.1 Explain the need for the use of unconventional machining processes compared to the conventional ones.

11.2 Give a comparison of the unconventional processes in terms of process, material removal rate and applications.

11.3 Explain the reasons why the unconventional machining processes are used.

11.4 Explain the principle of EDM with a neat sketch.

11.5 Explain why the tool shape in EDM should be complimentary to the final form.

11.6 Draw a typical relaxation circuit used for the EDM power supply and derive the expression for the material removal rate.

11.7 Explain the disadvantages of relaxation circuit and show the alternative arrangement of pulse generator used in EDM.

11.8 Briefly explain the working of an EDM machine showing important elements.

11.9 What are the functions served by the dielectric fluid in EDM?

11.10 Briefly explain the flushing techniques used in EDM giving their relative merits and applications.

11.11 What are the characteristics required for a good electrode material in EDM?

11.12 Explain the application of the following electrode materials in EDM:
(a) Copper (b) Graphite

11.13 What are the important parameters that control the material removal rate in EDM? Briefly explain any two factors.

11.14 What possible errors are caused in EDM?

11.15 Explain the advantages and disadvantages of EDM.

11.16 Write a short note on wire EDM process.

11.17 Explain the principle of ECM with a neat sketch.

11.18 What are the principal features of ECM process?

11.19 Briefly explain the working of an ECM machine showing important elements.

11.20 What are the functions served by the electrolyte in ECM?

11.21 Describe the factors that should be considered in selecting an electrolyte in ECM.

11.22 Briefly explain the electrolyte flow methods used in ECM giving their relative merits and applications.

11.23 What factors should be considered in selecting the tool materials in ECM?

11.24 Briefly explain the various process parameters that affect the material removal rate and surface quality in ECM.

11.25 Explain the advantages and disadvantages of ECM.

11.26 Write a short note on ECG.

11.27 Explain how material is removed in USM.

11.28 Briefly explain about the functions of transducer and tool cone in USM.

11.29 What is the function of abrasive slurry in USM? Explain how the abrasive selection is made.

11.30 Briefly explain the working of an USM machine showing important elements.

11.31 Explain the advantages and disadvantages of USM.

11.32 Briefly explain the steps involved in chemical machining.

11.33 Explain the various methods used for preparing the masks for chemical machining.

11.34 Explain the process of PCM including the mask preparation process.

11.35 Give a short note on LBM.

11.36 What are the types of lasers that are generally used in LBM? Explain their significance.

11.37 Give a short note on Laser drilling.

11.38 Give a short note on Laser cutting.

11.39 A series of 5 mm holes (total number 6) are to be drilled on a circle of 150 mm diameter on a 6 mm sheet made of glass. Specify the method of manufacture to be used with a neat sketch of the setup. What are the variables and their effect in the process, which affect the final hole quality?

11.40 The T55Ni2Cr65Mo30 steel block is to be used for making a progressive die for punching the following component. Specify the process used for manufacturing the punch and die along with the limitations of the process. Is an alternative process available for its manufacture? Explain why it is not used.

11.41 Explain the advantages and disadvantages of AWJM.

11.42 Write a short note on AWJM process.

11.43 Explain the principle of AWJM with a neat sketch.

11.44 Describe the factors that should be considered in selecting the abrasive in AWJM.

11.45 Explain how material is removed in AWJM.

11.46 Briefly explain the equipment used for electron beam machining.

11.47 Give the applications of electron beam machining.

11.48 Explain the advantages and limitations of electron beam machining.

11.49 Briefly write about the process of ion beam machining.

11.50 Briefly explain the plasma arc machining process.

11.51 Give the applications of plasma arc machining.

11.52 Explain the advantages and limitations of plasma arc machining.

Multiple Choice Questions

11.1 Unconventional (non-traditional) machining processes are used specifically for

(a) Very high hardness of the work material

(b) Complex surfaces that cannot be easily obtained by conventional machining operations

(c) Difficult geometries that cannot be easily produced by conventional machining operations

(d) All of the above

11.2 The dielectric fluid in electric discharge machining (EDM) process should

(a) Ionise rapidly after the spark discharge has taken place.

(b) Have a high viscosity

(c) Be chemically neutral so as not to attack the electrode

(d) Have a low flash point.

11.3 The following is **not** a dielectric fluid to be used in electric discharge machining (EDM) process

(a) Silicone-based oils
(b) Linseed oil
(c) Kerosene
(d) De-ionised water

11.4 What are the problems faced when the electrolyte is not properly flushed in electric discharge machining (EDM) process?
(a) Uneven surface finish
(b) Higher removal rate
(c) Stable arc conditions
(d) None of the above

11.5 Required characteristics of good electrode materials in electric discharge machining (EDM) process
(a) A good conductor of electricity and heat.
(b) Produce efficient material removal rates from the work pieces.
(c) Be easily machinable to any shape at a reasonable cost.
(d) All of the above

11.6 The material that is **not** a very good electrode in electric discharge machining (EDM) process
(a) Graphite
(b) Copper
(c) Gray cast iron
(d) Tungsten

11.7 Taper of hole produced in electric discharge machining (EDM) process is caused by
(a) Inaccuracy in the electrode size
(b) The side sparks between the tool and the machined surface produced
(c) Larger flow of current between the bottom of the electrode and the work piece surface
(d) None of the above

11.8 Improved surface finish of electric discharge machined (EDM) surfaces can be obtained by
(a) Increasing the current in the sparks
(b) Decreasing the frequency of sparks
(c) Increasing the frequency of sparks
(d) None of the above

11.9 Material removal rate in electric discharge machined (EDM) process can be increased by
(a) Decreasing the current in the sparks
(b) Decreasing the frequency of sparks
(c) Increasing the frequency of sparks
(d) None of the above

11.10 Function of the electrolyte used in electro chemical machining (ECM) process
(a) Completes the electrical circuit between the tool and the work piece
(b) Allow desirable machining reactions to takes place
(c) Carry away heat generated during the operation
(d) All of the above

11.11 The electrolyte used in electro chemical machining (ECM) process should have the following property
(a) Chemical stability
(b) Low viscosity and high specific heat
(c) High electrical conductivity
(d) All of the above

11.12 The non-conventional machining process that gives the highest material removal rate
(a) Electric Discharge Machining (EDM)
(b) Electro Chemical Machining (ECM)
(c) Ultrasonic Machining (USM)
(d) Chemical machining (CHM)

11.13 The non-conventional machining process that gives the best surface finish
(a) Electric Discharge Machining (EDM)
(b) Electro Chemical Machining (ECM)
(c) Ultrasonic Machining (USM)
(d) Chemical machining (CHM)

11.14 Increasing the feed rate in electro chemical machining (ECM) process results in
(a) High removal rate and improved surface finish
(b) High removal rate and decreased surface finish
(c) High removal rate and surface finish not affected
(d) None of the above

11.15 Better surface finish in electro chemical machining (ECM) process can be obtained by
  (a) Low electrolyte concentration
  (b) Low electrolyte temperature
  (c) Low voltage
  (d) All of the above

11.16 Limitation of the electro chemical machining (ECM) process
  (a) Use of corrosive media as electrolytes makes it difficult to handle
  (b) Poor surface finish
  (c) Poor accuracy of the work piece dimensions because of the large tool wear
  (d) There will be thermal damage to the work piece

11.17 The following process is suitable for machining brittle materials such as glass
  (a) Electric Discharge Machining (EDM)
  (b) Electro Chemical Machining (ECM)
  (c) Ultrasonic Machining (USM)
  (d) Chemical machining (CHM)

11.18 Limitation of the Ultrasonic Machining (USM) process is
  (a) Metal removal rates are large
  (b) Tool wear is high and sharp corners cannot be produced
  (c) It is affected by the electrical or chemical characteristics of the work material
  (d) Cannot be used with brittle materials

11.19 Advantage of laser beam machining (LBM) is
  (a) Laser beam machining does not apply any direct force because it is a non-contact machining
  (b) Laser beam machining can be localized to a small area thereby removing a very small amount of material
  (c) Since the heat is localized, Heat Affected Zone (HAZ) in laser beam machining is small
  (d) All of the above

11.20 Increasing the feed rate in abrasive water jet machining (AWJM)
  (a) Improves the surface finish
  (b) Decreases the drag
  (c) Surface finish gets deteriorated
  (d) None of the above

11.21 The machining process that will be most appropriate to drill a rectangular hole in a high strength alloy
  (a) Drilling
  (b) Ultrasonic Machining (USM)
  (c) Electric Discharge Machining (EDM)
  (d) Chemical machining (CHM)

11.22 The machining process that will be most appropriate to drill a rectangular hole in a ceramic material
  (a) Drilling
  (b) Ultrasonic Machining (USM)
  (c) Electric Discharge Machining (EDM)
  (d) Chemical machining (CHM)

11.23 The machining process that will be most appropriate to machine a turbine blade with an aerofoil cross section in a high strength material
  (a) Electro Chemical Machining (ECM)
  (b) Ultrasonic Machining (USM)
  (c) Electric Discharge Machining (EDM)
  (d) Chemical machining (CHM)

Answers to MCQs

11.1 (d)	11.2 (c)	11.3 (b)	11.4 (a)	11.5 (d)
11.6 (c)	11.7 (b)	11.8 (c)	11.9 (c)	11.10 (d)
11.11 (d)	11.12 (b)	11.13 (b)	11.14 (a)	11.15 (d)
11.16 (a)	11.17 (c)	11.18 (b)	11.19 (d)	11.20 (c)
11.21 (c)	11.22 (b)	11.23 (a)		

Case Study

TURBINE BLADE MACHINING

Engine components for airplanes which need to maintain low weight, high temperature resistance and increased thermal efficiency require that these be manufactured with high nickel and titanium alloys. The commonly used materials are Ti-6Al-4V and Inconel 718. The geometry of turbine blade is complex and these blades are arranged on the rotor disk with specialized geometry that facilitates the flow of gases. The gap between the blades is relatively small and has complex geometry, the machining of which requires a very careful planning. Being hard materials these alloys can be machined using conventional milling, Electro Discharge Machining (EDM) and Electro Chemical Machining (ECM) processes.

In this case study, these alloys were machined using conventional milling, EDM and ECM processes in order to compare the material removal rates and economics. The conventional milling is done with trochoidal milling which is a method of machining used to create a slot wider than the milling cutter diameter. This is accomplished by moving the cutter through a series of circular cuts known as a trochoidal tool path. This provides a low radial depth of cut and a high axial depth of cut and achieves good material removal rate. In the EDM setup, initial roughing was done using the regular EDM process and then Wire EDM was used. The table below shows the variation of MRR among the processes.

Manufacturing Process Used		Material Removal Rate of Ti–6Al–4V (mm^3/min)	Material Removal Rate of Inconel 718 (mm^3/min)
Conventional Milling		6035	3401
EDM	EDM (Roughing)	220	500
	Wire EDM (Finishing)	3090	2388
ECM		4838	4163

Economic analysis for titanium alloy machining led to the inference that with low tool development costs, ECM and milling costs are in similar range and not much change is observed with changes in the batch sizes. However, EDM cost increases with the production volume. Therefore, except for small batch sizes of 20 or less, EDM will be uneconomical.

1. Apparently, ECM works better than other processes in Inconel alloy, but not for Titanium alloy. Find out the possible reasons based on the governing mechanics.
2. What are the possible reasons for cost effectiveness of ECM compared to EDM?

CHAPTER 12

Micro-Manufacturing

Objectives

Miniaturization is taking place at a rapid rate and this chapter provides a summary of some of the micro-manufacturing methods that are employed for this purpose. After completing this chapter, the reader will be able to

- Understand the need for micro-manufacturing
- Recognize the classification of different micro-manufacturing methods practiced
- Study different traditional micro-manufacturing processes
- Study different nontraditional micro-manufacturing processes
- Understand different steps in semiconductor manufacturing methods used in MEMS and nano applications.
- Learn a case study in the processing of a MEMS device

12.1 INTRODUCTION

Technology is getting miniaturized at a relatively fast pace. This requirement is dictated by the need for enhancing the functionality and at the same time reducing the dimensions. At one point of time not long ago the wristwatch parts were the only industrial microproducts manufactured. But the desire is to have ever smaller parts for all the electronic gadgets that are utilized today such as cell phones, medical implants, fuel injection nozzle for automobiles, micro surgical tools, micro end effectors and semiconductor devices. Many of these products also need extremely fine finish in the range of nanometer level with complex geometries. The quality and reliability of the products that are demanded today is phenomenally high, and this requires a special class of manufacturing methods. A few examples are the microfluidic devices, drug delivery systems, diagnostic devices, micro pumps, micro engines, inkjet printer printing heads (cartridges), etc.

Achieving micro dimensions such as those used in the MEMS (Micro Electro Mechanical Systems) devices in the range of 1 to 999 μm require a very special class of processes that are different from the traditional machining processes that are normally used. Fig. 12.1 shows the progress of machining technologies over the years along with change in accuracy limits. In Fig. 12.1 on the right side some of the machining technologies corresponding to the accuracy requirements can be seen. It may be noticed that at this point in time the ability to manipulate at the atomic level is possible along with the corresponding accuracy requirements for working models of parts either mechanical or biological.

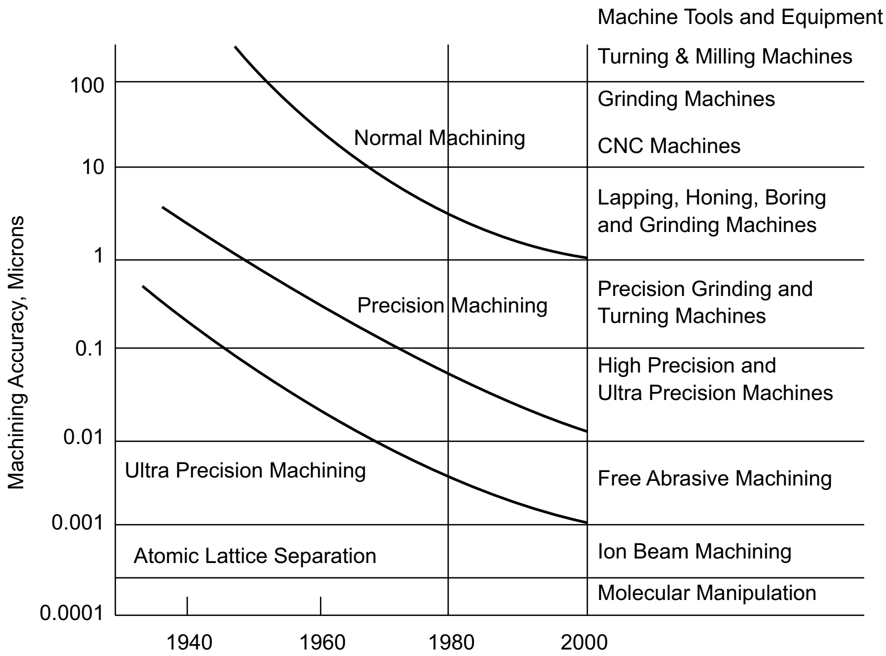


Fig 12.1 *The progress of machining technologies over the years along with change in accuracy limits*

An example of micromachining provided here is that of a micro-compressor. It is a micro-compressor machined on the tip of a 1-mm diameter cylinder. The geometry is two-dimensional with the third dimension being straight with no taper. The compressor is made of tungsten carbide while the electrode is of stainless steel. The centre hole of diameter 300 μm is micro-EDM drilled using a stock electrode. The blades are micro-EDM milled with the same tool after it was reduced to a diameter of 40 μm by wire EDG. The incremental depth for each micro-EDM milling layer is 0.5 μm. All the blades show good consistency with good shape accuracy and errors less than 1 μm. The path of the cutting tool is smooth with no path marks as well as no taper angle. The only error noticed is the eccentricity error of about 20 μm as measured. The estimated surface roughness is about 0.08 μm.

12.2 OVERALL CLASSIFICATION

A number of authors have classified the micro-manufacturing processes based on technologies and working principles. A simplified classification of the micro-manufacturing processes based on the manufacturing approaches is shown in Fig. 12.2. A brief description of some of these processes is presented in this chapter in terms of their processing technologies and applications.

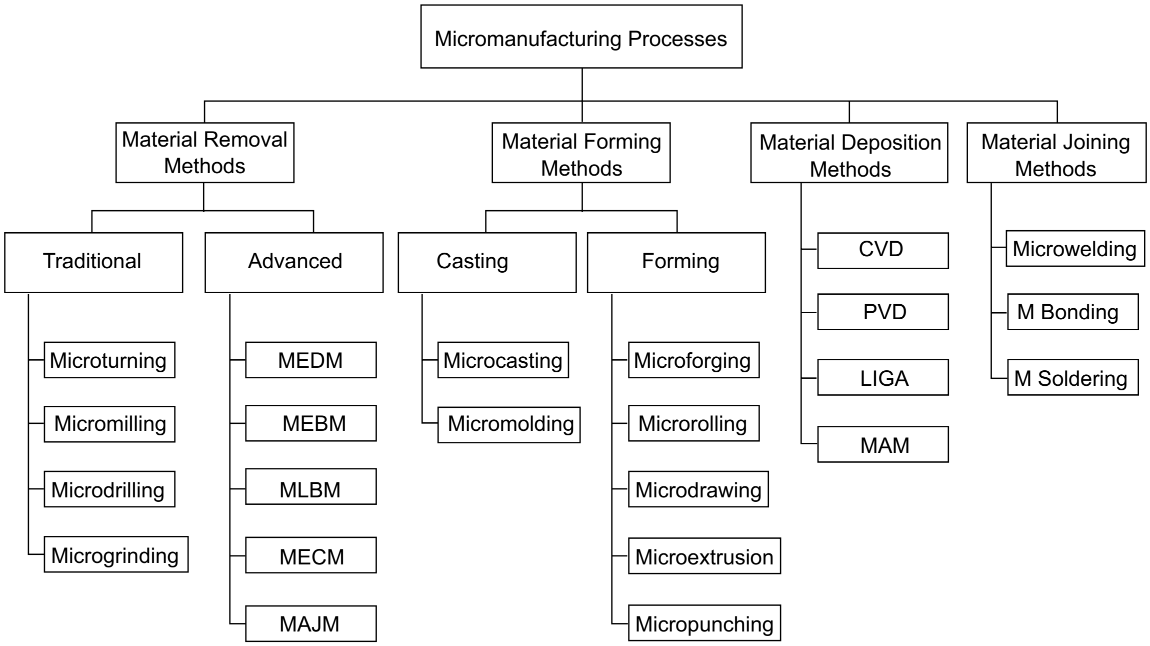


Fig 12.2 *Classification of micro-manufacturing methods based on the manufacturing principle. MEDM – Micro-EDM; MEBM- Micro EBM; MLBM- Micro-LBM; MECM-Micro-ECM; MAJM-Micro-AJM; CVD-Chemical Vapor deposition; PVD-Physical Vapor deposition; LIGA- Lithographie Galvanoformung Abformung (Lithography, Electroplating and Molding); MAM- Micro Additive Manufacturing methods; Adopted from [3].*

12.3 MATERIAL REMOVAL METHODS

Material removal processes are going through continuous improvements particularly with the availability of computers, and improving on the rigidity of the machine tools and the quality and reliability of the cutting tools. As a result, the accuracies that could be achieved with these processes are improving continuously. Miniaturization of the machine tools would allow for the manufacture of parts in smaller sizes that are required for the current generation of microproducts. The traditional processes have been miniaturized by making them small compared to the conventional machine tools in order to achieve the required accuracy. It is necessary that these machines provide ultra-precision and use tools that are very precise and rigid so that they minimize the machining errors to the minimum possible level. The micro-machine tool should be built with high precision drive systems, having rigid spindle with closed loop control. The bed should preferably be free of any thermal deflection to maintain the highest amount of accuracy required. It should preferably be a computer controlled machine tool to avoid any operator related errors. The basic processes can be broadly categorized as micro-turning, micro-milling and micro-drilling similar to the traditional processes.

12.3.1 Conventional Material Removal Processes

These processes rely on the force exerted by the cutting tool tip to plastically remove the material. One of the problems with the mechanical material removal processes is the resulting burrs that need to be removed by some other process. Since the micro parts are so small that it is almost impossible to manually clean the burrs. Therefore it is necessary to select the process parameters that would ensure that burr is not formed during the processing.

Micro-turning

Micro-turning is the process used for axi-symmetric parts that are small in size requiring high precision. The machine tool used should therefore be similar to a production lathe with miniaturized dimensions and high precision as explained earlier. Diamond tipped tools are used for precise turning. Micro-turning should be able to achieve accuracies of the order of 1 µm or less with surface finishes in the range of 0.002 to 0.005 µm R_a. Tools typically used are natural and synthetic diamond tips. Typical applications of micro-turning are small diameter shaft like components, optical lenses, molds for contact lenses, intraocular lenses, etc.

Micro-milling

Micro-milling is a versatile process that can be used for many different geometries. The diamond tool used for this purpose with the cutting point located away from the spindle in a cantilever form is a weak link. Monocrystalline diamond is the most suitable tool material. Two fluted end mills made from tungsten carbide powder (0.3 µm grain size) up to 100 µm diameter are commercially available with an edge radius of the order of 1–2 µm. Micro-milling similar to the conventional milling is used for prismatic shapes such as grooves, cavities, and 3D concave and convex shapes in miniaturized components. Micro-milling is also used for making structured molds that are used for micro-molding of structured optical elements such as Fresnel lenses.

Micro-drilling

Micro-drilling uses small drills similar to a spade drill and utilizes peck drilling cycle for efficient removal of the chips. Larger micro-drills can be similar to twist drills. The tool material used is micro-grain tungsten carbide. Micro-drilling being a mechanical process will not be affected by the electrical properties of the work piece similar to micro-EDM. Therefore, all metals, nonmetals, plastics and composites, can be machined easily. One typical example is the drilling of holes in laminated circuit boards. Very hard or brittle materials are difficult to machine.

Micro-grinding

Similar to the above processes micro-grinding is also a mechanical process that removes the material by means of mechanical force that is used for grinding very small work pieces that require very fine finish. Micro-grinding is the process that removes material using micro abrasive grains with very small depth of cut. As a result brittle materials can be mirror polished using this process. The grinding tool is generally in the form of a wheel consisting of the micro abrasive grains in a matrix. The grain depth of cut with this wheel has to be kept to less than 100 nm to obtain smooth surfaces of less than 10 nm peak to valley depth. The application of micro-grinding in the fabrication of 2D or 3D micro-cavities requires tools that are smaller than the cavity required. For this purpose a tool with a micro-sized tip is used. In such cases a micro grinding pin with the smallest diameter, typically 30 µm, tools will be used. Because of the considerable grinding force, deep micro-holes or deep, narrow cavities cannot be produced by micro-grinding.

12.3.2 Nontraditional Material Removal Processes

When the conventional material removal processes cannot serve the purpose of economically manufacturing the parts because of either material hardness or other requirements, the nontraditional processes would be used. These are also sometimes called unconventional micro manufacturing processes because of the material removal technology utilized.

Micro-EDM

As discussed in Chapter 11, Electro Discharge Machining (EDM) vaporizes the material to be removed by the intense heat generated by the electric spark between the electrode and the work material. As a result the shape of the electrode transfers to the work piece. The size of the tool used being small the corresponding detail produced in the work piece is also small. Typical tools are available with a tool diameter of 5 μm. The 3-dimensional control of EDM tool with CNC help to produce complex surfaces as well. Because the size of the tool is extremely small the arc need to be struck for a relatively lesser time to make sure that tool wear is minimized. Table 12.1 provides major differences between the conventional EDM and Micro-EDM. Micro-EDM can be further divided into Micro-EDM drilling, Micro-EDM die-sinking, Micro-EDM milling, Micro-wire EDM and Micro-wire EDG, depending upon the type of surface that needs to be obtained.

Table 12.1 **Operating parameters as used in conventional EDM vs Micro-EDM.**

Parameters	Conventional EDM	Micro-EDM
Size of the tool, mm	> 1.0	< 0.999
Inter electrode gap, μm	10 to 500	< 10
Open circuit voltage, V	4–400	10–120
Peak current, A	> 3	< 3
Pulse on-time, μs	500 to 8000	0.05 to 100

The preferred tool materials are tungsten carbide and copper. Electrically conductive CVD diamond is also used as tool material because of its almost zero electrode wear. Simple cylindrical tools are used most of the time. It is also possible to use hollow electrodes in drilling when the size is above 100 μm.

Micro-ECM

Electro chemical machining (ECM) is characterized by zero wear and as a result, the quality of the parts will be maintained over a complete batch and there is no need to change tools in between. It is also possible to remove higher amounts of material from the work. Since this process does not add stresses in any way to the parts, it is a process of choice for ultra-precision components used in avionics, biotechnology, etc. ECM is also adopted for machining micro-parts in the range of 5 to 500 μm with tolerances in the range of microns. Further impetus for adopting micro-ECM is the use of ultra-short pulses of nanosecond duration that help the process to be confined to very small locations down to nanometer precision, to achieve high resolution. A comparison of the operating parameters between the convention ECM and micro-ECM are given in Table 12.2.

TABLE 12.2 Operating parameters as used in conventional ECM vs Micro-ECM [Adopted from 12].

Parameters	Conventional ECM	Micro-ECM
Voltage, V	10–30	1–10
Current density, A/cm^2	20–200	75–100
Electrolyte flow, m/s	10–60	< 3
Inter electrode gap, µm	100 to 600	5–50
Machining rate, µm/min	200–10 000	5
Accuracy, µm	±20–100	±10

In view of the many advantages micro-ECM is the process of choice for the micro-electronic devices. It has been used for machining inkjet nozzle plates, conducting lines in PC boards, high accuracy holes, etc. The process variations are micro-hole drilling, micro-ECM using masks that are used in semiconductor manufacturing that is covered separately in 12.3.3, micro-electrochemical milling, wire electrochemical machining, laser electrochemical micro machining (LECMM), etc.

12.3.3 Semiconductor Manufacturing Processes

Single crystal silicon has been the material of choice for all semiconductor manufacturing processes from the very beginning. Silicon is the most abundant material in the earth's crust (27.8%) as an ore in the form of quartzite. Very effective extraction and purification methods of silicon from its raw material as well as the processes for single crystal formation from liquid silicon are available. Normal silicon is a very brittle material that breaks easily. The reason is that in the macro sizes, imperfections in the material are very rapidly accumulated into micro cracks, and this makes the material very brittle. In the micro domain these imperfections are reduced and almost eliminated giving it the necessary strength and elasticity. It has the same Young's modulus as steel (about 2×10^5 MPa) but as light as aluminium with a mass density about 2.3 g/cm^3. It has a high melting temperature of about 1400°C. For example silicon beams 2 µm × 2 µm × 100 µm are widely used with very large deflections in micro devices. Its thermal expansion coefficient is 8 times less than steel and shows no hysteresis to make it an ideal material for sensors and actuators in the micro domain.

It is mechanically stable and can be doped with suitable impurities to alter resistivity. It can be doped with p-type (boron) and n-type (phosphorous, antimony, arsenic) to affect their conducting characteristics. Silicon dioxide is used also as an insulator and passivation layer. Silicon has efficient response to solar radiation and light. Silicon has a relatively high dielectric strength and therefore is suitable for power devices. Silicon is used in the following form in MEMS devices:

- Single crystal silicon
  - Anisotropic crystal
  - Semiconductor, great heat conductor
- Polycrystalline silicon – polysilicon
  - Mostly isotropic material
  - Semiconductor
- Silicon dioxide – SiO_2
  - Excellent thermal and electrical insulator
- Silicon nitride – Si_3N_4
  - Excellent electrical insulator
- Semiconductor
  - Electrical conductivity varies over ~8 orders of magnitude depending on impurity concentration

- N-type and P-type dopants both give linear conduction, but from fundamentally different mechanisms
- N-type touching P-type forms a diode

Semiconductor manufacturing unfortunately is not a single process, but involves a number of processes depending upon the final result to be achieved. The basic steps to form microstructures are:

- Deposition (the additive process)
- Etching (the subtractive process)
- Materials modification (to tailor electrical chemical, or physical properties)
- Lithography (providing the pattern)

These processes are applied in any order and any number of times depending upon the final design that is required. Some of the steps may be skipped or repeated a number of times. Lithography is the most important step that controls where the materials stay or where they need to be removed by providing a patterned mask.

Typical sequence of processing steps in a photolithography process are shown in Fig. 12.3. The first step is getting the appropriate wafer that will act as the substrate. This then needs to be added with a layer of photoresist (in the form of liquid) for the purpose of transferring the required image of the surface. The coat thus obtained is then baked so that it can move to the next step for transferring the design. The required design in a transparent sheet is aligned with the wafer ultraviolet light is used to transfer the image to the photoresist. The latent image in the photoresist will be developed in the next stage so that subsequent operations can be carried out. The developed image provides areas that will be masked from UV light penetration so that the material under them will not be affected during the next stage of etching. Next step is etching where chemical solutions will be utilized to remove the metal that is not masked to get the necessary structure of details. The last operation will be to clean and remove the mask (photoresist left) to complete the operation.

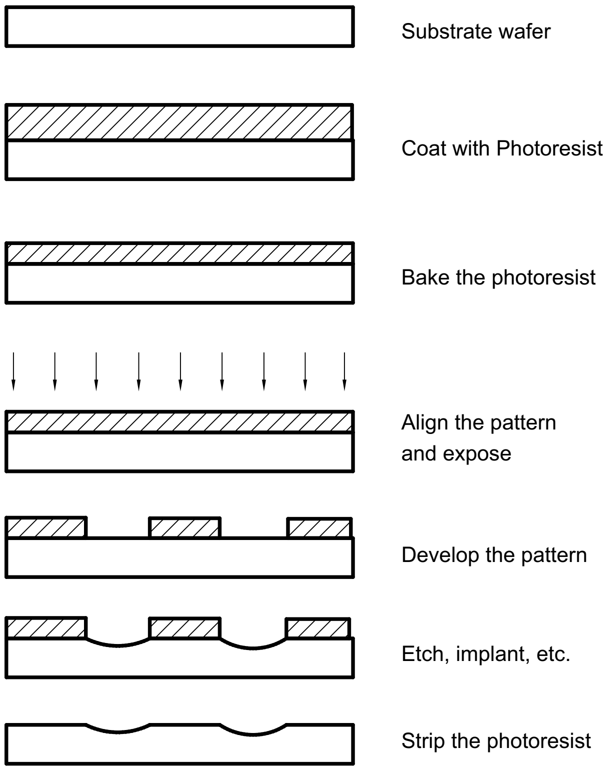


Fig 12.3 *Steps in lithography.*

Silicon Wafer

The starting point of any MEMS or semiconductor process is the single crystal silicon wafer. More than 75% of all single crystal silicon wafers grow via the Czochralski (CZ) method. In this method silicon is heated above 1420°C in a quartz crucible. The required dopant combinations are added to the liquid and once they are liquefied a seed crystal that has the same crystal orientation required in the finished ingot is brought to the top of the melt surface. In order to achieve the uniformity in doping of the silicon boule, the seed crystal and the quartz crucible rotate in opposite directions. Once the system reaches proper conditions for crystal growth, the seed crystal slowly lifts out of the melt. The seed crystal is rapidly pulled to continue the growth of the crystal. This minimizes the number of crystal defects within the seed at the beginning of the growing process. Later the speed is reduced to allow the diameter of the crystal to increase. When the desired diameter of the crystal is reached, the growth conditions are stabilized to maintain the diameter. As the seed is slowly moving above the melt, the surface cools, with the atoms in the melted silicon orienting themselves to the crystal structure of the seed. Growing a silicon ingot can take anywhere from one week to one month, depending on many factors, including size, quality and the specification.

Table 12.3 Standard sizes of silicon wafers

Diameter, mm	Thickness, µm	Weight, g
150	675 ± 20	28.00
200	725 ± 20	53.08
300	775 ± 20	127.64
400	825 ± 20	241.56

The standard size of a monocrystalline silicon wafers are given in Table 12.3. The ingot is ground to the rough final diameter and a flat is machined to indicate the type of wafer. The ingot is then sliced with a diamond edge saw and inspected for quality. The wafers are now lapped to remove any imperfections on the surface because of the slicing process. Then the surface is etched and cleaned to remove any microscopic cracks and/or surface damage that may have come about during lapping process. Then the wafer is polished in a clean room with a mirror finish. This polished surface is used for device fabrication and must be free of topography, micro-cracks, scratches, and residual work damage. Now the wafer is ready as raw material for any intended work.

Material Modification and Deposition

The first step in any device fabrication is either material growth or deposition that create the layers for micro device fabrication. Common processes for this are:

Growth by chemical reaction (e.g., oxidation) The silicon wafers are heated and exposed to ultra-pure oxygen in the diffusion furnaces at 800–1200ºC under carefully controlled conditions forming a silicon dioxide film of uniform thickness on the surface of the wafer. Silicon is consumed as the silicon dioxide grows. The film thickness up to a maximum of 2 µm can be grown. Oxidation can be masked with silicon nitride, which prevents O_2 diffusion.

Physical application (e.g., spinning layers onto a substrate) There are many types of physical application processes available such as dipping, spraying, and spin-on. Photoresist layer (mask) is generally deposited in this method using spin-on. The procedure is that the wafer spins at 1000-5000 RPM for a specified time (~30s) after a viscous liquid is poured on center of the wafer for the given thickness control of the liquid film. Then it is baked on a hotplate at about 80–500°C for 10–1000s (volume reduction by ½). Due to the high thermal conductivity of silicon, the photoresist is heated to the hot plate temperature quickly (in about 5 seconds for hard contact). When the wafer is removed from the hotplate, baking continues as long as the wafer is hot. After cooling, the wafer is ready for its lithographic exposure.

Physical vapor deposition PVD is fundamentally a vacuum coating technique that can be done in two ways. In one method the metal is vaporized to a plasma of atoms or molecules in a vacuum and deposited on the silicon wafer. The wafer is secured in a fixture and placed in the vacuum deposition equipment chamber. The metal to be coated such as aluminum or gold is kept in a tungsten crucible and is evaporated by a high intense heat source. The evaporated metal then coats the wafer which is directly above, so that the metal coating is produced simply by line of sight.

PVD has the advantage that it does not require any gases or other chemicals for operation and provides an even coating on flat surfaces. However, since the process depends on vapor pressure, low vapor pressure species will require a very high vacuum environment. Also alloys tend to be difficult to deposit as different metals have different vapor pressures. High aspect ratio features such as sidewalls will not be evenly deposited. It is still widely used for optical coatings and other surface processes.

Another method of depositing is sputtering. Sputtering is a plasma-assisted technique that creates a vapor from the source target through bombardment with accelerated gaseous ions (typically Argon). Argon plasma sputters atoms off target (coating material), takes ballistic path to wafers and deposits on the wafer. It requires high vacuum depending on material. The wafer substrate is spun to achieve uniform thickness.

Chemical vapor deposition CVD process requires precursor gas or gases to flow into a chamber containing the heated wafer to be coated. Gases dissociate on surfaces at high temperature. Chemical reactions occur on the hot surfaces, resulting in the deposition of a thin film on the surface. This is accompanied by the production of chemical by-products that are exhausted out of the chamber along with unreacted precursor gases. The main problem is the use of dangerous gases such as silane, arsine, phosphine, etc. It is typically done at low pressure (LPCVD) rather than atmospheric (APCVD). In LPCVD pressures are around 300 mTorr (0.0004 bar).

Lithography

Lithography is the process that uses light or other forms of radiant energy to change the chemical properties of thin layers of films that have been deposited on the wafer that acts as the substrate. When the pattern is transferred using lithography then the latent image of the shape of the object to be obtained will be embedded into the coating. This image then will be developed and can be used as a mask for selective removal of the material from the substrate. The terms used in lithography are:

Lithography – The transferring (writing) of a pattern-usually to a “resist”

Resist – Medium into which pattern on a mask, on a mold, or in computer file is transferred. Used in most types of lithography

Developer – Needed in some types of lithography to bring out the pattern written in the resist

There are a number of lithography types that are employed in semiconductor manufacturing depending upon the resolution required. Some of them are:

- Photo Lithography
- Electron Beam Lithography
- Ion Beam Lithography
- Dip Pen Lithography

- Embossing Lithography
- Stamp Lithography
- Molding Lithography

Photo-lithography In this book only photolithography is discussed, as the others are beyond the scope of this book. The basic principle behind the operation of a photoresist is the change in solubility of the photoresist in a developer upon exposure to light (or other types of exposing radiation). Photolithography uses light energy passing through a patterned mask that is already deposited on the photoresist with a silicon substrate. The light is focused onto the photosensitive surface through a photomask typically made of quartz with a chrome plating that controls where the radiant energy will strike the photoresist. Photomasks can be made with electron beam patterning tools to create fine features. There are several ways to accomplish this:

- *Contact/proximity printing:* Contact and proximity lithography are the simplest methods and offer high resolution (down to about the wavelength of the radiation), but practical problems such as mask damage and resulting low yield make this process unusable in most production environments.
- *Projection printing:* An optical system focuses the light source and reduces the mask image for exposure on the surface. It provides higher resolution and the lens system reduces diffraction error. The disadvantages could be errors due to the focus of lens system , and the limiting factor in resolution can be due to the optical system used.
- *Projection scanning*: Used for higher resolution.

Lithography opens up certain areas on a silicon wafer surface for further processing. When the substrate is exposed to radiation through a mask, properties of the resist changes chemically. Subsequent development removes the exposed resist, leaving these regions defined, but not permanently changed. The etch process removes the species from the exposed areas, permanently changing the makeup of those areas.

Etching

Etching is a process to remove material chemically or by some other process. Any etch process is characterized by certain properties:

The *etch rate*, often given in angstroms/minute, indicates how quickly the process proceeds. In determining throughput, this is a key indicator. It gives the amount of material removed from the wafer over a defined period of time.

Uniformity, the evenness of the removal over the entire surface of the wafer. A non-uniform etch could over etch certain areas of the wafer, removing protective masking or more of a layer than necessary, while leaving other areas correctly processed.

Profile could be isotropic in which etching proceeds at equal rates in both horizontal and vertical directions Fig. 12.4(a) or anisotropic where etching proceeds faster in one plane than in another Fig. 12.4(b).

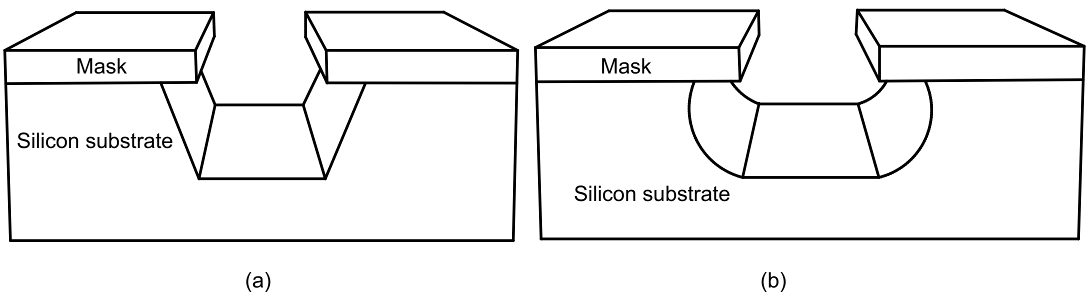


Fig. 12.4 *Anisotropy in etching*

Selectivity is the ability of the etch process to distinguish between the layer to be etched and the material not to be etched.

Etching can be carried out using one of the following processes:

- Wet chemical etching
- Dry physical etching
- Dry chemical/physical etching or Reactive ion etching (RIE)

Wet chemical etching The process of using liquid chemicals or etchants to remove material from the wafer is termed as wet etching. The actual pattern to be obtained is already present on the wafer in the form of mask. Materials that are not protected by the masks are etched away by liquid chemicals. Some of the wet etching agents for silicon are potassium hydroxide (KOH), ethylene diamine pyrocatechol (EDP), or tetra methyl ammonium hydroxide (TMAH). Because of the nature of silicon crystal planes the etching with these etchants will be anisotropic as shown in Fig. 12.8 (a). To get isotropic wet etching, a mixture of hydrofluoric acid, nitric acid, and acetic acid (HNA) is the most common etchant solvent used for silicon. The concentrations of each etchant determines the etch rate. Silicon dioxide or silicon nitride is usually used as a masking material against HNA.

Dry physical etching In dry physical etching plasmas or etchant gases are used to remove the substrate material. Etching properties of a plasma depend on the gas or gases it is made from, and the power pumped into the plasma. In dry physical etching, the high energy particles knock out the atoms from the substrate surface, and the material evaporates after leaving the substrate. It will be anisotropic because ions move from plasma toward substrate and therefore tend to bombard normal to the overall surface, knocking everything off.

Dry chemical/physical etching or Reactive ion etching (RIE) It is a mixture of chemical and physical etching using plasmas. Argon is used in a plasma as a physical etching process. The anisotropy of this process is retained, and the physical removal process causes the surface to continually be exposed to the chemical etching taking place from the radicals in the gas that have been formed from the plasma. Etching properties are tailored by adjusting relative importance of bombardment (physical) etching and chemical etching (from sources such as radicals). Degree of anisotropy can be adjusted by adjusting gas composition and power into plasma.

SUMMARY

In this chapter an overview of the micro-manufacturing methods is presented that are commonly used for making small, precision part making. The developments have progressed gradually with the developments in the miniaturization of electronics. These methods can be classified, similar to the traditional manufacturing processes, into material forming, material removal, material joining and material addition methods. In each category there are a variety of processes developed for specific applications. By miniaturizing the traditional processes it is possible to achieve the nanometer surface finishes by mechanical removal processes. In a similar way the micro nontraditional removal processes are able to produce micro and nanometer sized functional parts that are similar to their macro parts. Semiconductor manufacturing is essentially the non-traditional material removal operation for a special class of materials that are augmented with many processes specific to the type of parts that are mass produced. The operation involves a number of processes that need to be completed in sequence in order to achieve the required functionality. Fortunately the processes are well developed and could be used with certainty to achieve the mass manufacturing with identical functionality. Micro-pressure sensor manufacturing demonstrates the sequence of operations in semiconductor manufacturing methods as a case study.

Questions

12.1 Explain the need for micro-manufacturing in the modern scenario.
12.2 Give some examples of products that require micro-manufacturing.
12.3 Give a classification of the micro-manufacturing methods based on their manufacturing approaches.
12.4 Give the capabilities and applications of micro turning method.
12.5 Give the applications of the following processes: micro-milling, micro-drilling and micro-grinding.
12.6 Write a short note on micro EDM process with its applications.
12.7 Compare micro-EDM and micro-ECM processes from the standpoint of process and applications.
12.8 Why is silicon the preferred choice for the semiconductor manufacturing?
12.9 Give a list of all the materials that are used in semiconductor manufacturing.
12.10 What are the basic steps used in semiconductor manufacturing? Give a brief description of each of them.
12.11 Write a short note on material modification as related to semiconductor manufacturing.
12.12 What is the purpose of lithography in semiconductor manufacturing? Give the names of different methods that are used.
12.13 Give some of the characteristics that are relevant to etching as related to semiconductor manufacturing.
12.14 Give the various process steps that are required for making the micro pressure sensor.

CASE STUDY

MEMS PRESSURE SENSOR MANUFACTURING PROCESS

(*This part is developed with assistance from Dr. Matthias Pleil, Support Center for Microsystems Education (http://scme-support.org/), The University of New Mexico*)

Micro pressure sensors are extensively used in a number of MEMS applications such as tyre pressure measurement, automobile ABS, digital blood pressure monitors and ventilators, etc. Pressure sensing is done using the diaphragm technology where the differential pressure on both sides of the diaphragm is measured by the change in the resistance of piezo-resistive material that is attached to the diaphragm. Pressure induced strain increases the resistance value of the radial resistors. The transverse resistors also increase in resistance, but much less because of the low strain in that direction. The principle of the pressure sensor is shown schematically in Fig. 1. The piezo-resistors are integrated in the diaphragm as shown in Fig. 1. The pressure acting on the diaphragm deflects it because of the differential pressure on both sides of the diaphragm. The piezo-resistors integrated in the diaphragm are part of a Wheatstone bridge circuit whose voltage output is directly proportional to the pressure difference after calibration.

The resistor components are formed by deposition and patterning of piezo-resistive materials onto the pressure sensor diaphragm, which is a thin layer of silicon nitride. The diaphragm seals the top of a cavity which is used as the reference pressure chamber. The other side of the cavity is open to the environment and subjected to air pressure variations. As the diaphragm moves due to pressure difference, the piezo-resistors mounted on the diaphragm also stretch. The shapes of the resistances formed on the pressure membrane side of the diaphragm are as shown in Fig. 2.

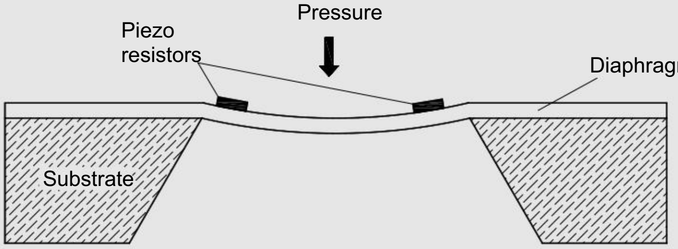


Fig. 1 *The principle of measuring pressure using piezo-resistors*

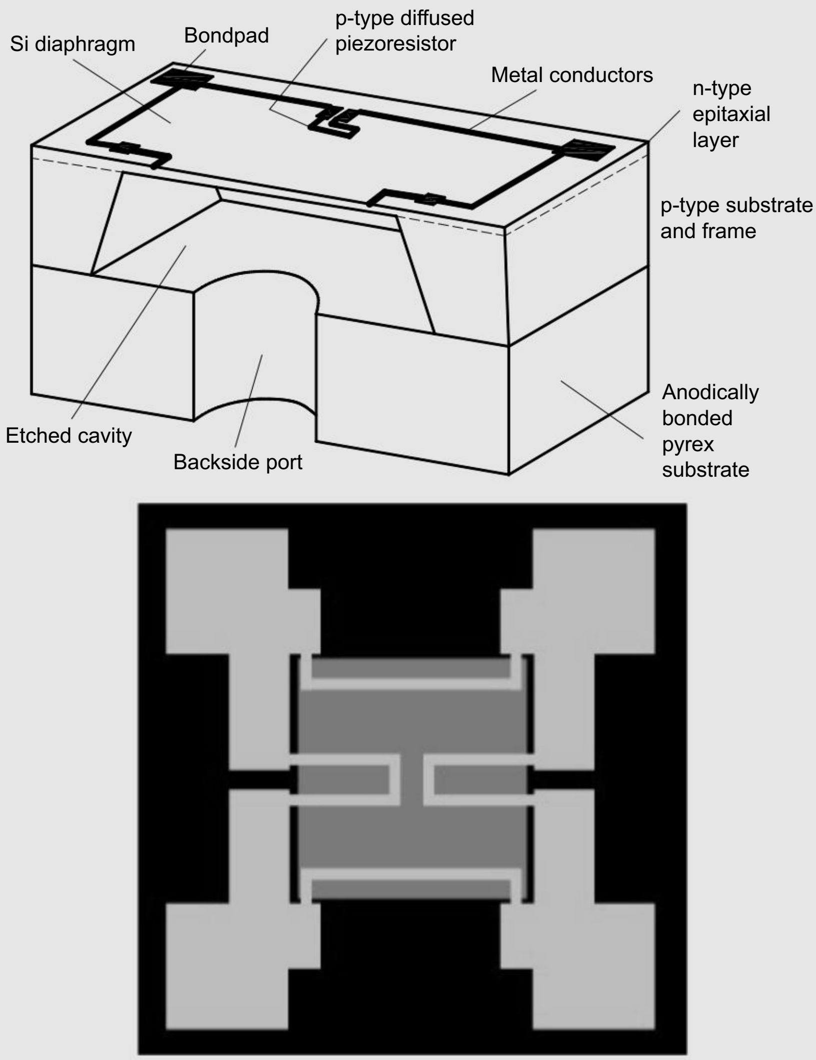


Fig. 2 *The layout of the pressure measuring piezo-resistors R1 to R4 with a photograph of an actual sensor. (Courtesy SCME, UNM)*

The process steps for micromachining the pressure sensor are schematically shown in Fig. 3.

1. *Bare Silicon*
   Starting point is standard monocrystalline silicon wafer having <100> crystal orientation.
2. *Silicon Nitride Deposition*
   The surface of the silicon wafer is deposited on both sides with 1 μm silicon nitride with low pressure chemical vapor deposition (LPCVD) process. The silicon nitride layer is used to form the pressure sensor's membrane on the wafer's frontside. The silicon nitride acts as a membrane on the frontside and also acts as a hard mask for etching the cavities on the backside of the wafer.
3. *Backside Photolithography*
   The backside of the wafer is patterned using a three step photolithography process: coat, expose, and develop. This process identifies the areas to be etched to create the reference chamber.
   (a) *Frontside and Backside Coat*
   The wafer is initially coated with HMDS (hexamethyl disilazane) and followed with the photoresist. HMDS helps the photoresist to adhere to the wafer surface. This step will coat the front side of the wafer first for protection against scratches, then the backside of the wafer will be coated.
   (b) *Backside Photolithography – Expose*
   In this step select areas of the photoresist are exposed to ultraviolet (UV) light. The areas where UV light has exposed the photoresist are subsequently developed. The mask contains the pattern that will be used to create the reference chambers.

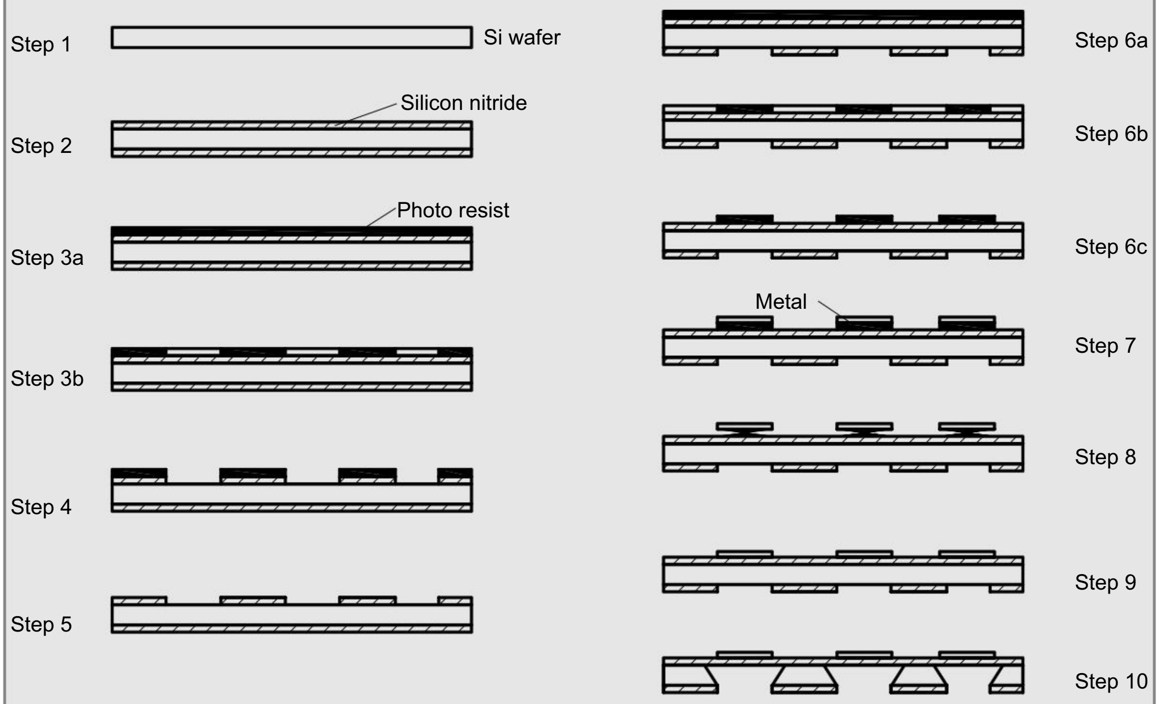


Fig. 3 *Schematic flow diagram of the process steps in the manufacture of the pressure sensor.*

(c) *Backside Photolithography – Develop*
Develop the exposed photoresist to open the windows and expose the silicon for the subsequent silicon nitride etch, which becomes the hard mask for the subsequent backside potassium oxide etch below. The developer is potassium based solution which only reacts with the UV-exposed photoresist. The unexposed regions of photoresist remain on the wafer.

4. *Backside Plasma Etch – RIE (Reactive Ion Etch)*
This step uses a plasma etcher to etch the backside silicon nitride layer through the open resist windows, exposing the underlying monocrystalline silicon wafer. The Reactive Ion Etcher (RIE) is used for a very highly selective etch. The positive ions bombard the wafer's surface to create openings in the silicon nitride layer using this dry etching technique.
5. *Frontside and Backside Photoresist Strip*
The wafers are placed in a 'Piranha bath'. Piranha is a mixture of H_2SO_4 and H_2O_2. This chemical solution aggressively removes photoresist. Extreme care should be taken when using a Piranha solution.
6. *Frontside Photolithography*
The next step in the process is to pattern the frontside of the wafer with the Wheatstone bridge pattern.
(a) *Frontside Photolithography – Coat (Lift-off resist and Photoresist)*
In this step, a lift-off resist in addition to the normal pattern resist is applied in order to create a desired undercutting during the develop process. This undercut is necessary for the subsequent deposition and successful lift off process that defines the Wheatstone bridge pattern.
(b) *Frontside Photolithography – Expose*
Exposes the Wheatstone bridge circuit pattern. The openings in the resist defines where the resistors will be.
(c) *Frontside Photolithography – Develop*
Develops the photoresist and creates the requisite resist undercut.
7. *Metal Deposition (Chrome and Gold)*
The purpose of this process step is to deposit the conductive metal layer used for the resistor bridge and contacts in the Wheatstone bridge. A vacuum evaporator deposits chrome and then gold onto the wafer. Chrome is used because it adheres well to the silicon nitride and the gold adheres well to the chrome. The gold acts as the conductive layer and strain gauge for the resistor bridges in the Wheatstone bridge.
8. *Metal Lift-off*
Metal lift-off removes select areas of chrome and gold, leaving only the chrome and gold for the Wheatstone bridge pattern. The wafer is soaked in acetone to dissolve the photoresist causing chrome/gold to lift off leaving the resistors behind. The wafer must stay wet or the metal may stick randomly to the wafer surface. LOR will remain.
9. *LOR Strip (Lift Off Resist)*
Once the PR has dissolved and the excess metal has lifted off, the remaining lift off resist (LOR) is removed using a developer solution.

10. *Backside bulk etch KOH (Potassium Hydroxide)*
    Wafers are now submerged in a heated potassium hydroxide (KOH) bath. The silicon nitride acts as a hard mask and the exposed silicon etches anisotropically following the crystal planes. The only openings in the silicon nitride that allows the etch to proceed into the silicon crystal substrate were defined during the RIE process step. The front of the wafer has no exposed silicon crystal, and silicon nitride does not etch in KOH. At the end of the etch, the wafers will be fragile since the bulk of the silicon is removed.

CHAPTER 13

Machine Tool Testing

Objectives

Testing machine tools for their accuracy and closeness to the specifications is an important element in operating them. After completing this chapter, the reader will be able to

- Understand the instruments used for checking the machine tool accuracy
- Carry out various acceptance tests for common machine tools
- Carry out various alignment tests for common machine tools

13.1 INTRODUCTION

The surface of components produced by the machining processes depends mostly on the process of their generation. As a result, the quality of surface produced depends upon the accuracy of the various movements of the machine tools concerned. It therefore becomes important to know the capability of the machine tool by evaluating the accuracy of the various mechanisms that are directly responsible for generating the surface. For this purpose, a large variety of tests have been designed. A brief review of these is given in this chapter.

13.2 MEASURING INSTRUMENTS USED FOR TESTING

The accuracy of the machine tools employed should be higher than the accuracy of the components it will be producing. Similarly the quality of the measuring equipment used for machine tool testing should be commensurate with the quality expected from such testing. A few of the more commonly used equipment is detailed below:

Dial indicators This is the most common instrument used for measuring the accuracy of the machine tool elements. A least count of 0.01 mm is generally sufficient for carrying out the tests. The plunger pressure should not be too low, which may interfere with some measurements involving the swing over, where the dial indicator may have to be in an inverted position. The dial indicator should be fixed to a rigid base with a sufficiently large and flat bearing surface.

Test mandrels Test mandrels of large variety are used in machine tool testing for various purposes. As a result, the quality of the test mandrel is important. The roundness and straightness of the mandrel is important for measuring various accuracies. In case of large length mandrels, careful consideration will have to be given to the deflection of the mandrel due to its own weight while testing. It may therefore be desirable to have hollow mandrels for large lengths. The diameter of the mandrel may have to be increased to improve the rigidity of the mandrel. The measuring length of the mandrel may vary from 100 mm to 500 mm for most cases.

Straight edges Heavy and internal stress free straight edges of sufficient length made of steel or cast iron would be used. They should have sufficiently large bearing area and ground to close tolerances with squareness of the order of ± 0.01 mm.

Spirit levels Before carrying out the machine tool tests it is necessary to level the machine tool both in the longitudinal direction as well as the lateral direction. For this purpose, the spirit levels are used. The spirit level with sensitivity of the order of 0.01 mm per metre length would be sufficient.

13.3 TEST PROCEDURES

The major tests that are to be conducted on the machine tools are:

(a) Testing the quality of the slideways and the locating surfaces,
(b) Testing the accuracy of the main spindle and its alignment with respect to other parts of the machine tool, and
(c) The accuracy of the parts produced by the machine tool.

Before the tests are conducted it is important to ensure that the machine tool is at the normal working temperature in order to establish the accuracy of the machine tool in the normal working conditions. For this purpose the machine tool should be run for a period (at least 30 minutes) which will bring all the bearing surfaces and spindles to the normal working temperature. The test when conducted under this condition will reflect the actual accuracy that can be ensured throughout its working range.

13.4 ACCEPTANCE TESTS

13.4.1 Quality of Slideways

To test the quality of the slideways, it is necessary to mount the dial indicator on a good datum surface. Then the plunger is moved along the longitudinal direction of the slideways, which will provide an indication of the undulations present on the surface of the slideways. The readings can be taken at fixed intervals (at least 10) along the length.

13.4.2 Accuracy of Spindle

These tests are related to the true running of the spindle and the centre located in the spindle along with the alignment, parallelism and perpendicularity of the spindle with the other axes (slideways) of the concerned machine tool.

True Running of the Centre

The live centre may be loaded into the lathe spindle and a dial indicator is mounted as shown in Fig. 13.1. This test is required only for the machines where the work piece is held between centres. The readings of the dial indicator are taken while rotating the spindle through a full rotation.

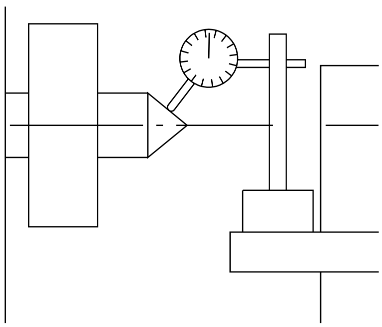


Fig. 13.1 *Test setup for true running of the centre in the lathe spindle*

True Running of the Spindle

Taper shank of the test mandrel of about 300 mm in length is mounted into the spindle as shown in Fig. 13.2 for a lathe. The plunger of the dial indicator rests on the cylindrical surface of the mandrel. The spindle is rotated slowly and the readings of the dial indicator are noted. The deviation should normally be less than 0.01 mm. The test is to be repeated with the dial indicator positioned close to the spindle bore as well as at the extreme end of the test mandrel.

Test position for milling machines is shown in Fig. 13.3 while that for the radial drilling machine is shown in Fig. 13.4. This test is to be carried out on all machines, which have the main running spindle.

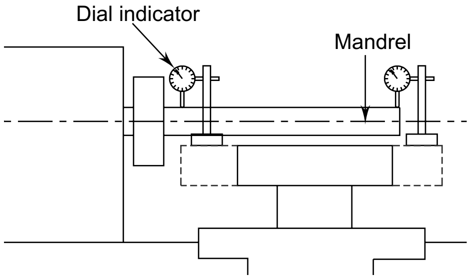


Fig. 13.2 *Test setup for true running of the spindle of a lathe*

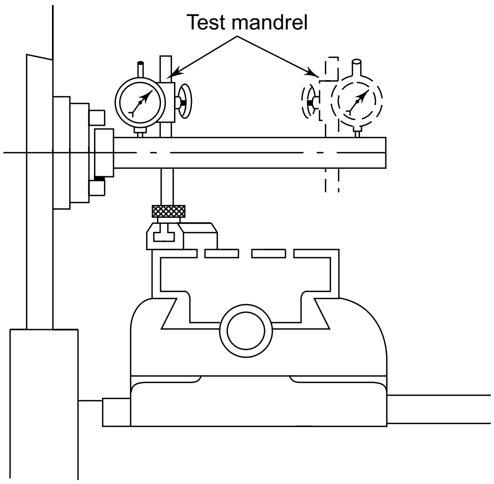


Fig. 13.3 *Test setup for true running of the spindle of a milling machine*

Squareness of the Face

This test is used to measure the squareness of the shoulder face with reference to the spindle axis. The plunger of the dial indicator rests on the extreme radial position of the shoulder face and the reading taken. It is repeated by slowly rotating the spindle till the dial indicator comes to a point that is diametrically opposite to the reading taken earlier.

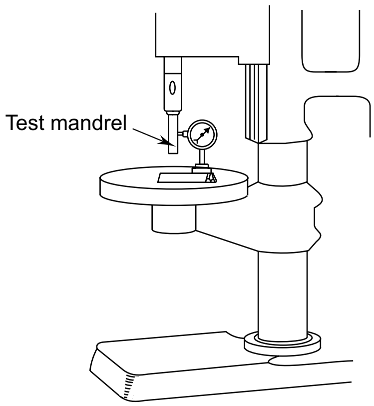


Fig. 13.4 *Test setup for true running of the spindle of a radial drilling machine*

13.4.3 Alignment Tests

These tests relate to the relative position of the spindle axis with the other axes of the machine tool.

Spindle Alignment

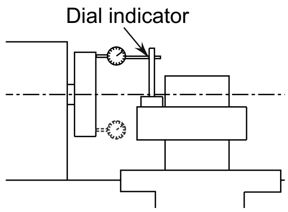


Fig. 13.5 *Test setup for the squareness of the face shoulder in a lathe*

In this test, the dial indicator is mounted on one of the surfaces whose alignment is to be tested with another surface. For example, in the case of a horizontal axis milling machine, the testing of the alignment between the spindle and the over arm support can be done as shown in Fig. 13.6. The dial indicator is mounted on the spindle while a test mandrel will be mounted in the over arm support with the plunger of the dial indicator resting on the cylindrical surface of the test mandrel. The spindle is rotated and readings are taken when it is at different positions on the periphery of the test mandrel. The test may be conducted at the two extreme ends of the mandrel.

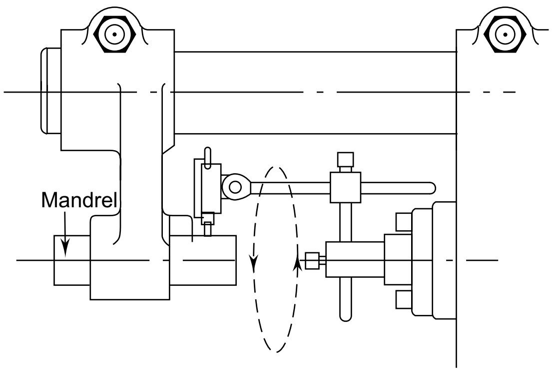


Fig. 13.6 *Test setup for the spindle alignment of a milling machine with that of the over arm support*

Parallelism and Perpendicularity

Parallelism or perpendicularity between two axes or two surfaces is normally to be measured in two planes, horizontal and vertical. For this purpose the test mandrel is mounted in the spindle as shown in Fig. 13.7 for the lathe with the dial indicator mounted on the saddle or carriage. The plunger of the dial indicator touches the mandrel surface as shown in Fig. 13.7. The saddle is moved for a specified distance and the dial reading is noted. The test is to be repeated in the horizontal direction as well.

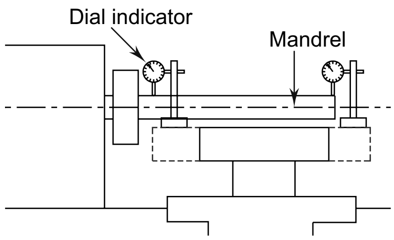


Fig. 13.7 *Test setup for parallelism between the spindle axis and the slideways in a lathe*

In the case of lathes, other tests that can be conducted are:

- Parallelism between the outside diameter of the tail stock sleeve (Fig. 13.8) and the slideways.
- Parallelism between the tail stock sleeve taper and the slideways by mounting the test mandrel into the tail stock sleeve.
- Parallelism between the lead screw axis and the slideways by mounting the dial indicator directly on the lead screw.

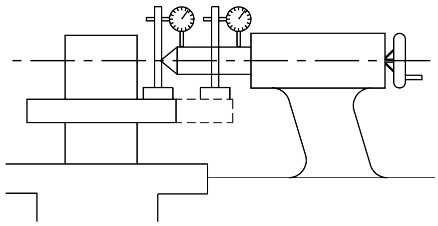


Fig. 13.8 *Test setup for the parallelism of the tail stock sleeve*

- Parallelism between the line of centres and the slideways (Fig. 13.9).

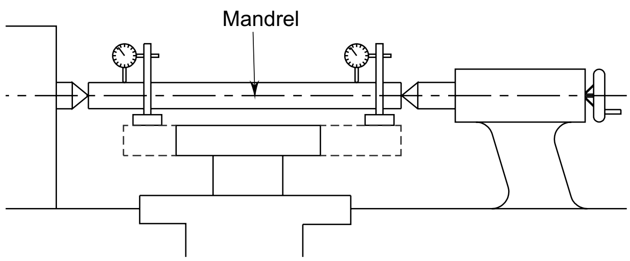


Fig. 13.9 *Test setup for the parallelism of the line of centres in a lathe*

In the case of a milling machine, the following test can be conducted:

- Parallelism between the table and the spindle axis (Fig. 13.10). A test mandrel of 300 mm long is mounted in the spindle axis and the dial indicator is mounted on the table. The reading of the dial indicator is taken at the two extreme positions without the table movement as shown in Fig. 13.10.

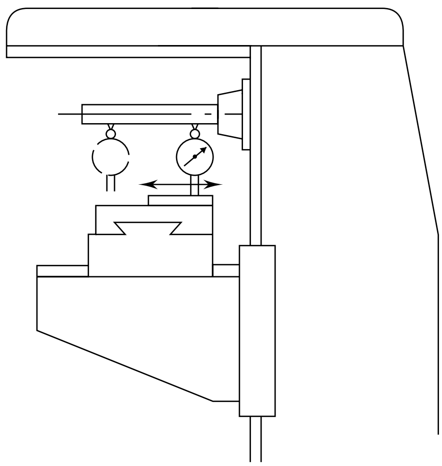


Fig. 13.10 *Test setup for the parallelism between the table and the spindle axis in a horizontal milling machine*

- Parallelism between the table surface and the longitudinal movement of the table is shown in Fig. 13.11.

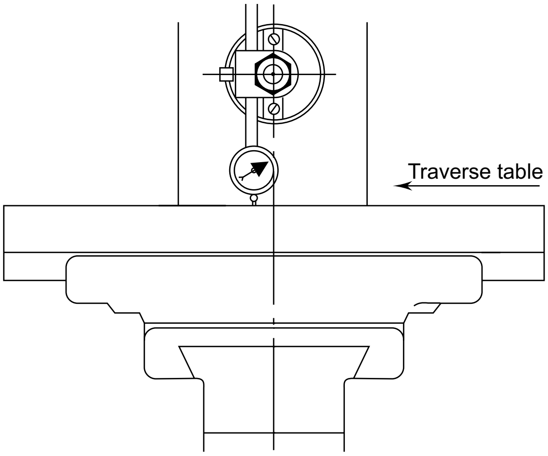


Fig. 13.11 *Test setup for the parallelism between the table and the longitudinal movement in a horizontal milling machine*

- Parallelism between the spindle axis and the transverse movement of the table. The test setup is similar to the test in (a), except that this time the reading is taken with the table moved to the extreme positions in the transverse direction (Fig. 13.10).
- Perpendicularity between the spindle and the vertical column by moving the knee. The dial indicator is fixed to the spindle and a square is clamped to the table as shown in Fig. 13.12. The dial indicator plunger touches the vertical leg of the square. The knee is then raised to take the reading at the two extreme ends of the vertical leg of the square.

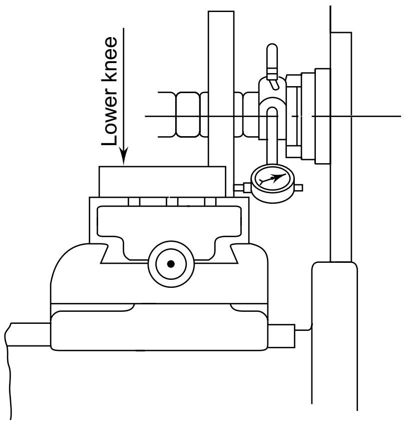


Fig. 13.12 *Test setup for the perpendicularity between the spindle and the vertical movement in a horizontal milling machine*

In the case of radial drilling machines, the following tests can be conducted:

- Parallelism between the drilling head slideways and the base plate.
- Perpendicularity between the spindle axis and the base plate (Fig. 13.13). The spindle is rotated by 360° and the reading of the dial indicator whose plunger touches the base plate is taken.

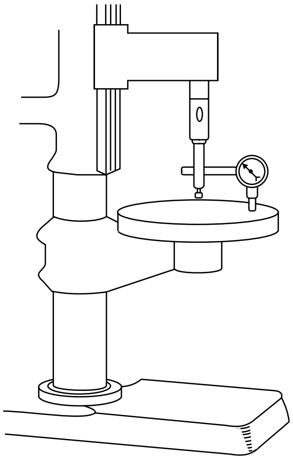


Fig. 13.13 *Test setup for the perpendicularity between the spindle and the base plate of a radial drilling machine*

- Perpendicularity between the feed movement and the base plate (Fig. 13.14). A square is placed on the base plate with the dial indicator mounted in the spindle. Readings are taken by moving the spindle up and down the vertical leg of the square as shown in Fig. 13.14.

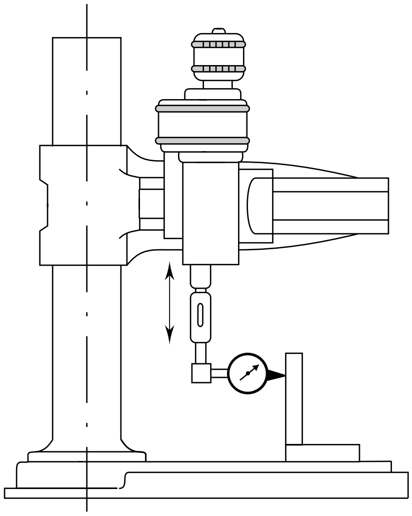


Fig. 13.14 *Test setup for the perpendicularity between the feed movement and the base plate of a radial drilling machine*

The above are only a sample of the possible testing that can be done. More details can be obtained from the Schlesinger's book or from the manufacturer's catalogues.

SUMMARY

Testing machine tools for their ability to deliver the required precision is an important attribute and needs to be done periodically.

- The various elements of machine tools that need to be tested are the slide ways, spindle and other associated components that directly result in the quality of the component.
- Alignment tests make sure the elements being tested are true to their design specifications.
- These tests are designed depending upon the motions executed by the different types of machine tools.

CHAPTER 14

Designing for Machining

Objectives

Machining being an expensive process, it is necessary that the part geometry be designed to take advantage of the process capabilities most efficiently. After completing this chapter, the reader will be able to

- Understand the various methodologies to be used for designing for machining of prismatic parts
- Understand the various methodologies to be used for designing for machining of axi-symmetric parts
- Understand the various methodologies to be used for designing for hole making operations

14.1 INTRODUCTION

During the component design stage, it is important to consider whether any of the surfaces present in the component need to be machined. If so, it then becomes necessary to do the analysis of the part, to determine if the machining is cost effective or not. The methodologies for such analysis are now well established and are normally grouped into what is called 'Design for Manufacture and Assembly' or DFMA in short. Assembly is combined with manufacture since a product rarely consists of a single component.

The Design for Manufacture and Assembly (DFMA) allows for a reduction of the assembly costs and component count along with a reduction of the overall costs while improving the reliability of the product. Boothroyd and Dewhurst have developed the methodologies and computer solutions for the same. The methodology to be adopted is shown in Fig. 14.1. The following three principles are recursively applied to all the assemblies to develop a low cost assembly.

1. During the operation of the product, does the part move relative to all other parts already assembled?
2. Must the part be of a different material or be isolated from all other parts already assembled?
3. Must the part be separate from all other parts already assembled because otherwise necessary assembly or disassembly of the other separate parts would be impossible?

Also each of the components is further analysed to see if the selected material and manufacturing process is the best or a better low cost option can be obtained. The concept of features will help a lot in this process.

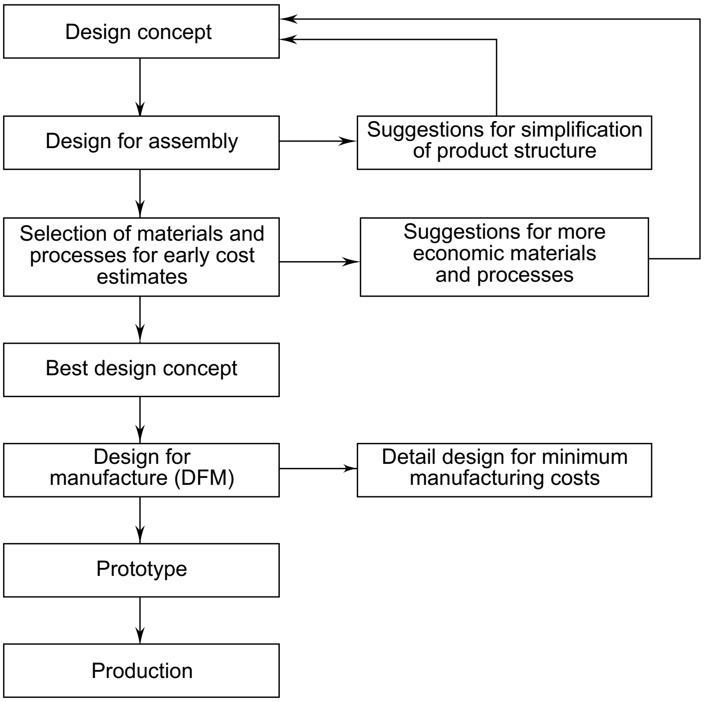


Fig. 14.1 *Methodology of Design for Manufacture and Assembly*

14.2 GENERAL GUIDELINES FOR DESIGN FOR MACHINING (DFM)

There are a certain number of points that one should consider while generally designing a part to be machined. The following are some such general guidelines that one may have to consider while carrying out the manufacturability analysis for the machined components:

1. As far as possible avoid the machining operation if the design permits. This is necessary since the cost of machining is always higher than the other manufacturing processes. The component shown in Fig. 14.2(a) is with a close tolerance as well as fine surface finish. This requires a milling operation followed by grinding to achieve the specified qualities. However, it is possible to reduce the finish and tolerance as shown in (b) such that only milling will be enough to generate the requisite quality. If the tolerance is reduced further with a specification that casting finish is good enough, then the machining process is altogether eliminated. Thus cost of the option (c) is the lowest while that of (a) is highest.
2. Use standard processes and methods. The choice of standard processes and methods allow the low cost to be achieved with the available resources. For example, use a standard cutting tool to produce a feature rather than going for a non-standard cutting tool such as form tool or special milling cutters. An example of utilising an available stock form such as hexagon is shown in Fig. 14.3(b). As shown in (a) if a round is used then an addition milling of the flats is required, which is expensive.

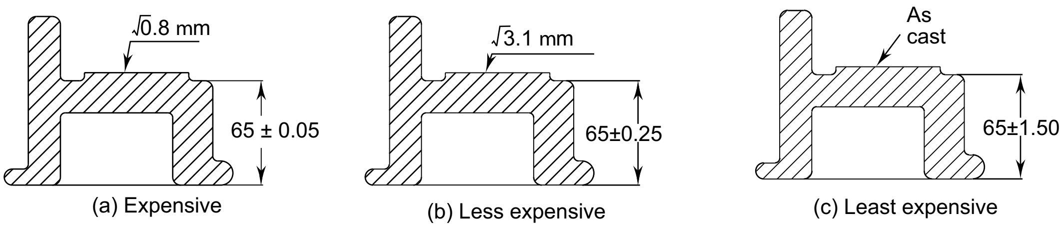


Fig. 14.2 *Type of tolerances and surface finishes that necessitates machining. Decreasing them may reduce the cost of machining or can eliminate it*

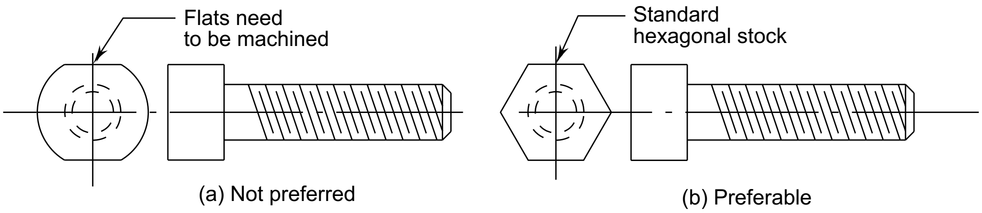


Fig. 14.3 *Use of standard stock to reduce the cost of machining*

3. Limit the manufacturing processes to those that are readily available, including the necessary expertise in them. This will make the available expertise to be better utilised while reducing the cost of acquiring new methods and technologies. However, care has to be taken while applying this principle, since sometimes adopting a new technology may be expensive initially, but may reduce the overall costs in the long run.
4. Reduce the variety of machining processes used. The total cost of machining increases with the number of setups used which automatically increases with the variety of processes used. Also the tolerances start increasing with the variety of processes used.
5. Use standard (off the shelf available) components in the design. Off the shelf components such as bearings, bolts and nuts, are normally produced in large volumes which allow for higher tolerances and lower costs, which cannot be achieved by the small batch volumes.
6. Provide liberal tolerances such that overall manufacturing cost could be lowered (Fig. 14.2).
7. Use more standard shapes such as rectangular or circular shapes, which can be easily produced by simple motions with the conventional machine tools. Surfaces such as tapers and contours call for special tools or attachments, which increase the machining cost.
8. Use materials that have better manufacturability.
9. The cutting forces in machining are generally very high and will be acting on the parts. Hence the parts which needs to be produced by machining needs to be rigid enough to withstand these forces. As shown in Fig. 14.4(a), the cutting force is likely to deflect the thin rib as shown.
10. Since many of the secondary operations such as grinding and finishing require additional cost, they be minimised or avoided.
11. The design process should be commensurate with the level of production expected of the part.
12. When a particular process is identified, exploit the special features of the process to get better economies.

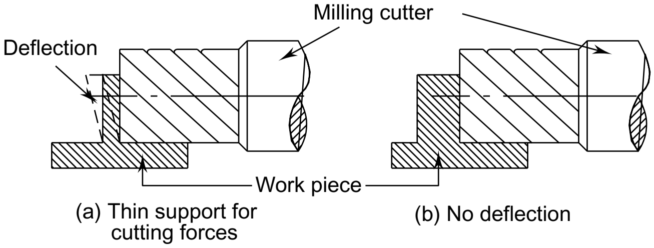


Fig. 14.4 *Parts to be machined should be rigid enough to withstand the cutting forces*

14.3 DESIGN FOR TURNING

Designing care has to be taken in cases where turning operations are involved. Turning machines such as centre lathes and automatic lathes allow components that are cylindrical to be used. In addition, these machines allow a number of different types of tools and attachments to be used. The variety of surfaces that can be produced on these machines is described in chapters 4 and 5. A few principles are presented below which may be considered while designing the parts that are to be processed in turning machines.

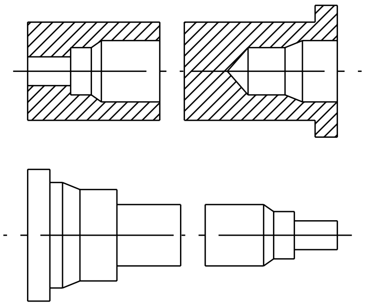


Fig. 14.5 *Parts that can be machined in a single setup in a turning machine*

1. Design the parts in such a way that the complete machining of the part can be completed in one machine itself. If possible the part should be machined in a single setup only. For this purpose, it is necessary that all the surfaces should be concentric and that the diameters of external surfaces should increase from one side while the internal surfaces should decrease from the same end as shown in Fig. 14.5.
2. External surfaces can be produced easily compared to the internal surfaces, since the cutting tools are more rigid. For example, the grooves in the parts require special tooling such as the parting tool or a boring bar with parting tool bit (Fig 14.6). As such the internal groove will be more difficult than the external groove.

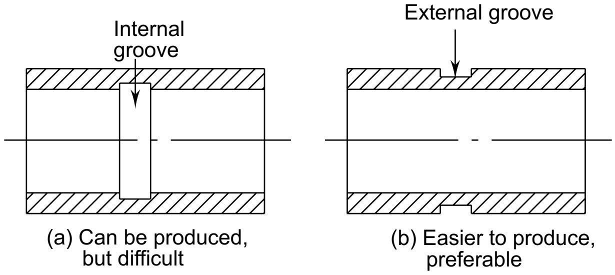


Fig. 14.6 *Machining of external surfaces is easier compared to the internal surfaces*

3. The grooves other than the rectangular grooves should have the side faces with sufficient taper to allow for a normal contouring tool to enter. For example as shown in Fig. 14.7(a), if a surface is

normal, then the tool approach becomes difficult. Generally the grooves as shown in Fig. 14.7(b) are preferable since they can be machined relatively easily.

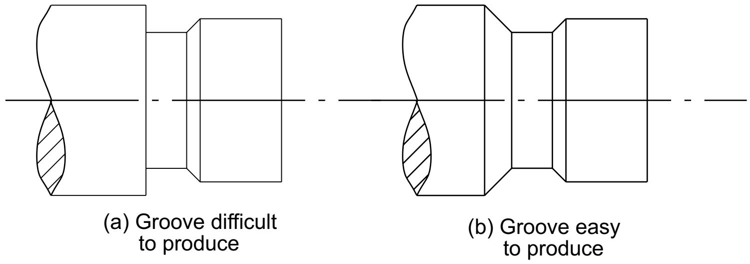


Fig. 14.7 *Parts to be machined should be rigid enough to withstand the cutting forces*

4. The exact blending of different surfaces such as cylindrical and spherical surfaces as shown in Fig. 14.8(a) will be very difficult to achieve because of the uncertainty in the machining process. Hence the two nearby surfaces are made deliberately different (the larger diameter in the case of Fig. 14.8(b) such that during the machining process no mismatch in the surfaces will be noticeable.

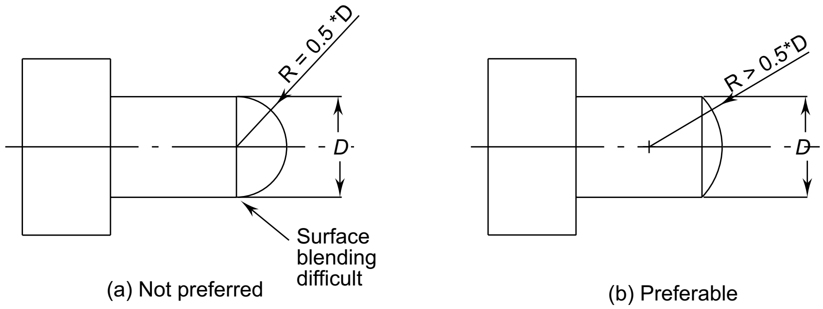


Fig. 14.8 *Different part surfaces should not be exactly matched*

5. When holes are produced by a turning machine, it is very difficult to produce a blind hole with a flat bottom. Hence it should be avoided. Ideally the bottom of the hole should have the same configuration as that of the twist drill to be used, so that no additional machining is to be done to produce the bottom surface as shown in Fig. 14.9(c). If it is not possible to have the same geometry, a small conical shape of at least 3 mm size would be required to help in the movement of the boring bar to that location as shown in Fig. 14.9(b).

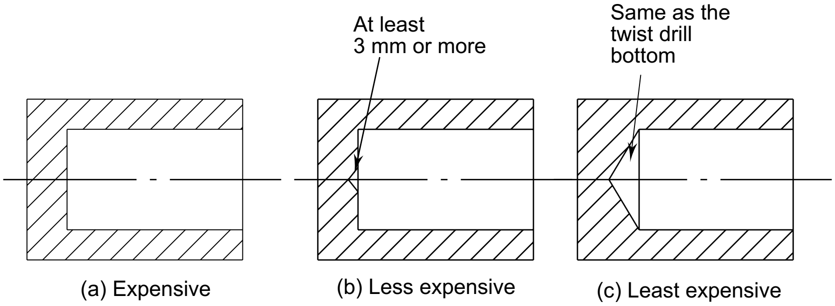


Fig. 14.9 *Parts to be machined should be rigid enough to withstand the cutting forces*

6. Sharp corners are very difficult to produce at the intersecting surfaces as shown in Fig. 14.10(a). The two cylindrical surfaces should preferably be separated with the help of an undercut between the two as shown in Fig. 14.10(b). The undercut will also be useful if a further operation is to be done on the smaller cylinder such as thread cutting or knurling. However, if only turning is to be done, then the corner radius at the intersection of the surfaces, same as the nose radius of the turning tool to be used, should be provided.

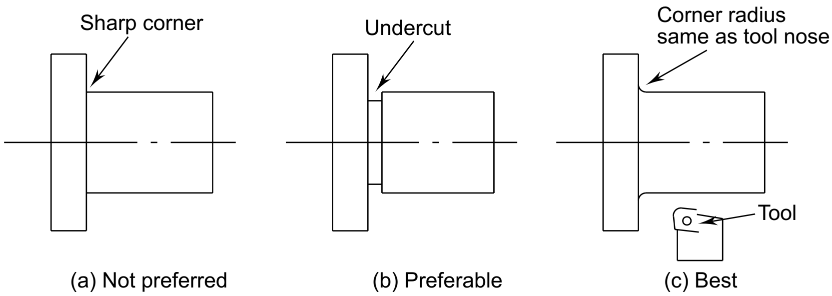


Fig. 14.10 *Designing for machining*

7. Special contoured surfaces can only be justified for a large volume production since they normally entail the use of special form tools which need to be designed.
8. The work piece should be provided with rigid surfaces to withstand the action of the cutting forces. It is generally preferable to have short and large diameter work pieces compared to the long and small diameter work pieces as shown in Fig. 14.11.

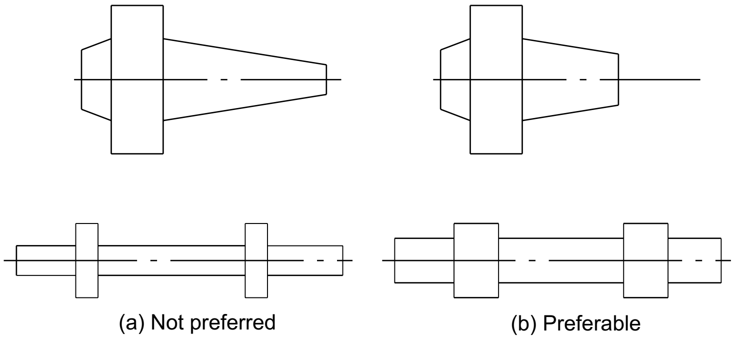


Fig. 14.11 *Parts to be machined should be rigid enough to withstand the cutting forces*

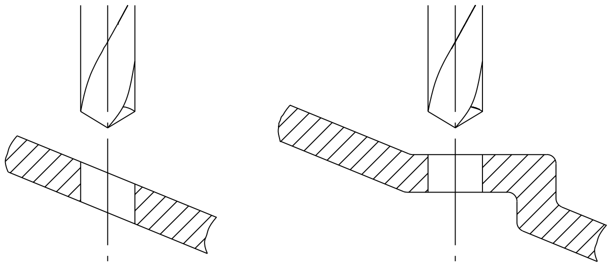
9. The design should not have interrupted surfaces because they are likely to cause impact forces.

14.4 DESIGN FOR HOLE MAKING OPERATIONS

As discussed earlier, hole making is one of the more common machining operations found in any industrial part. There are a variety of hole making operations such as drilling, boring, reaming, etc. which are covered under this common head.

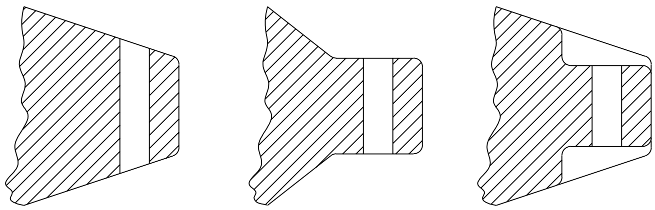
1. Hole when made in a part should preferably have the entry surface perpendicular to the drill axis. Similarly the exit surface should also be perpendicular. Otherwise the cutting will be done by a part of the cutting edges which provide an unbalanced side thrust on the drill bit causing the locational

accuracy to suffer as shown in Fig. 14.12(a) and 14.13(a). In such cases, the part geometry needs to be modified to make the entry surface perpendicular as shown in (b).



(a) Unacceptable (b) Recommended

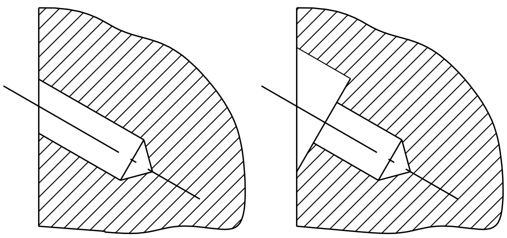
Fig. 14.12 *Drill axis should be perpendicular to the hole entry surface*



(a) Not preferred (b) Preferable

Fig. 14.13 *Drill axis should be perpendicular to the hole entry surface*

2. Use the standard twist drill sizes as far as possible. This reduces the cost of grinding the standard tool for the given size.
3. Through holes are easier to produce compared to the blind holes. Through holes facilitate the easy flow of chips and cutting fluid compared to a blind hole.
4. Too small hole sizes should be avoided, since the small size drills break easily and cause interruptions in the production process.
5. Standardise the hole sizes to be used, to help reduce the inventory of various hole making operations such as drills, boring bars and reamers.
6. Drilling of holes with inclined axis should be avoided. The web portion of the drill is likely to drift, which can affect the location of the hole. If necessary, the inclined holes should be provided with an entry surface normal to the drill axis as shown in Fig. 14.14(b).



(a) Unacceptable (b) Recommended

Fig. 14.14 *Parts to be machined should be rigid enough to withstand the cutting forces*

7. Preferably, drilled holes should not have interrupted surfaces during the drilling process. The interrupted surface as shown in Fig. 14.15(a), will cause the removal of material by only one of the cutting edges, thus providing an unbalanced force on the drill. This would cause the drill to deflect.

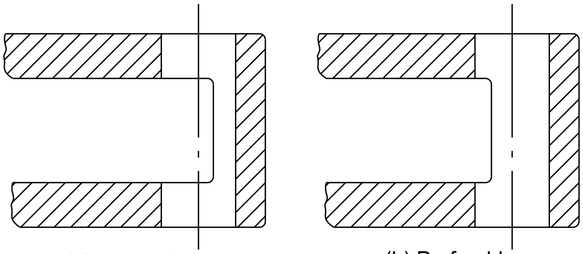


Fig. 14.15 *No interrupted surfaces in drilling*

8. Deep holes more than three times the diameter of the hole is difficult to be produced by conventional drilling. This is because of the large volume of chips generated. The work materials that produce continuous chips will further compound this problem. Special deep hole drilling methods are to be used which are expensive.
9. Deep holes that are to be bored should not be more than five times the diameter, since the boring bar becomes very slender, causing chatter. Special boring bars are to be used for such purposes.
10. Production drilling operations require that jigs be used for hole making operations. The jig bush is used to locate and support the drill during the drilling operation. This requires that the jig be designed such that the access of the jig bush is as close to the hole as possible, as shown in Fig. 14.16(b).

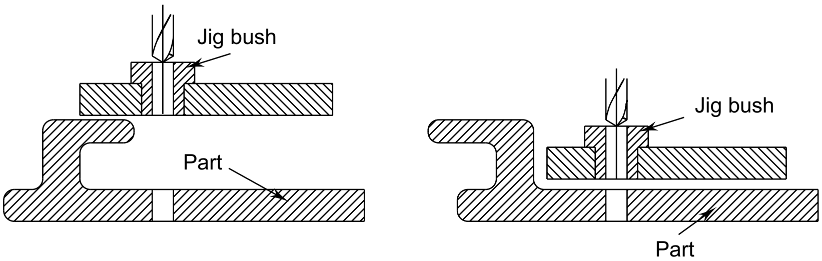


Fig. 14.16 *Provision for jig bush to stay as close to the hole entry surface should be ensured*

SUMMARY

The cost of machining can be reduced a lot by the careful analysis of the geometric design of the part.

- Make sure that the tolerances on part dimensions be specified as required, which will reduce the cost.
- Use only the processes that are commonly available and the standard stock sizes to reduce the costs.
- Try to reduce the number of setups required for machining a part which will reduce the overall cost while improving the accuracy.
- Holes when they are made should be such that the drill should enter the part surface perpendicularly and they should not go through interrupted surfaces.

CHAPTER 15

Jigs and Fixtures

Objectives

Jigs and fixtures are the common elements to be used in large volume manufacture. After completing this chapter, the reader will be able to

- Understand the need for jigs and fixtures
- Know the different types of surfaces in parts
- Understand the principles of location, support and clamping
- To select various types of locators to cater to the range of part geometries involved
- Identify the clamping requirements for jigs and fixtures
- Know the different varieties of jigs in practice
- Design jigs for practical application
- Understand the various principles for fixture design

15.1 INTRODUCTION

The accuracy achieved during a machining process depends upon the precision with which the tool and the work piece are mounted in the machine tool along with their accurate movement. Normal work holding devices such as chucks and vices are suitable for general purpose work, but when required to be used repeatedly for a large number of identical parts, the setup and clamping time become unacceptable. Hence for production work involving large number of parts, it is necessary to have separate jigs and fixtures to be designed for specific application such that the setup time is reduced to the absolute minimum that is possible.

Jigs and fixtures are the production devices that are used for the accurate production of repeated parts essentially for mass production. The required accuracy is achieved by maintaining the precise relationship between the various surfaces of the fixture and the part to be manufactured. A jig or fixture needs to provide the following functionality to be an effective production device:

- Location
- Clamping
- Support
- Resistance to Cutting forces
- Safety

Often the terms 'jig' and 'fixture' are confused and/or used interchangeably, which is not a correct practice. Both jigs and fixtures hold, support, and locate the work piece. However a jig, also guides the cutting tool. In a similar fashion a fixture has a reference point for setting the cutting tool with reference to the work piece. A typical jig component is shown in Fig. 15.1 where the drill bush supports and guides the cutting tool. In comparison, in a fixture as shown in Fig. 15.2, a setting block is used to locate the cutting tool in relation to the work piece surface to be produced.

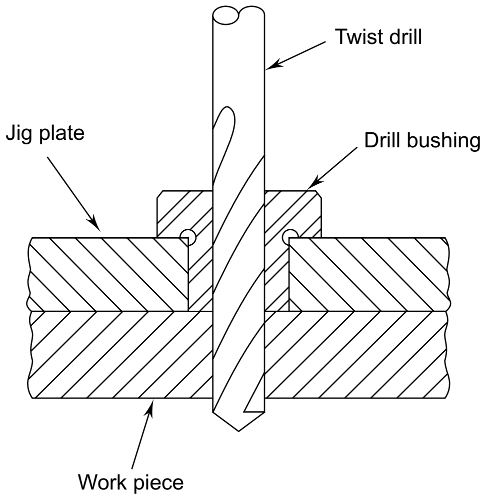


Fig. 15.1 *Part of a jig showing the jig bush and the cutting tool*

Jigs are commonly used for making parts that contain holes. The tools used for making holes are generally slender and need to be supported because of the long overhang ratios. The jig bush is employed for this purpose. The most common application of jigs is for drilling and boring. Though both of them are similar in nature the sizes are generally distinct with the boring jigs being larger in size. In addition to these applications, jigs can be used for all the hole making operations such as tapping, reaming, chamfering, counter boring, and countersinking.

Jigs are also identified by its construction as open and closed. Open jigs carry out operations generally on only one side of a work piece while closed jigs operate on two or more sides. The most-common open jigs are template jigs, plate jigs, table jigs, sandwich jigs, and angle plate jigs. Typical examples of closed jigs include box jigs, channel jigs, and leaf jigs. Other forms of jigs rely more on the application of the tool than on their construction for their identity. These include indexing jigs, trunion jigs, and multi-station jigs. Details of these types of jigs are given later.

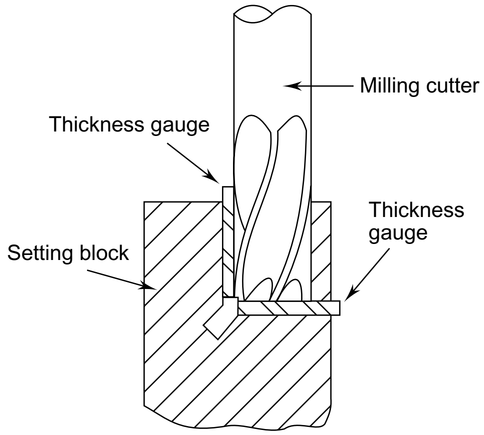


Fig. 15.2 *Part of a fixture showing the cutting tool and the setting block*

Fixtures are used for a much wider applications compared to jigs. These can be used for all other machining applications, depending upon the type of the cutting tool used and the geometry of the work piece. The motion of the cutting tool is more complex, involving the movement in two different axes and hence cannot be guided like a drill. These will be provided with setting blocks for the tool setting. The fixtures are generally identified by the machine tool in which they will be used along with the type of machining to be performed. For example, milling fixture or string milling fixture can be used to completely classify the operation performed.

Modular Fixtures

In Chapter 3, some general purpose work holders were discussed. These are used for all types of work pieces and as such are inexpensive in the long run. However they require a lot of time for setting up and therefore cannot be used even for small volume manufactures. For small volume manufactures a dedicated fixture may become expensive in terms of cost as well as lead time involved.

Modular fixturing is used for quickly rigging up a fixture for a specific component using the off-the-shelf components. Modular elements are available such as grid plates shown in Fig. 15.3. Grid plates are generally used as one of the fixturing bases. The grid plates are provided with precisely drilled and tapped holes to facilitate the clamping operation. Since the holes on these grid plates are made at precise positions, the operator would know the exact location of the component depending upon where he is clamping. These grid plates can be permanently clamped on the machine tool table if necessary.

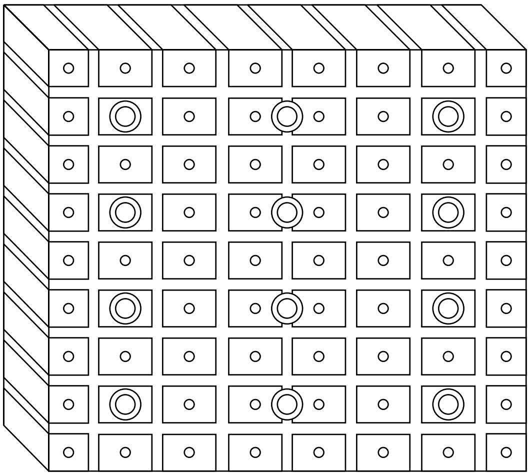


Fig. 15.3 *A grid plate with holes which can be used as a machine table*

In addition to these standard fixture bases, a large number of fixture elements such as angle blocks, base elements, locators, and clamping elements (Fig. 15.4) are available for assembling a fixture. Depending upon the outer contour of the component, it is possible to identify the various elements required from the stock elements and assemble a fixture. Since these elements are reusable, the overall cost of fixturing is less.

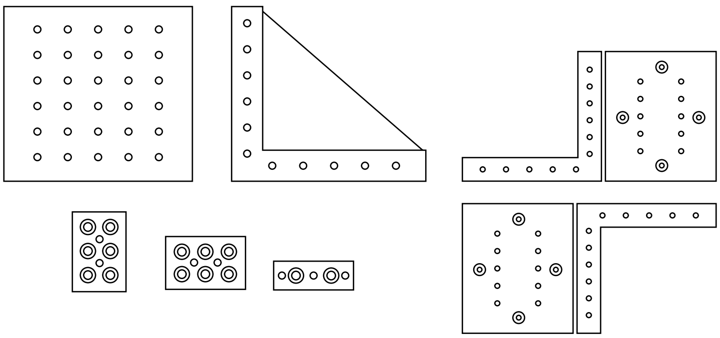


Fig. 15.4 *Modular fixture elements used for supporting complex work pieces*

Applications of Jigs and Fixtures

In a typical machine shop, jigs and fixtures are used for all types of machining operations. Sometimes they are also used for other operations such as assembly, inspection, testing, and layout. Following is a sample list of different types of fixtures that are used in machine shops. There are many distinct variations within each general classification, and many work holders are actually combinations of two or more of the classifications shown.

External Machining	Internal Machining	Non-Machining
Milling fixtures	Drill jigs	Assembly fixtures
Surface-grinding fixtures	Boring jigs	Inspection fixtures
Planing fixtures	Electrical-discharge-machining fixtures	Finishing fixtures
Shaping fixtures	Punching fixtures	
Lathe fixtures	Internal-broaching fixtures	
Cylindrical-grinding fixtures		
Band-sawing fixtures		
External-broaching fixtures		

15.2 FUNCTIONAL SURFACES

It is necessary to understand the functional surfaces present in the component and their utility from the standpoint of its manufacture. The essential reason for machining is that these surfaces are to be mated with surfaces machined in the other parts. There is no need to machine surfaces which are not likely to come into direct contact with other surfaces, except when the appearance may be important. Alternatively in some cases of high speed rotors or flywheels, machining all over the surface may be resorted to reduce the possibility of unbalance. It is always necessary to consider the fact that machining increases the final cost of the component and hence should be minimised as far as possible.

With that in background we may identify, from the basic shape of the part, the critical surfaces to be machined based on the following three criteria:

- Surface finish to be achieved
- Basic geometry
- Tolerance on the dimension or surface

The finish of the surface to be achieved is often a limiting factor in choosing a given surface for machining besides the other criteria. More discussion on the choice of machining surfaces would be done in the latter part of this chapter. There are certain surfaces in the part which should be identified because of their functional importance from the following stand point of

- location
- support (if necessary)
- clamping

for each of the machined surface.

Location Surfaces

Location surfaces are the most critical surfaces in a component from the machining point of view. They are required to be correctly identified since the accuracy achieved in a dimension depends upon the correct choice of location surface, with a view to get the lowest cost. They can generally be identified easily with the help of base lines in dimensioning or high finish already achieved in the previous operation. A complete discussion on their choice would be done at a later stage.

Support Surfaces

This is the type of surface chosen in the end after all the surfaces are identified. It is not necessary that all work pieces are to be provided with support for all operations. At times, for some components the cutting forces or clamping forces would distort a part of the work piece, because of the low rigidity of that section. As far as possible this should be avoided, but the geometry of the work piece may not allow it. In such cases proper support would have to be provided before clamping the work piece at critical points. Thus the need arises for the methods to be followed in order to identify the surfaces for support. The following are some principles that could be used while selecting the support surfaces.

- Select a surface where there is maximum likelihood for the part to deflect under the action of clamping and cutting forces.
- Support areas selected should not disturb the location of the work piece in any manner nor displace the locators while providing the support.
- Support areas selected should not interfere with the loading and unloading of the component into the work holding fixture.

Clamping Surfaces

The selection of clamping (holding) surfaces should be done in such a way that clamping of the part can be done easily in the shortest possible time with little skill. The following are some principles that should be considered while selecting areas for clamping.

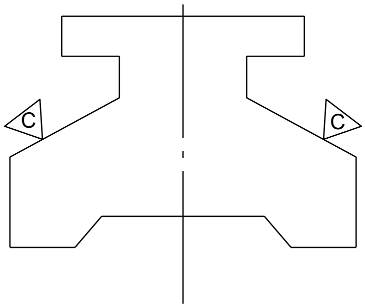


Fig. 15.5 *Choice of alternate surfaces for clamping to allow the resultant clamping force opposite to the locators*

- Generally the clamping surface should be opposite to that of a location surface, for clamping to be effective. However, normally the surface opposite to location would be the surface to be machined. Hence this choice would only be possible if that entire surface is not to be machined or a parallel surface to this is available.
- If the surface opposite to location is not available for clamping, alternate surfaces should be chosen for clamping such that resultant clamping force is acting against the locators as shown in Fig. 15.5.
- As far as possible already machined surfaces should be avoided as clamping surfaces, as they are likely to be spoiled under the clamping forces. If there is no alternative then the sequencing of the operations may be altered in such a way that this operation can be done earlier.
- Always choose the clamping surface area large enough such that the clamping forces are properly distributed and no surface plastic deformation takes place on the component. The clamping force used should take care of the cutting forces likely to develop and maintain the stability of the work piece within the fixture. Thus care has to be exercised to distribute this large clamping force over a large area of the work piece surface.
- Choose a surface with enough rigidity such that no deformation of the component takes place under the clamping forces.

15.3 LOCATION PRINCIPLES

The location of a component is one of the most important parameters to achieve the required accuracy at the lowest cost. Any wrong choice in this may lead to inaccuracy or higher manufacturing costs. The choice of location is essentially to control the equilibrium of the component in the work holding fixture under all the disturbing forces during the machining operation.

Any free body has six degrees of freedom (3 linear and 3 rotary) as shown in Fig. 15.6 (for a simple cube). Whenever location is planned it is necessary to plan the arresting of all these six degrees of freedom to ensure the mechanical stability of the component in the fixture.

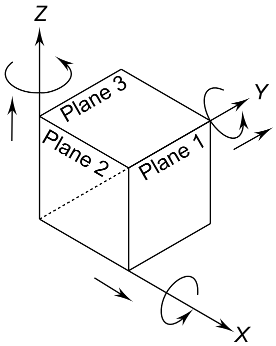


Fig. 15.6 *The maximum number of degrees of freedom for a component*

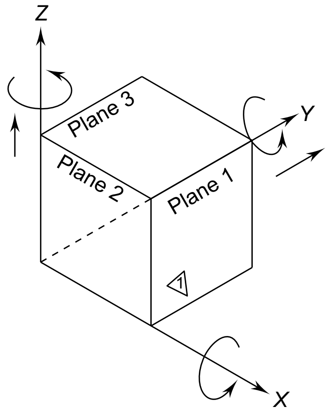


Fig. 15.7 *A component with single locator*

A single locator in plane 1 would arrest the linear motion along the X-axis as shown in Fig. 15.7. A second locator in the same plane would arrest the rotary motion about the Z-axis. Another locator placed in the same plane would arrest the rotary motion about the Y-axis.

Adding one more locator in plane 1 would not serve any purpose. Also to locate a plane only three locators are required. Fourth locator in any plane thus would be redundant and should not be placed on any single plane. Hence fourth locator can be placed in plane 2 which is perpendicular to plane 1. This would restrict the linear motion along the Y-axis. Fifth locator can also be placed in plane 2 which can arrest the rotational motion about the X-axis. Sixth locator placed in plane 2 would not serve any purpose. Hence the sixth locator would have to be placed in plane 3 which is perpendicular both the planes 1 and 2 (Fig. 15.8). This would arrest the linear motion along the Z-axis.

This is the basic location principle called 3-2-1 principle. Application of 3-2-1 principle generally gives rise to proper arresting of all the motions, but there are a few points to be considered while applying these principles.

When more than one locator is placed on a surface (plane), they should be distributed as far apart as possible on the surface.

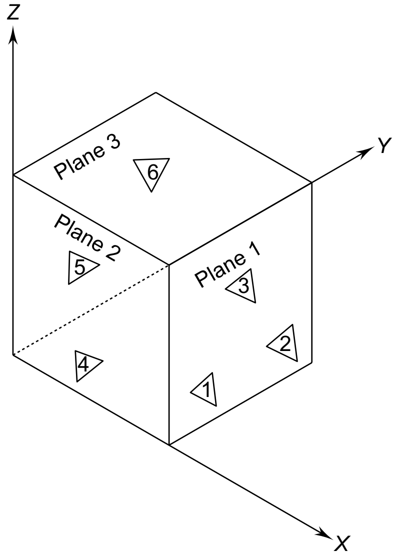


Fig. 15.8 *A component with six locators*

- This would help in placing the work piece on locators without much skill.
- Also the clamping forces would not be able to shift the work piece from such locators.
- A blank with irregular surface (such as sand casting) would be better located on such distributed locators.
- Machining forces would not be able to disturb the equilibrium of the work piece in the fixture with properly distributed locators.
- Wear of any locator contributes less to the inaccuracy of location if the locators are placed far apart. This can be examined from the Fig. 15.9. In Fig. 15.9(a) the two locators are too close, and thus due to small unequal wear on one locator a large error is caused. When the locators are moved apart as in Fig. 15.9(b), the possible error gets minimised.

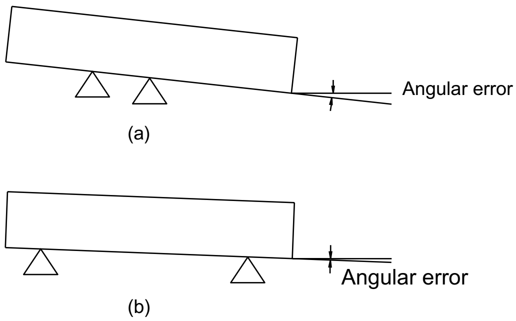


Fig. 15.9 *The effect of positioning the locators apart on a given surface*

While selecting the surface for the largest locators, consideration should be given to the largest area of the work piece. The two locators should be placed on the surface with the next largest area, and the single locator on the surface with the least surface area. This would provide for better mechanical control of the component as shown in Fig. 15.10 where because of not following this principle, the component became top heavy and is very difficult to handle.

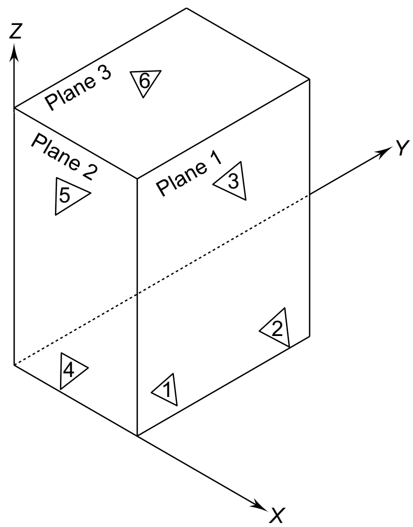


Fig. 15.10 *Mechanical stability of the component can be achieved by locating the largest number of locators on the largest surface area*

Location Examples

In the above example the discussion was based on a prismatic component having only plane faces. However, a large number of other types of components would also be part of regular production.

Cylindrical Component

In the case of cylindrical components there are two surfaces available, one the circular surface and the other is the end face. In the case of short cylinder (height small compared to the diameter as in Fig. 15.11) the three locators can be placed on the end face (since it is the largest surface) followed by two locators on the periphery as shown in Fig. 15.11. There is no other surface available for placing the sixth locator. The five locators would stop the five degrees of freedom but the sixth (rotation about the Z-axis) is not restrained.

However the friction present between the locators on the cylindrical surface will function as the sixth locator in stopping this rotation. In fact in all cylindrical components friction forms the sixth locator and only five locators would be placed.

For long cylindrical surfaces the method of holding would be slightly different since the largest surface is the curved surface. Four locators can be placed on the curved surface instead of the maximum three for the plane surface as shown in Fig. 15.12. The arrangement would be to place two locators on the cylindrical surface at one end, and the other in the opposite end along the axis. The fifth locator would be placed on the end face as shown in Fig. 15.12. The four locators while arresting the four degrees of freedom would also locate the centre line of the cylinder which is a very important criterion for location in a cylindrical work piece. As in the earlier case friction may be assumed as the sixth locator.

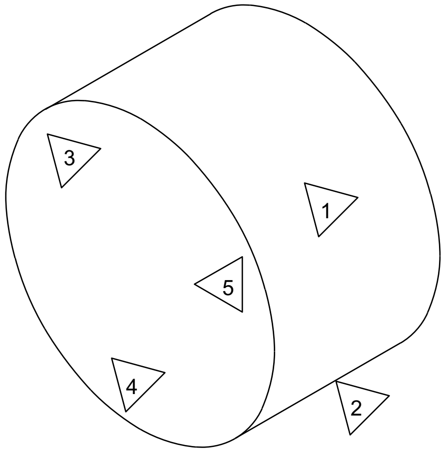


Fig. 15.11 *Location method for a disc type (short cylindrical) component*

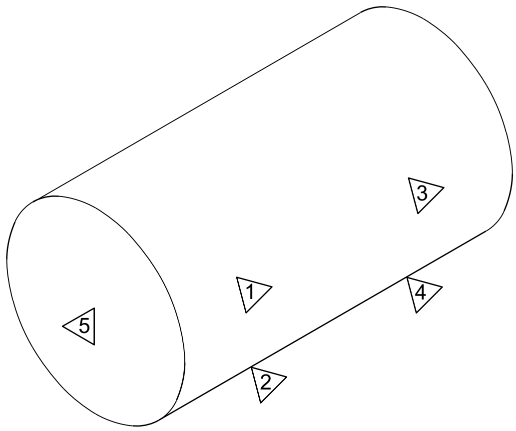


Fig. 15.12 *Location method for a long cylindrical component*

Component with Holes

The same principles can be applied to components where through holes are present, and the hole is to be used for location. For short work pieces with hole the location method is similar to the short cylindrical case. The three locators would locate the end plane whereas the two locators placed inside the hole would locate the axis of the work piece. For long work pieces with through holes the concept of long cylinder can be used where the four locators would now be placed inside the hole. Fifth locator may be placed on one of the end face.

Conical surfaces would have a similar arrangement to that of cylindrical surfaces. Though the arrangements shown above are general and would be useful for finalising the location arrangement for a majority of work pieces, the odd shaped components require careful analysis to see how the location surfaces can be identified.

15.4 LOCATING DEVICES

It is important that the fixture should locate the part consistently with respect to the work piece quickly and easily. This will help in achieving the required accuracy of the machining operation with a very small amount of time spent in the setting up of the work piece. The six degrees of movement of the work piece need to be arrested by the appropriate choice of locators.

Depending upon the nature of surface to be located, there are three general forms of location: plane, concentric, and radial. Plane locators are used to locate a flat surface on a work piece. The surface may be flat, curved, or have an irregular contour and the locators are accordingly used to nest that surface as shown in Figure 15.13(a). Concentric locators on the other hand locate a work piece from its axis. The most-common type of concentric location is a locating pin placed in a hole as shown in Figure 15.13(b). Some work pieces may have a cylindrical projection that requires a locating hole in the fixture, which is also a concentric locator as shown in Fig. 15.13(b). Radial locators restrict the movement of a work piece around a concentric locator as shown in Figure 15.13(c). Again the locator can used in an existing hole in the work piece for that purpose or an external surface can be used as shown in Fig. 15.13(c). It is also possible to use a combination of these location methods in a given work piece depending upon the geometry requirements.

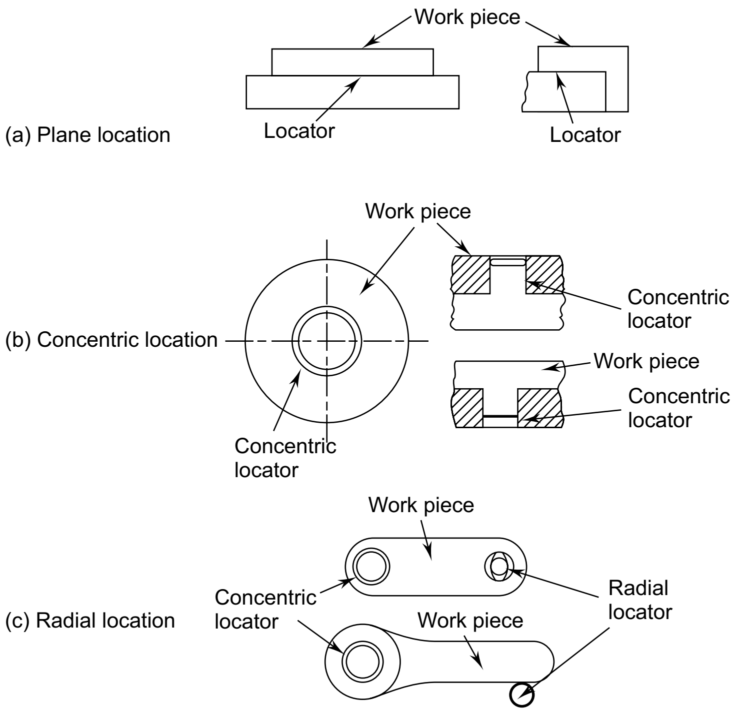


FIG. 15.13 *Three types of locators used in various fixture*

The cylinder is the best shape for a locator. It is easy to produce and at the same time arrests the 5 degrees of freedom. Also, when a hole is produced in a work piece it is generally produced to a higher tolerance and that helps in improving the locational accuracy. The next best available location is two finished plane surfaces that are at right angles. It is also easy to use for loading and unloading the work piece from a fixture. Typical design of a round locator is shown in Fig. 15.14. However a round locator with a flat surface can be used to locate a flat surface as shown in Fig. 15.15.

Locators are sometimes relieved to minimize the area of contact between the work piece and the locator as shown in Fig. 15.16. This reduces the possibility of locator jamming inside the part.

Diamond pins are often used for radial location as shown in Fig. 15.17. As explained earlier one cylindrical locator (pin A) arrests 5 degrees of freedom, which is termed as the principal locator. The second cylindrical locator at position B will arrest the sixth degree of freedom. The pin A will be slightly longer than the other pin such that the part is located on it and then rotated till it is engaged with the second locator. If the two

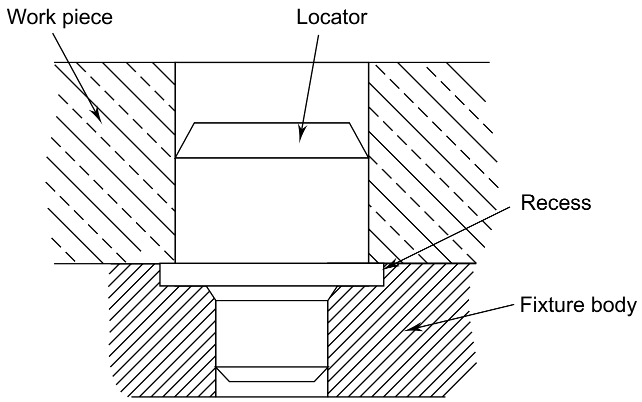


Fig. 15.14 *The round locator used in locating a work piece with an already finished hole*

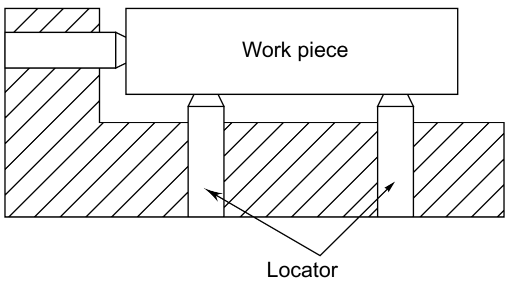


Fig. 15.15 *Three locators are used in a fixture to locate a planar surface*

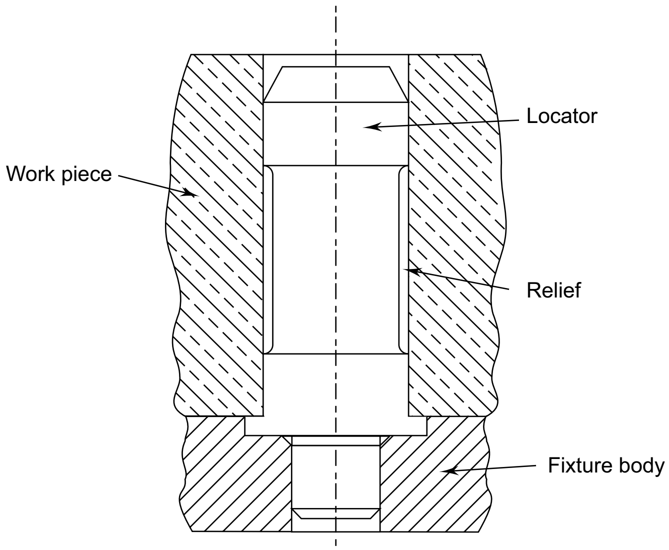


Fig. 15.16 *Relieved locator showing the reduction in the contact area between the locator and the work piece surface*

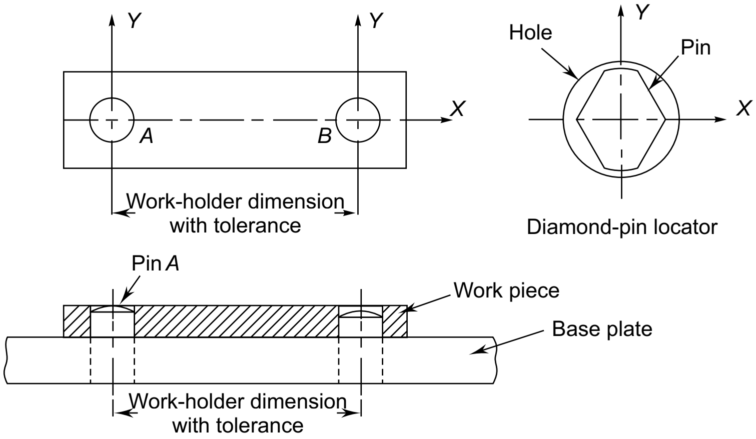


Fig. 15.17 *Diamond pin used for radial location*

holes are identical in size then any pin can be made the principal locator. However if one of the holes is larger, then the principal locator will be placed in the larger hole. The second locator is made slightly smaller than the hole and relieved from both sides to take care of the variation in the X direction. The cylindrical surfaces will locate the part in the Y-direction. The construction details of the round pin and diamond pin locators are shown in Fig. 15.18.

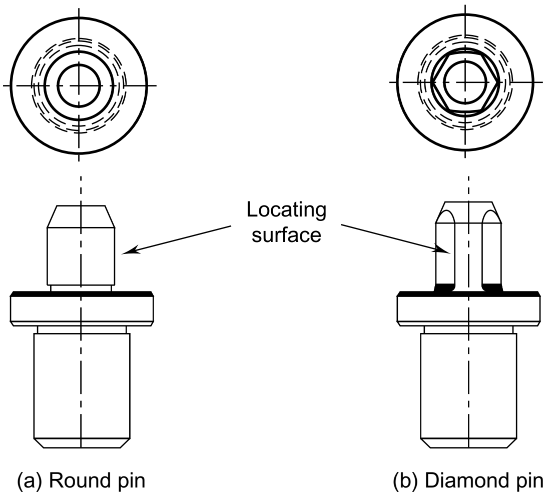


Fig. 15.18 *Round pin and diamond pin locators used in fixture design*

Bullet-nose round pins (Fig. 15.19) and diamond pins are often used together (with the mating bushes) to align two pieces of a fixture similar to a dowel. Diamond pins as explained earlier are relieved to locate only in 1 axis. The pin's shank diameter and the bush's OD are the same size, to allow boring the installation hole in both fixture pieces at the same time, for greater accuracy.

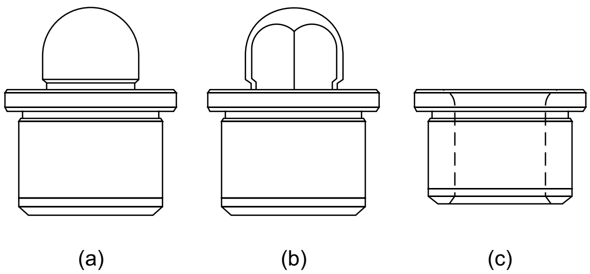


Fig. 15.19 *Bullet nose designs used as locators for fixture elements*

Redundant Locators

A redundant location is the case when two locators are trying to constrain one degree of freedom from two different locations. An example is shown in Fig. 15.20(a). The part at (a) shows how a flat surface can be redundantly located. The part should be located on only one and not both the surfaces. Since the sizes of parts can vary, within their tolerances, the likelihood of all parts resting simultaneously on both surfaces is remote. The part in Fig. 15.20(b) is fully located from the two pins in the holes while the other three locators are nesting the outer surface of the part, restricting the already constrained degrees of freedom. It is impossible to have part dimensions to be exactly located within these constraints. The solution is that depending upon the locating surface used either the design shown in Fig. 15.21(b) or (c) is acceptable.

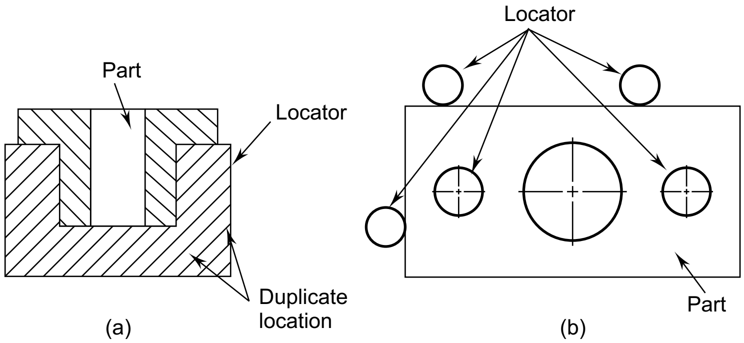


Fig. 15.20 *Redundant location*

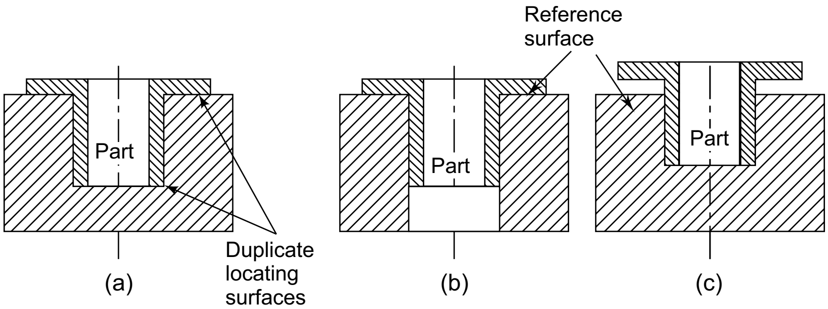


Fig. 15.21 *Improved location by removing the redundant locations*

For parts that are symmetrical it is often a problem for the operator to correctly place the part in the fixture. In such cases a fool proofing pin is located on the fixture base such that the operator will be able to place the part in the correct orientation. If the part is loaded wrongly as shown in Fig. 15.22(b), the part will not fully enter because of the presence of the pin as shown.

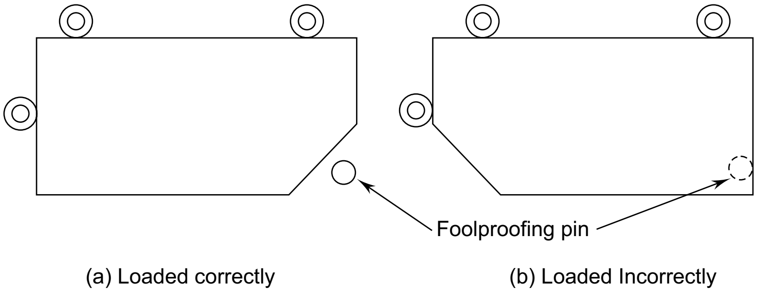


Fig. 15.22 *Fool proofing a fixture*

The work piece dimension that is used for location determines the locating element size. The principal rule to determine the size of the work piece locator is that the locators must be made to suit the MMC (Maximum-Material Condition) of the area to be located. The MMC of a feature is the size of the feature where it has the maximum amount of material. In the case of a cylindrical locator, make the locating pin slightly smaller than the hole. For example, if the hole is specified as 10.00 – 10.25 mm in diameter, the locator must fit the hole at its MMC of 10.00 mm. Allowing for a 0.015 mm clearance between the pin and the hole, desired pin diameter is calculated at 9.985 mm. Standard locating pins that ground to several different hole tolerances are readily available from fixture element manufacturers such as Carr Lane.

Supporting

A simple device that can often be used for supporting is the rest button. The construction of a press fitted rest button is shown in Fig. 15.23. It actually serves the purpose of supporting the part from any direction as shown in Fig. 15.24.

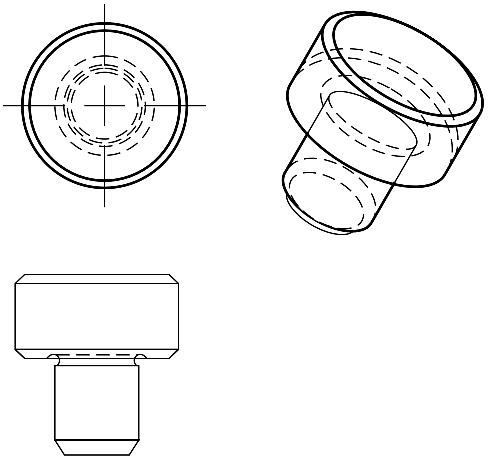


Fig. 15.23 *Rest button (Courtesy Carr Lane Manufacturing)*

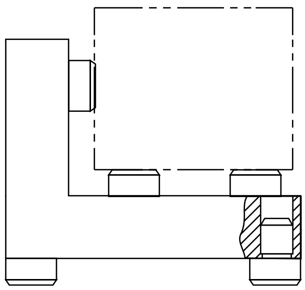


Fig. 15.24 *Use of Rest button for supporting the part in the fixture*

15.5 CLAMPING DEVICES

The function of clamping is to ensure the part remains stable during the cutting process. Clamps should hold the work piece against the locators and keep it stable against all the disturbing forces that work on it during the cutting process. It is important to understand the nature and magnitude of the cutting forces in order to properly decide on the type of clamping arrangement desired in a fixture. Vice is a good example of a fixture as shown in Fig. 15.25. The fixed jaw is used to locate the work piece. The movable jaw clamps the work piece against the fixed jaw with the help of friction between the work piece and the jaws. When the machining operation is planned, it is necessary to make sure that the force due to the cutting operation should act against the locator. The arrangement shown in Fig. 15.25, makes the work piece slip between the jaws when the cutting force exceeds the frictional clamping force of the jaws. Hence it is an unsatisfactory arrangement for clamping. The cutting force should always be aimed against the fixed jaw as shown in Fig. 15.26.

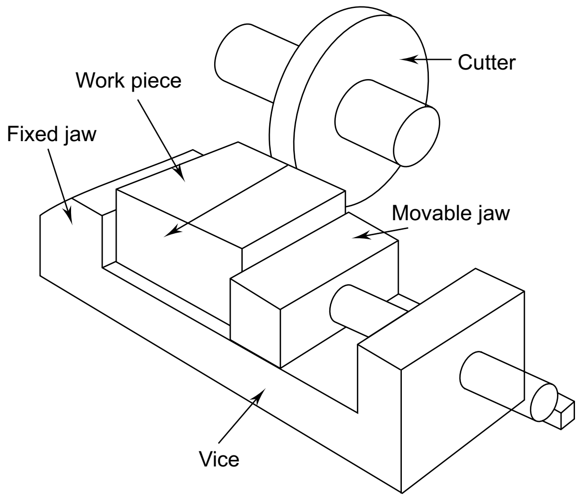


Fig. 15.25 *Vice as a fixture showing the relationship between the cutter and the clamping arrangement*

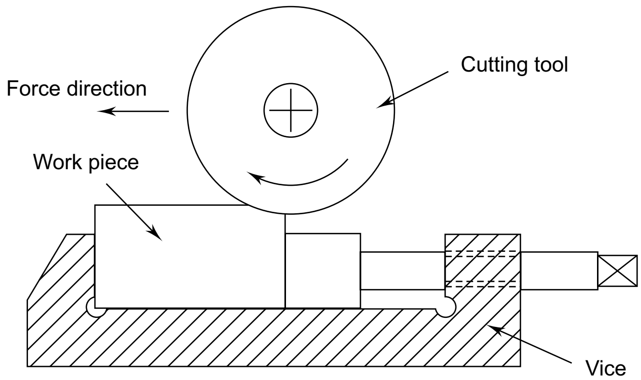


Fig. 15.26 *Correct arrangement for directing the cutting force against the fixed jaw thereby ensuring that the part remains stable in the vice during the cutting operation*

The following are a few simple rules that could be observed during the selection of the clamping elements required for a fixture:

1. Always use simple clamps since complicated ones may lose effectiveness as they wear.
2. Rough work pieces call for a longer travel of the clamp in the clamping range, but clamps may be made to dig into rough surfaces to hold them firmly.
3. The type of clamp required is determined by the kind of operation to which it is applied. A clamp suitable for holding a drill jig leaf may not be strong enough for a milling fixture.
4. Clamps should not make loading and unloading of the work difficult, nor should they interfere with the use of hoists and lifting devices for heavy work.
5. Clamps that are apt to move on tightening, such as plain straps, should be avoided for production work.
6. The anticipated frequency of setups may influence the clamping means. For example, the use of hydraulic clamps, even if simple and of low cost might be inadvisable if frequent installation and removal of piping and valves is necessary.

Basic Type of Clamps

There are a number of types of clamps that are used by tool designers for clamping the part properly as follows:

- Strap Clamps
- Screw Clamps
- Cam Clamps
- Toggle Clamps
- Equalizers

The tool designer has to choose the type of clamp that is simple and easier to use and at the same time provides the right kind of productivity.

Strap Clamps

By far these are the simplest type of clamps used in jigs and fixtures. There are a variety of designs to be used. Most of these clamps are based upon the lever principles to amplify the clamping force required. A typical strap clamp application is shown in Fig. 15.27. By tightening the stud in Fig. 15.27, the clamping force is transferred to the part. Heel pin is the fulcrum about which the lever acts, while the clamping force is applied at the stud by tightening the screw. The actual force is transmitted to the part at the end of the strap as indicated in Fig. 15.27.

The actual amplification of the applied force depends upon the distance between the stud and the heel pin (B), and that between the stud and the part (A) as shown in Fig. 15.28. The distance A should be made as small as possible compared to B to increase the mechanical advantage of the clamp to increase the clamping force on the part. A variety of strap designs as shown in Fig. 15.29 are used in strap clamps. The choice of these depends upon the clamping requirement, part geometry and the relationship of the cutter in relation to the clamping surface.

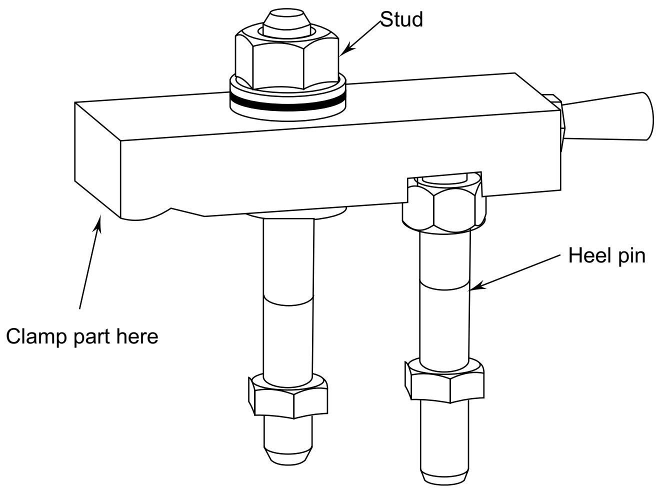


Fig. 15.27 *Strap clamp used for clamping in fixtures*

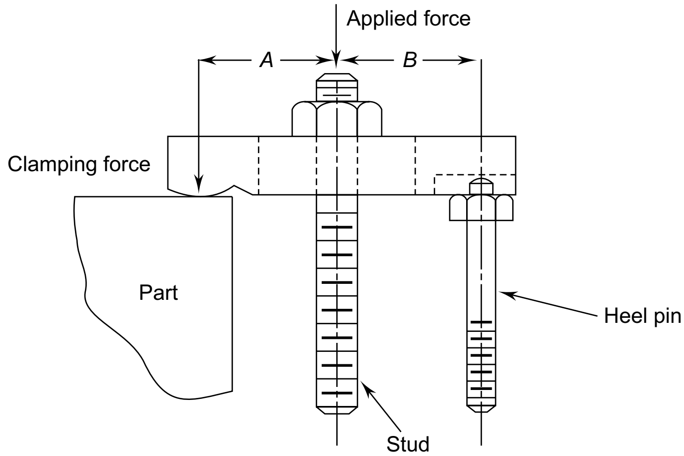


Fig. 15.28 *Strap clamp showing the mechanical advantage by using the lever principle*

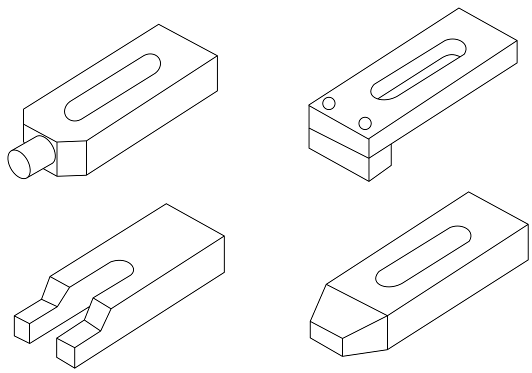


Fig. 15.29 *Variety of strap designs used in strap clamps*

Further variation of the strap clamp design depends upon the way the clamping force is applied on the part. Three different designs are shown in Fig. 15.30. The differences between the three types of clamps is the way the force is transferred to the part, by means of a hinge, sliding through a closed slot, or an open slot like a latch.

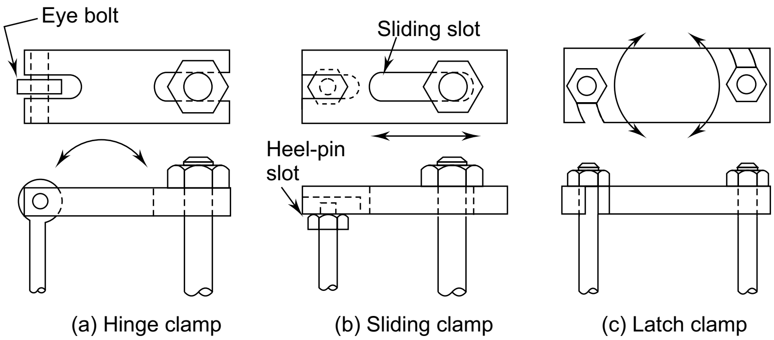


Fig. 15.30 *Variety of strap designs used in strap clamps*

Screw Clamps

Screws rarely are used for direct clamping. However, practically a large number of clamps make use of screws for the purpose of clamping. In the strap clamps discussed earlier screws are used to apply the clamping force. However these clamps require considerable time to fasten. A much faster way of applying clamping is to make use of either a swing washer or a cee-washer if the work piece has a bore for clamping. A swing washer as shown in Fig. 15.31 can be used to clamp a part having a hole. In order to release the part, the nut needs to be opened slightly so that the swing washer becomes loose, at which time it can be swung to the side thereby releasing the part. This helps in loading and unloading the part quickly. The only condition is that the hole used for the clamping should be larger than the nut used for clamping as shown in Fig. 15.31. A cee-washer as shown in Fig. 15.32 is similar to swing washer, but remains loose unlike a swing washer. Otherwise applications of both are very similar.

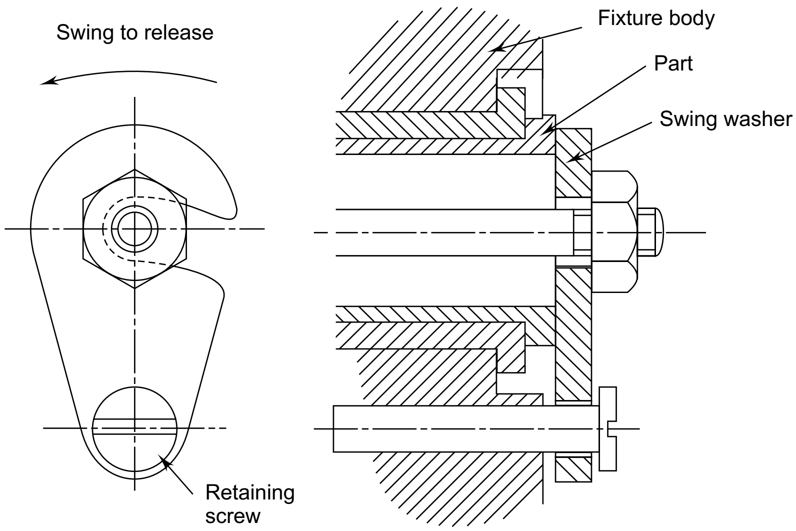


Fig. 15.31 *A swing washer used for clamping a part with a hole*

Cam Clamps

Cam clamps provide clamping force because of the contour of the cam surface that comes into contact with the plate used for the clamping. Typical cam clamp design is shown in Fig. 15.33. Use of cam clamp is shown in Fig. 15.34. Notice that a plate is pushed down by the cam against the spring pressure to hold the part in place. Cam clamps are quick in operation. Cam clamps are of three types, eccentric cam, flat spiral cam and cylindrical cam. The design shown in Fig. 15.33 is flat spiral and is the most commonly used clamp. The design shown in Fig. 15.34 is indirect pressure clamping where the pressure is transmitted to the part through the plate. This is more stable and the vibrations during machining do not affect the clamping.

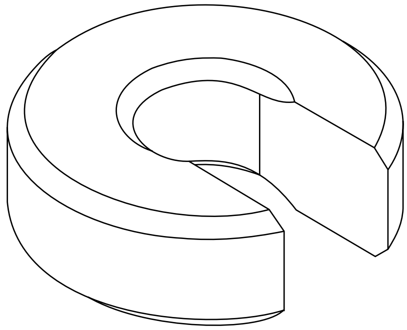


Fig. 15.32 *A cee-washer used for clamping a part with a hole*

Fig. 15.33 *A cam clamp used for quick and easy clamping a part*

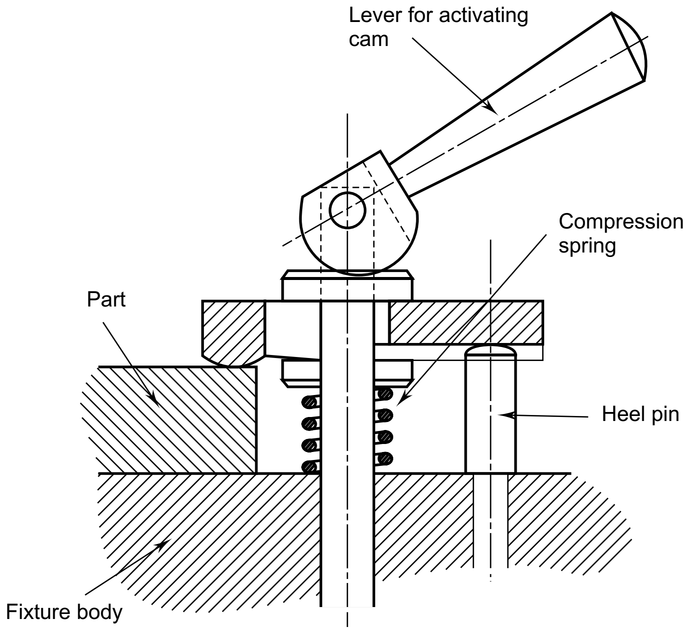


Fig. 15.34 *An example of a fixture held by a cam clamp*

Toggle Clamps

These are commercially available clamps in various designs. A toggle clamp is a quick-acting mechanical linkage where two of the elements make up a toggle action. Actuating the clamp first moves it into position, then applies clamping force by compressing or stretching the linkage elements after contacting the work piece, then positively locks it by moving the toggle action's centre pivot past the centreline of the other two pivots, against a stop. There are a number of designs, and a few types of toggle clamps are shown in Figs. 15.35 and 15.36. Toggle clamps are mainly used because of their fast action for clamping and unclamping, their ability to completely clear from the work piece, and the force amplification possible for clamping.

Fig. 15.35 *A toggle clamp of hold down action type with horizontal handle*

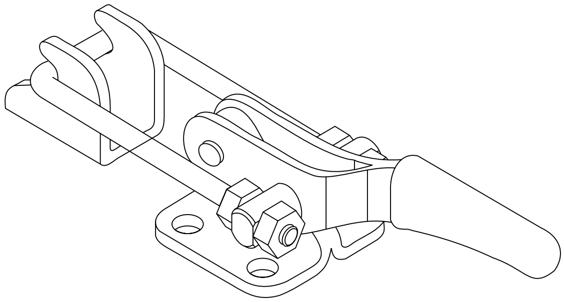


Fig. 15.36 *A push pull type toggle clamp*

Equalizers

When the clamping force is to be applied at more than one location, then an equalizing clamp is useful. In this type of clamp as shown in Fig. 15.37, the link arm system is used to apply an equally divided clamping force to a pair of clamps acting on the same component. It is also possible to use this system of clamping to clamp two parts. This is particularly useful in a condition where the operator may be denied easy access to one or other of the clamps.

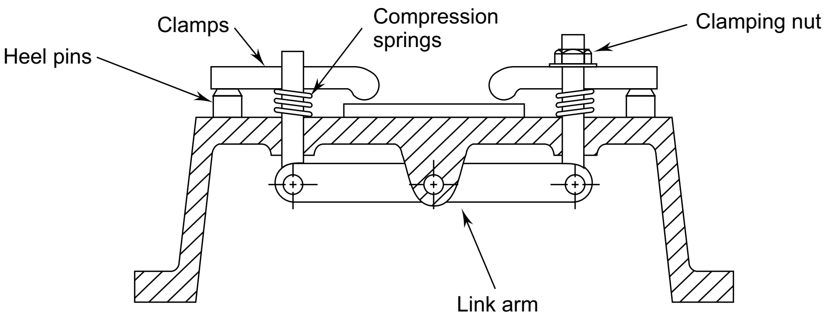


Fig. 15.37 *An equalizing clamp*

15.6 JIGS

A fixture is a device used to securely fasten a part to the machine tool table to accurately locate, support and hold the part during the machining operation. A jig is a special class of fixture, which in addition to providing all the functions as above, also guides the cutting tool during machining. In a fixture normally a gage will be provided to locate the setting of the cutter with respect to the work piece. Jigs are generally used for the operations such as drilling, boring, reaming, tapping, counter boring, etc.

The main advantage of jigs is that it minimizes tool breakage because it supports the tool during the operation. It also minimizes the possibility of human error by loading the part into the jig in only one way against the locators. It allows the use of less skilled labour since the skill is built into the jig. The overall manufacturing time is reduced since the setup time for the part and the tool is reduced to the minimum by the use of jig bushes, locators and clamps.

Drill Jig Bushes

The main problem with hole making operations is the slenderness of the tool. It further gets complicated by the geometry of the hole as well as its location. Hence a jig bush is used to position and guide the cutting tool for the cutting operation. Jig bushes are made of materials with sufficient hardness to ensure long life. Typical materials used are hardened steel, carbide, bronze and stainless steel.

There are a number of varieties of jig bushes used to cater to the requirements of hole making. Fig. 15.38 shows a headed drill bush, which is the most commonly used. These are used when the hole depth must be controlled. To reduce the drill bending and improve the hole accuracy, drill bushes are mounted as close to the work piece as possible while allowing adequate chip clearance. The necessary clearance depends on work piece material and type of chip formed. In the case of cast iron, discontinuous chips are formed and hence would require only about 0.5 times drill diameter for chip clearance. However materials that produce long and continuous chips, such as cold-rolled steel and aluminium, require at least one-drill-diameter clearance. It is not advisable for the drill bush to directly contact the work piece surface. Headless drill bushes are used when the drill depth is not important as shown in Fig. 15.39. This is least expensive and used for light axial loads.

Sometimes when the drilling axis is not perpendicular to the work piece surface it is necessary to locate the exit end of the bush as close to the part as possible as shown in Fig. 15.40. Otherwise the drill will tend to wander. For maximum drill guiding it is recommended to specify bushes with an angle milled on the exit end, tangent to the work piece surface at the point of entry.

If a hole requires multiple operations such as drilling followed by reaming or counter boring, then slip bushes are used as shown in Fig. 15.41. A liner bush is located

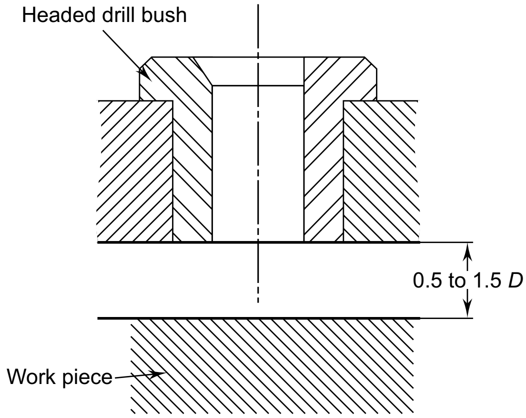


Fig. 15.38 *A headed drill bush*

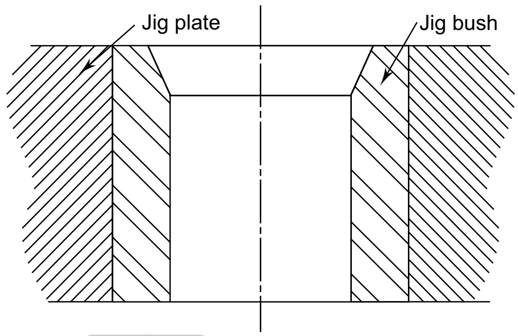


Fig. 15.39 *A headless drill bush*

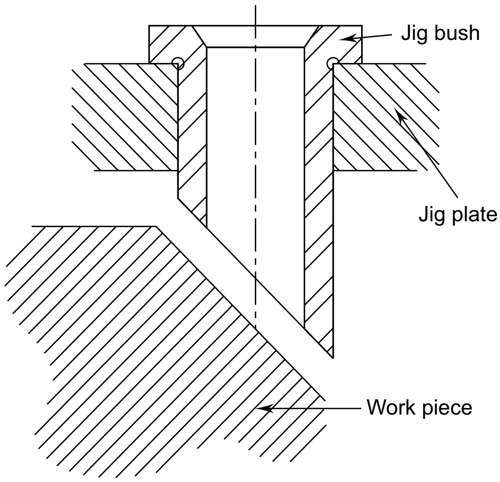


Fig. 15.40 *A special drill bush shaped to reduce the side thrust on the drill for making a hole on an inclined surface*

in the jig plate and a slip bush is located inside the liner bush. To prevent the rotation of the slip bush, a retaining screw is used as shown in Fig. 15.41. The liner bush guides the large diameter cutting tool. A renewable bush is similar to the slip bush, but it will only be replaced when the bush is worn out due to the large volume of work done. For replacing the renewable bush the retaining screw has to be completely removed as shown in Fig. 15.42.

The jig bush is mounted into the jig plate by means of a press fit. Follow the recommendations of Carr Lane Manufacturing company for the recommended hole sizes in unhardened steel or cast iron jig plates to reduce the distortion of the jig plate. Other factors to be considered are: (1) headed bushes require less interference to resist drilling thrust; (2) longer bushes in thick plates require less interference; (3) bushes with thinner walls are more prone to distortion; (4) less-ductile jig-plate materials require less interference.

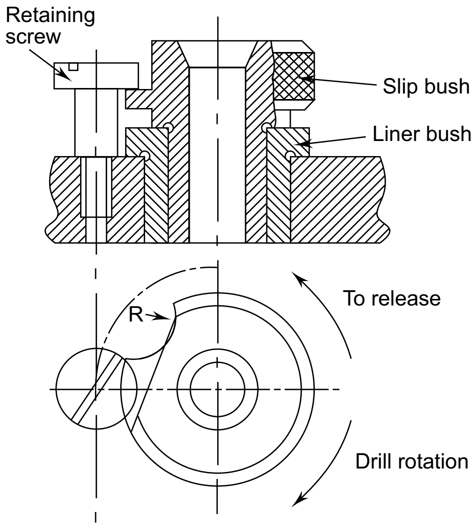


Fig. 15.41 *A slip renewable bush used for multiple operations such as drilling followed by reaming*

Types of Jigs

Drill jigs as explained earlier would have the following provisions:

- Correctly locate the work piece with respect to the tool
- Securely clamp and rigidly support the work piece during the operation
- Guide the tool
- Position and/or fasten the jig on a machine (normally for small jigs)

There are a large variety of jigs used for the hole making operation. The type of part geometry involved dictates the actual choice of these designs. The common varieties of jigs are

- Template jigs
- Plate jigs
- Leaf jigs
- Channel and tumble jigs
- Indexing jigs

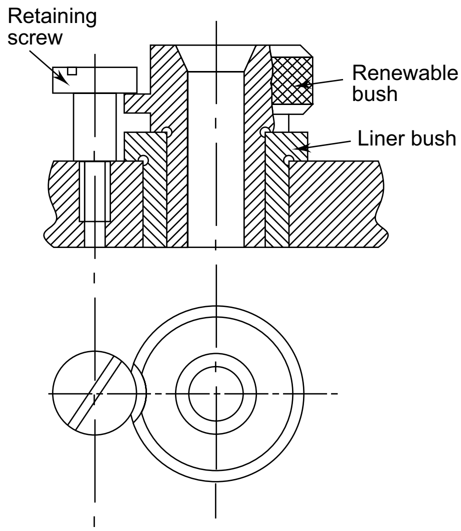


Fig. 15.42 *A renewable bush used for large volume drilling when the bush needs to be replaced due to the wear*

Template Jigs

It is not a true jig since it does not incorporate a clamping device. However these are used in a number of situations. A template jig consists of the jig plate with the necessary locating arrangement and the provision of the jig bushes where the hole needs to be made as shown in Fig. 15.43.

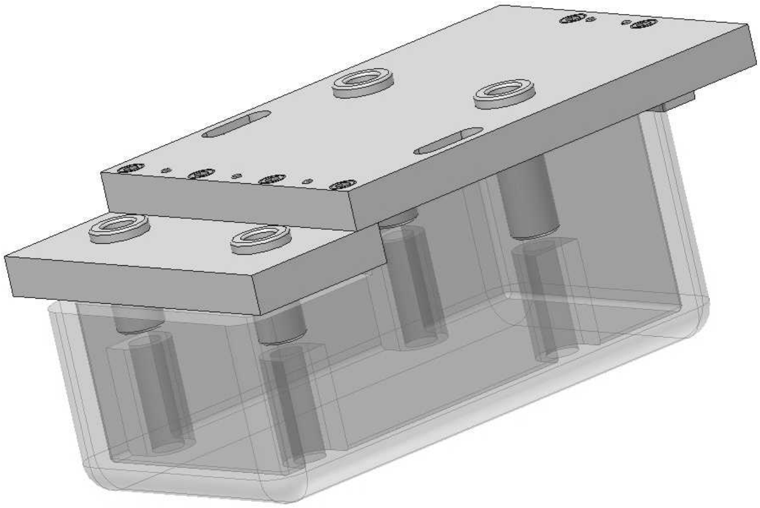


Fig. 15.43 *A template jig*

Advantages

- No clamping arrangement
- Plates with bush to guide the tool
- Directly placed on the part
- Simple
- Least expensive

Disadvantages

- Not as fool proof as other types
- Orientation of the hole pattern to work piece datums may not be as accurate as other types
- They are usually not practical when locating datums are dimensioned

Plate Jigs

This is an improvement over the template jig. A plate jig is a template jig with an added clamping arrangement. Fig. 15.44 shows a plate jig for drilling the six holes in a part. The part, shown as transparent, is clamped to the central cylindrical locator underneath the plate with the help of a C-washer. The jig plate has the six bushes arranged around the locator as per the part print dimensions. Because of the open construction employed in the plate jig, it is easy to load and unload parts and also dispose of the chips.

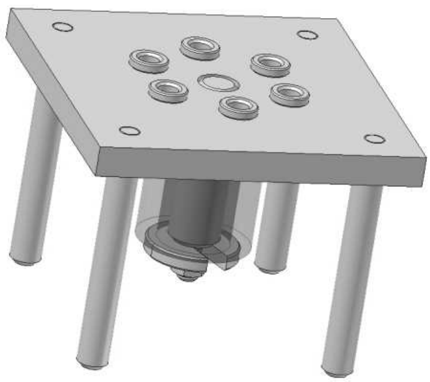


Fig. 15.44 *A plate jig*

Leaf Jigs

Plate jig is the simplest of all jigs in which the component is positioned between location elements, sandwiched between the base and the jig plate. A pair of alignment dowels ensures that the jig plate is correctly orientated relative to the base. The two parts are clamped together by a cam latch as shown in Fig. 15.45. The hinged leaf with bushes will also apply the clamping force. Most of the designs are normally limited to small and simple parts for easy handling. The main disadvantage is that as wear or distortion takes place in the pivot pins, the accuracy of machining will deteriorate.

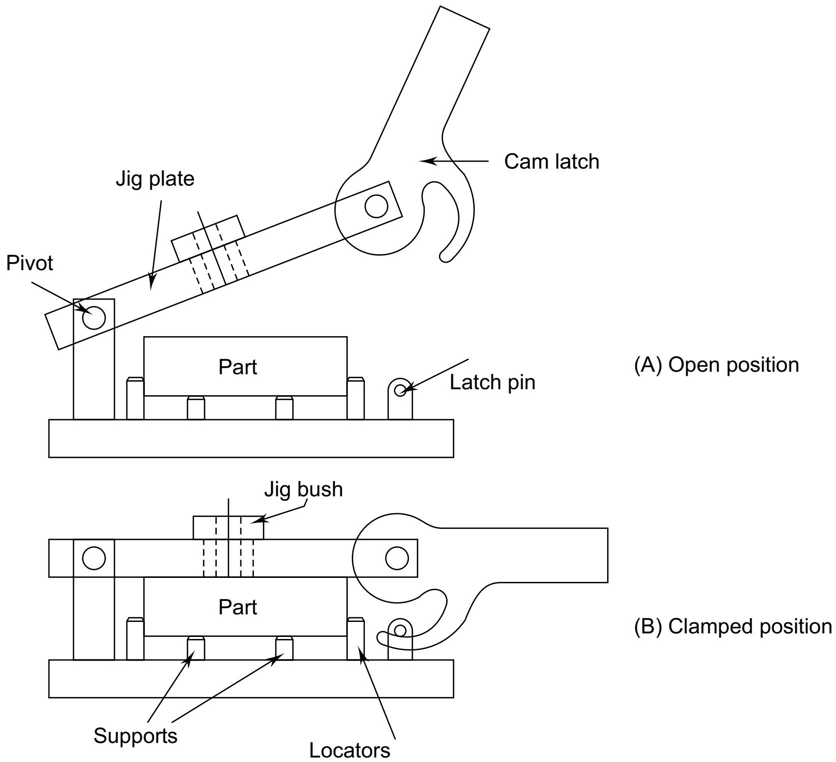


Fig. 15.45 *A leaf jig*

Channel and Tumble Jigs

Channel and tumble jigs allow for drilling in more than one surface of a part without relocating it in the jig. As a result the accuracy of the part is higher, and less handling of the part is required to complete the machining operations. However the jigs are more complicated and expensive compared to other jigs discussed so far. A simple channel jig is shown in Fig. 15.46.

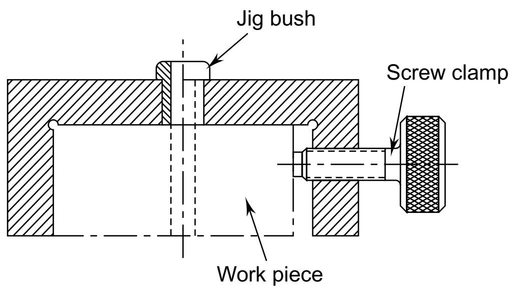


Fig. 15.46 *A channel jig*

Indexing Jigs

Indexing jigs are used to drill holes in a pattern. The location for the subsequent holes are normally done through the prior hole drilled. An indexing arrangement will be provided in the jug from an appropriate datum to ensure the required accuracy. An example is shown in Fig. 15.47 where the part is located from the central hole and then indexed about its axis by means of a plunger located to the left side, to drill the 4 holes around the cylindrical surface.

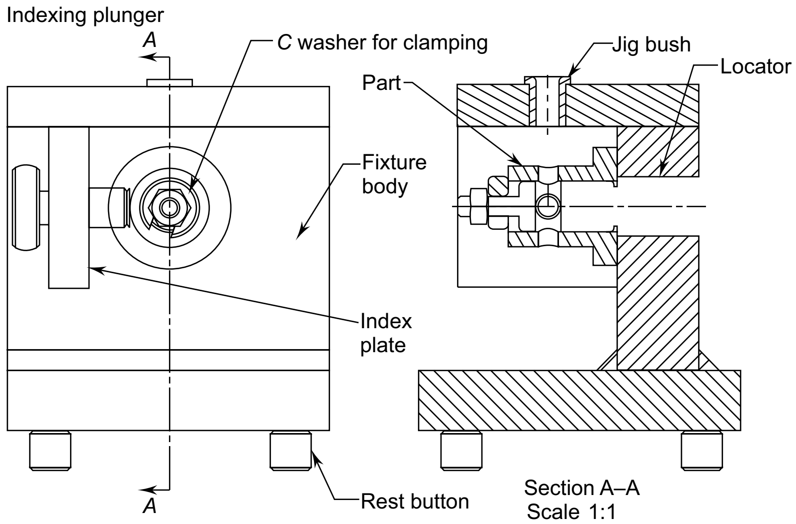


Fig. 15.47 *An indexing jig*

In addition to the above, there are a number of variants of these types used in the industry. In this book only the basic types are covered, while specialized books on 'Jigs and Fixtures' normally provide a more detailed coverage on all the possible types.

15.7 DESIGNING A JIG

To design a jig it is very important to follow the steps methodically. It is necessary to think and plan various elements that will be forming part of the final jig. It often helps in sketching the details around the part to understand the interactions between the various elements. With the availability of 3D CAD systems, it is far more easier now to design jigs compared to a manual process on the drafting board.

The following are some steps that are identified to help in designing a jig.

- Method of locating the part – identify the standard components required for locating purpose.
- Design the clamping method. Make a proper choice of clamps – C-washer, swing washer, nut, strap clamp, toggle clamp, etc.
- Design any supports required
- Design the jig bushes required.
- Design the jig body.

Let us now go through the jig design for the part shown in Fig. 15.48 to apply the above principles. The part has a central hole, which is already finished and could be used for location. The four identical holes around the cylindrical body need to be machined using this jig.

Based on the preliminary examination of the part, the following points emerge:

- Method of locating the part – The central hole, which is already finished, can be used for locating as well as indexing for the holes to be drilled.

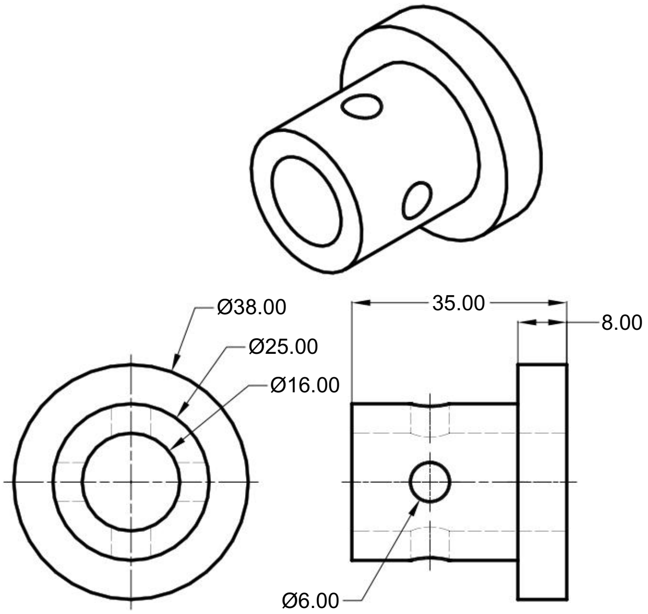


Fig. 15.48 *A part for designing a jig*

- Clamping can be done with a nut and a C-washer since the central hole is large enough.
- Indexing can be done with a retracting type plunger going into the previous hole drilled.
- The jig can be a simple indexing jig with a post for locating the part.

Armed with this preliminary information let us design the jig. Because of the nature of the part, it is not possible to make use of any standard components for locators. It is necessary to design a cylindrical locator, which could be used for the purpose of indexing as well as clamping of the part. Such a design is shown in Fig. 15.49. The large diameter of the locator is the surface used for locating it in the jig body. The locating surface is to mesh with the hole in the part (16 mm diameter). Since the part is well supported by the locator, no separate supports are required.

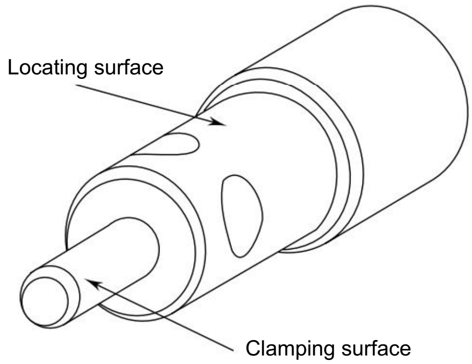


Fig. 15.49 *Locating arrangement for the part shown in Fig. 15.48*

The next arrangement that is required is the indexing to drill the holes, which are at 90º to each other around the central axis of the part. For this purpose a retractable plunger will suffice. It is a relatively simple and a standard component available from the fixture component manufacturers. The locator should be relieved where the plunger will be entering for the purpose of indexing as shown in Fig. 15.50.

Next step is identifying the type and size of jig bush required. A headless type of bush will not serve the purpose, since it has to go beyond the jig plate. Hence a standard head type jig bush is selected as shown in Fig. 15.51. This completes all the essential elements required for the jig.

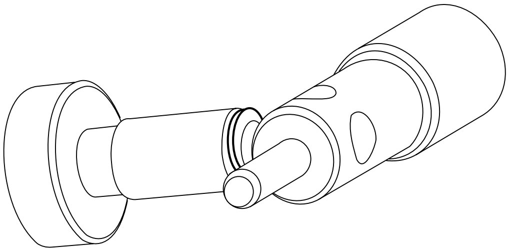


Fig. 15.50 *Indexing arrangement for the part shown in Fig. 15.48*

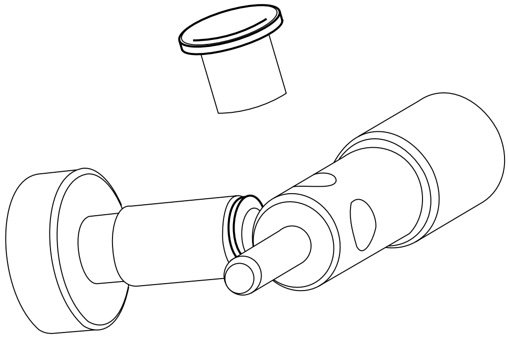


Fig. 15.51 *Jig bush location for the part shown in Fig. 15.48*

Having all the essential elements in their expected locations, it can be observed that there is no interference between any of these parts and the system will work as intended. The next step therefore is to add the jig base to keep all these elements in their respective positions. For this purpose an L-shaped structure as shown in Fig. 15.52 should be sufficient. It can be an integral shape (cast or extruded section) or a welded structure depending upon the requirements. Notice the provision of the dowels for the purpose of mounting the jig plate and the index plate. Next add rest buttons to the structure for stability as shown in Fig. 15.53. They are standard parts, and a total of four are used so that the operator can make sure that the jig is standing square on the machine table and is not 'rocking'.

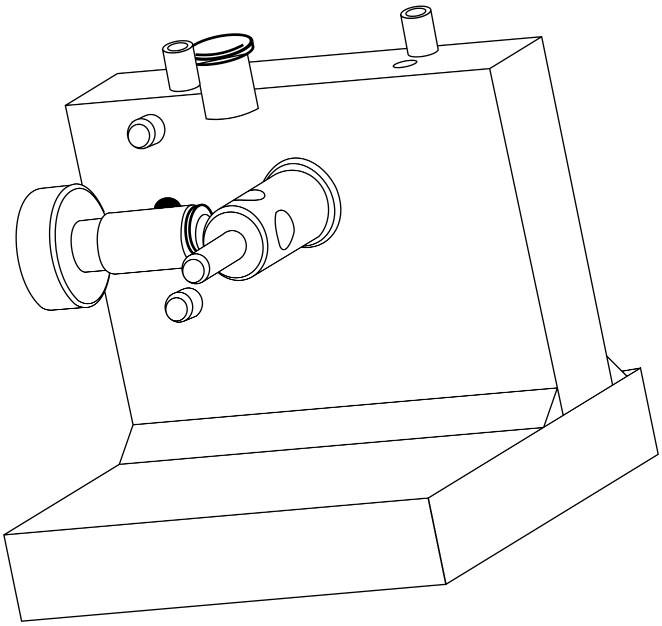


Fig. 15.52 *Support structure for the locator for the part shown in Fig. 15.48*

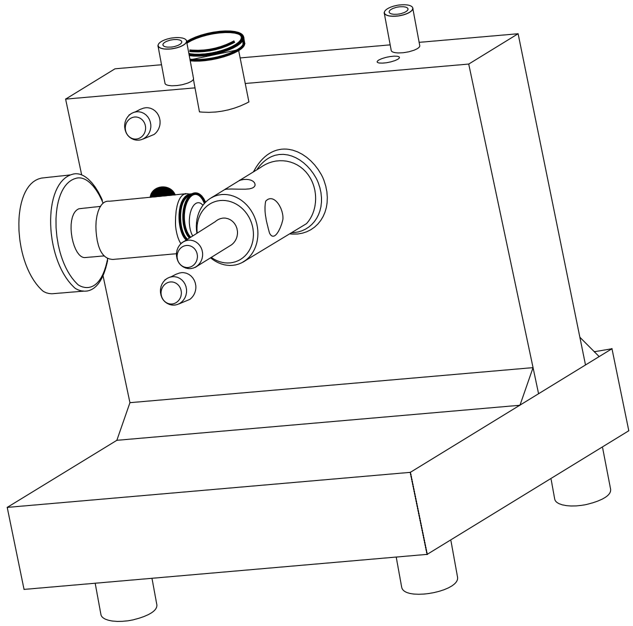


Fig. 15.53 *Rest buttons added to the jig base for the part shown in Fig. 15.48*

Add the jig plate to the jig base. It is located with two dowel pins and then secured by means two socket head screws as shown in Fig. 15.54. The last part is to add the plunger plate as shown in Fig. 15.55 to support the indexing plunger in proper position relative to the locator.

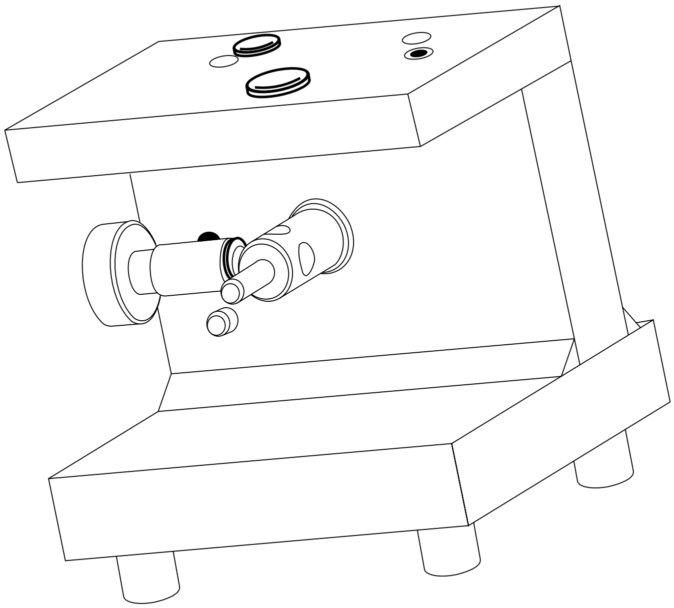


Fig. 15.54 *Jig plate added to the jig base for the part shown in Fig. 15.48*

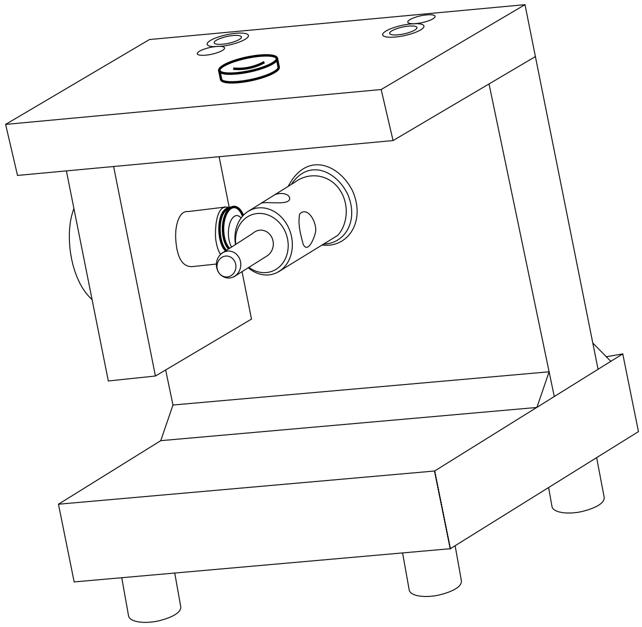


Fig. 15.55 *Support plate for the indexing plunger for the part shown in Fig. 15.48*

The last components to be added are the clamping screw and the C-clamp for holding the part in place as shown in Fig. 15.56. Now the jig is complete. It can be tested with a sample component added as shown in Fig. 15.57.

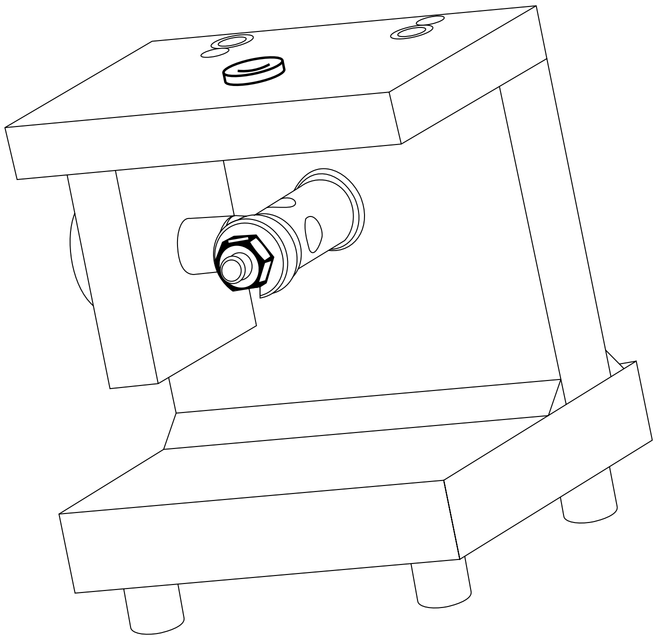


Fig. 15.56 *Clamping arrangement with a C-washer and a nut for the part shown in Fig. 15.48*

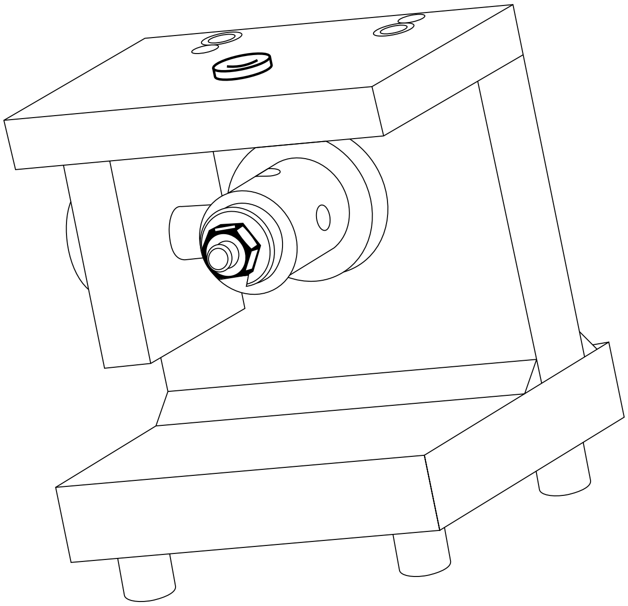


Fig. 15.57 *Completed jig with a sample part loaded in the jig for the part shown in Fig. 15.48*

A few principles are enunciated below, that if adhered to, will ensure a better-designed drill jig.

1. Drill jigs should be as light as possible consistent with rigidity to facilitate handling.
2. A jig, which is not bolted to the machine table, should be provided with feet, preferably four, opposite all surfaces containing jig bushes.
3. Make the jig fool-proof so that the component cannot be loaded in the wrong way.
4. Clearance holes or burr slots should be provided in the jig to allow for the burr formed when the drill breaks through the component and for swarf clearance, particularly from locating faces.
5. Make all component clamping devices as quick acting as possible.
6. Locate clamps so that they will be in the best position to resist the pressure of the cutting tool when at work.
7. Avoid complicated clamping and locating arrangements, which are liable to wear or need constant attention.
8. If possible make all locating points visible to the operator when placing the component in position in the jig so that the component can be seen to be correctly located. The operator should also be able to have an unobstructed view of the clamps.
9. Clamps should be positioned above the points supporting the component, in order to avoid distortion and should be strong enough to hold the component without bending.
10. The process of inserting and withdrawing the component from the jig should be as easy as possible. Ample space should be left between the jig body and the component for hand movements.

15.8 FIXTURES

As explained earlier the function of a fixture is to securely fasten the part to the machine tool table, with accurate location of the part during the machining operation. In addition to the function of holding the work piece, the fixtures also provide for setting the cutting tool for the actual machining operation as shown in

Fig. 15.58. Generally a fixture is supposed to be securely fastened to the machine tool table during the machining process.

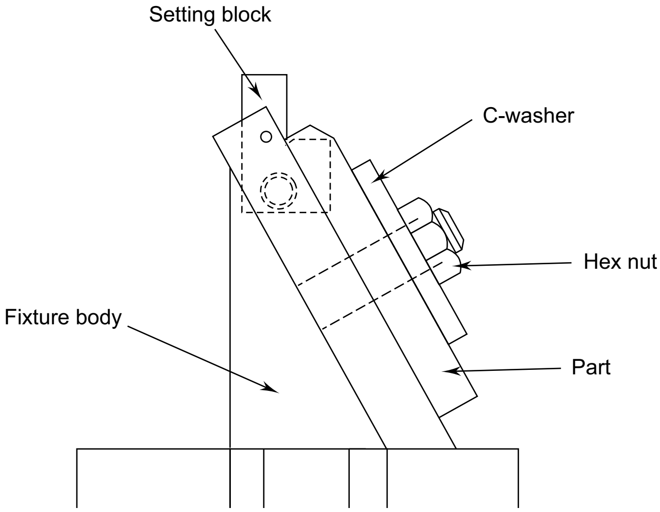


Fig. 15.58 *A typical milling fixture*

Fixtures are widely used in large batch production to ensure the easy setup and achieve the desired accuracy. They can be used in a variety of machine tools such as lathe, milling, grinding, etc. though the milling fixtures are the most widely used in view of the complex requirements for the milling operation. These are called by the name of the machine tool on which it is supposed to be used. For example milling fixture, lathe fixture or broaching fixture.

15.8.1 Vise Fixtures

The simplest type of milling fixture is a vise mounted on the milling machine table. As explained earlier the standard machine vise has all the elements of a typical fixture. Hence standard machine vises can be adapted with special jaws depending upon the contour of the part to be machined. For example for a cylindrical holding surface the plain jaws of a vise can be replaced by V-shaped jaws to transform it as a fixture with minimal change. As shown in Fig. 15.59 the shape of the jaws will be designed to specifically fit the contour of the work piece to be machined. Care also needs to be taken to see that surface of the part is not marred by the excessive force applied during clamping.

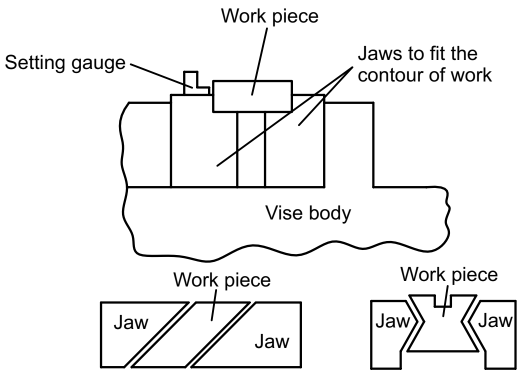


Fig. 15.59 *Different types of vise fixtures*

15.8.2 Milling Fixtures

Milling fixtures are the most common type of fixtures that are in general use today. The reason for this is the geometric complexity of the work pieces that are milled.

However, as the work piece size, shape, or complexity becomes more sophisticated, so too must the fixture.

Similar to a jig, the fixture consists of five main parts, the base, locators, clamps, supports, and a setting block (Fig. 15.58).

Base

It consists of a base plate which has a flat and accurate bottom surface and provides a base for all other components of the fixture to be mounted. The base is provided with slots for the purpose of clamping the fixture to the milling machine table. The bottom surface of the base mates with the milling machine table and thus forms the reference plane for all other components of the fixture. The materials used in its construction are either mild steel or cast iron, depending upon the size and complexity of the part. The major consideration in its choice is the cost and the ability to maintain dimensional accuracy. The type of construction used in the base could be either welding followed by stress relieving to make sure that the base is stress free and consequently distortion free, or sand casting.

Locators

These are similar in principle and design to jigs, which has been covered earlier.

Clamps

The types of clamps used are similar in principle to that used in jigs. However the cutting forces are high in milling, and also because of the nature of interrupted cutting in milling there is the possibility of vibrations. So the clamping design has to take this into consideration.

Supports

These are also similar in principle to jigs, covered earlier.

Setting Block

It is necessary to have a proper location on the fixture for setting the correct location of the milling cutter. It is called setting block or setting gage and is permanently attached to the fixture body away from the work piece as shown in Fig. 15.60. The reference surfaces of the setting block are located at a predetermined distance (usually 3 mm) below the proper cutter setting, as shown in Fig. 15.60. The operator uses a feeler gage to determine when the cutter is in the correct position. This helps to keep these surfaces accurate for the life of the fixture, since the milling cutter will not be directly contacting these surfaces.

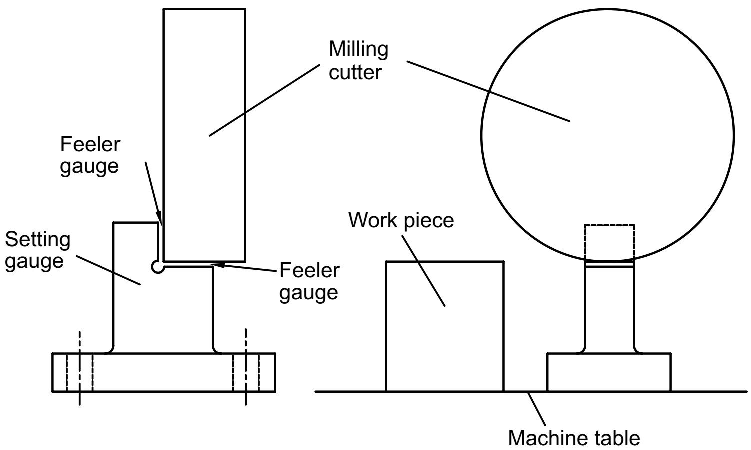


Fig. 15.60 *Setting gage or Setting block*

Before discussing the milling fixture design it is important to know the type of production milling operations for which the fixtures are required. This helps in understanding the design requirements for different situations. The types are shown in Fig. 15.61.

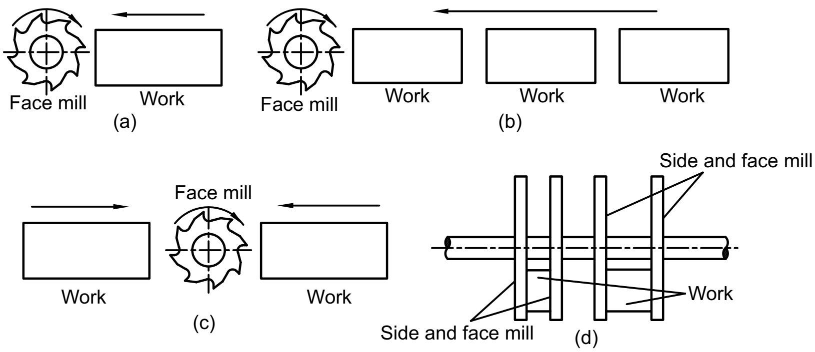


Fig. 15.61 *Different types of production milling operations; (a) simple milling, (b) String milling, (c) Reciprocal milling, (d) Straddle milling*

Simple Milling

This is the traditional milling Fig. 15.61(a) where a single work piece is held in a vise or fixture and fed through the milling cutter.

String Milling

In this, a series of identical small work pieces are mounted in the direction parallel to the table feed movement Fig. 15.61(b) and fed into the milling cutter. This process saves on the over travel allowance but the work pieces should be kept as close together as possible.

Reciprocal Milling

In this there are two fixtures that are mounted on the milling machine table with the milling cutter in the middle Fig. 15.61(c). The operator can unload and load one work piece, while the other is being machined.

Straddle Milling

In this operation two or more milling operations with side and face milling cutters are performed simultaneously Fig. 15.61(d).

Some principles to be considered while designing the milling fixtures are:

- The design should permit as many surfaces of the part to be machined as possible without removing the part.
- Whenever possible, the tool should be changed to suit the part. Moving the part to accommodate one cutter for several operations is not as accurate or as efficient as changing cutters.
- Locators must be designed to resist all tool forces and thrusts. Clamps should not be used to resist tool forces.
- Clearance space or sufficient room must be allotted to provide adequate space to change cutters or to load and unload the part.
- Milling fixtures should be designed and built with a low profile to prevent unnecessary twisting or springing while in operation.

- The entire work piece must be located within the area of support of the fixture. In those cases where this is either impossible or impractical, additional supports, or jacks, must be provided.
- Chip removal and coolant drainage must be considered in the design of the fixture. Sufficient space should be permitted to allow the chips to be easily removed with a brush.
- Set blocks or cutter setting gages must be provided in the fixture design to aid the operator in properly setting up the tool in production.

The design procedure outlined for jigs will hold good here as well. Two examples of milling fixtures are shown in Figs 15.62 and 15.63. Fig. 15.62 shows a simple milling fixture where the base is fixed to the milling machine table by means of the two bolt slots and is provided with the cutter setting block at one end. The work piece is located by means of a round locator and a diamond pin, to fully arrest the six degrees of freedom for the part. Two strap clamps are used to clamp the fixture to the base. Fig. 15.63 shows the string milling fixture to mill slots in five cylindrical work pieces. The work pieces are located by the external cylindrical surface by means of the sliding V-blocks and are clamped by means of a clamping bolt in the end. The setting block is located at one end similar to Fig. 15.62.

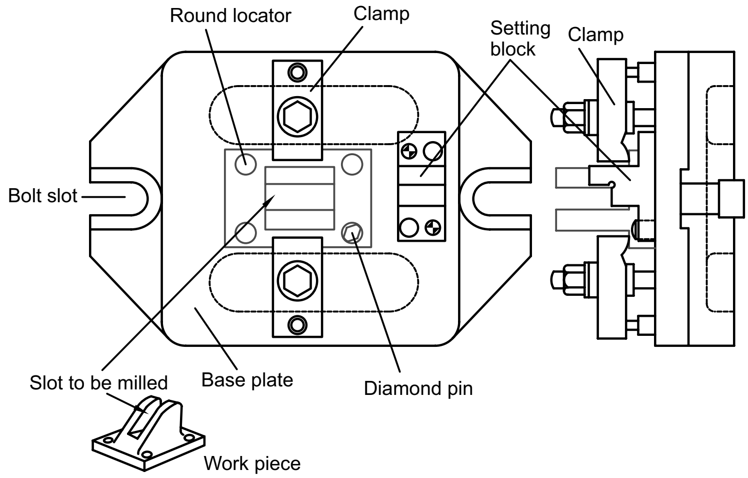


Fig. 15.62 *Milling fixture to mill the slot as a single milling operation*

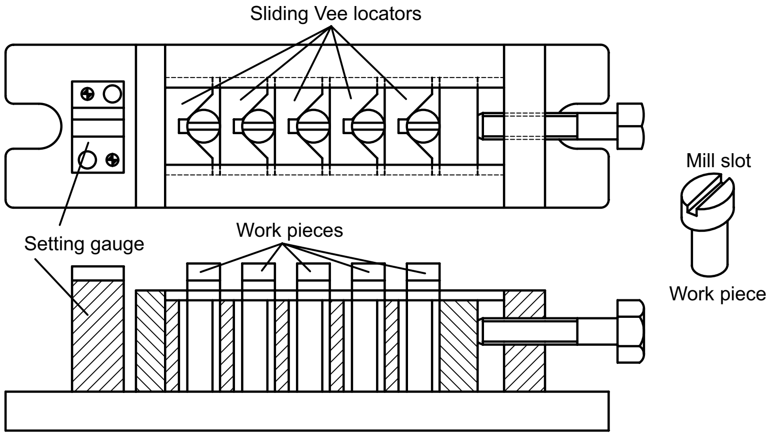


Fig. 15.63 *String milling fixture to mill the slot*

15.8.3 Lathe Fixtures

Majority of the lathe operations are completed using chucks and mandrels since most of them are axi-symmetric in nature. However for some cases that have odd shapes, a face plate will be required. Procedure to be followed is similar to the design of milling fixtures.

In milling, the work piece is stationary and the cutting tool revolves. In turning operations, the work piece revolves and the cutting tool is stationary.

Some principles to be considered while designing lathe fixtures are:

- Tool designer must deal with centrifugal force. The complete fixture must be designed and constructed to resist the effects of the rotational, or centrifugal, forces present in the turning.
- Since lathe fixtures are designed to rotate, they should be as lightweight as possible.
- Lathe fixtures must be balanced. While perfect balance is not normally required for slow-speed turning operations, high rotational speeds require the fixture to be well-balanced.
- Projections and sharp corners should be avoided since these areas will become almost invisible as the tool rotates and they could cause serious injury.
- Parts to be fixtured should, whenever possible, be gripped by their largest diameter, or cross section.
- The part should be positioned in the fixture so that most of the machine operation can be performed in the first fixturing.
- Clamps should be positioned on surfaces, or areas, which are rigid before and after machining.
- As with other fixtures, some means of cutter setting should also be incorporated into the design. However, since the work holder will be rotating, this setting device should be removed.

A typical lathe fixture for the face plate is shown in Fig. 15.64. The work piece has two holes that need to be bored. The slots in the face plate are used for locating the lathe fixture. A counter weight is added to provide for balancing the mass of the work piece that is off centre.

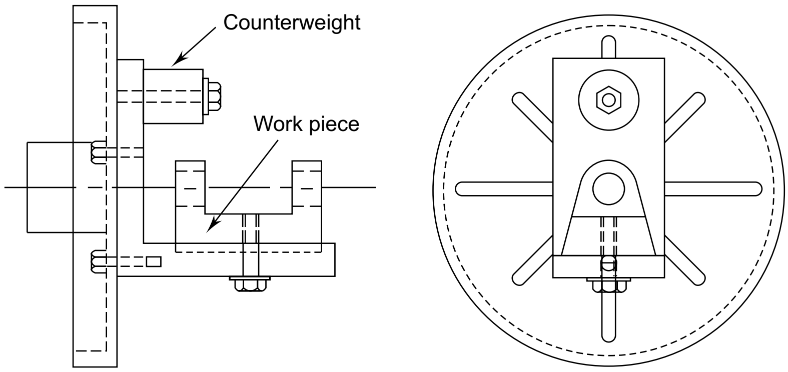


Fig. 15.64 *Lathe fixture for inline boring using a face plate*

15.8.4 Surface Grinding Fixtures

The procedure used to design surface grinding fixtures is similar to that of milling. However since grinding is used as a finishing operation, the accuracies required in manufacturing the grinding fixtures is higher compared to that of milling. The locating methods to be used must be precise and clamping pressure should not affect the work piece in any manner. Also because of the large amount of heat generated during the grinding operation, a large amount of cutting fluid is used. So the grinding fixture should make the necessary provision for draining swarf and the cutting fluid. Whenever possible using magnetic chucks to hold the ferrous work pieces greatly simplifies the grinding operation. Provide coolant containment devices or splash guards to keep the fixture from spilling coolant on the floor around the machine. Also include provisions for rapid wheel dressing and truing in the design of the fixture, if not built into the machine.

15.9 TOOLING ECONOMICS

A systematic estimation of the tool cost is important as that will indicate the utility of the design for the given purpose. Usually the price is determined on the basis of experience. This "calculation" leans more on a rough estimate made by an experienced tool designer based on the process time for each technology and cost of external services. Ideally the tool designer should have sufficient knowledge of the in-house and external costs that are involved in the tool making. Broadly the tool cost can be estimated by breaking down the components of the total cost as:

- Cost of material
- Cost of manufacturing
- Cost of standard parts
- Cost of assembling and try-out

Cost of Material

Material cost estimation is the easiest. Once the designer has the designs of all the components completed in CAD, the system will be able to provide the volume and weight of the part. By knowing the prevailing cost of the raw material, it should be possible to calculate this component.

Cost of Manufacturing

This is one of the most difficult components as it depends upon the cost of machining as well as any other finishing operations that are to be done, such as heat treatment. Since most of the time the tool engineer will be making single pieces, the online estimation calculators will not be of much use. The ability of estimation comes from experience. Often the experience with the making of similar parts comes very handy. Some of the parameters that should be considered in estimating the manufacturing costs are:

- What type of machine is used to manufacture the component?
  - Lathe, horizontal mill, vertical mill, and so on.
  - Cost of machine tool, and tools used
- What are the major dimensions of the component?
  - Size of the machine required
  - The machine overhead cost
- How many machined surfaces are there, and how much material is to be removed?
  - Gives a good estimate of time required for machining
- What tolerance and surface finishes are required?
- What is the labor rate for machinists?

Cost of Standard Parts

This is relatively easy to estimate based on the standard parts and their availability.

Cost of Assembling and Try-Out

The final cost to be considered is that of the assembly of the fixture and its ability to make parts with tolerances. The assembly cost depends upon the number of parts, their complexity, precision, skill and judgement of the operator. Many a times thumb rules are used to assess this component based on the experience of the tool designer.

Once the cost of the tool is identified, then the choice of the desired alternative is based on simple calculations based on the costs that are to be apportioned to the parts made. This can be explained with the help of examples.

Example 15.1

Using the listed alternatives, prepare a comparative analysis for the following tooling problem: A total of 950 flange plates require four holes accurately drilled 90 degrees apart to mate with a connector valve. Which of the listed alternatives is the most economically desirable?

(a) Have a machinist who earns ₹200.00 per hour for layout and drilling each part at a rate of 2 minutes per part.
(b) Use a template jig, capable of producing 50 parts per hour and costing ₹900.00, in the production department, where an operator earns ₹130.00 per hour.
(c) Use a duplex jig, which costs ₹1875.00 and can produce a part every 26 seconds, in the production department, where an operator earns ₹130.00 per hour.

Solution:

Option a: Cost per piece $= \frac{200}{30} =$ ₹6.67

Option b: Production rate $= \frac{60}{1.2} = 50$ per hour

Cost per piece $= \frac{900}{950} + \frac{130}{50} = 0.95 + 2.60 =$ ₹3.55

Option c: Production rate $= \frac{3600}{26} = 138$ per hour

Cost per piece $= \frac{1875}{950} + \frac{130}{138} = 1.97 + 0.94 =$ ₹2.91

Example 15.2

Same as example 1, except a total of 135 flange plates required to be made instead of 950. Which of the listed alternatives is the most economically desirable?

Solution:

Option a: Cost per piece $= \frac{200}{30} =$ ₹6.67

Option b: Production rate $= \frac{60}{1.2} = 50$ per hour

Cost per piece $= \frac{900}{135} + \frac{130}{50} = 6.67 + 2.60 =$ ₹9.27

Option c: Production rate $= \frac{3600}{26} = 138$ per hour

Cost per piece $= \frac{1875}{135} + \frac{130}{138} = 13.89 + 0.94 =$ ₹14.83

SUMMARY

Jigs and fixtures are used to improve the productivity and quality of machined components. Depending upon the production requirements either custom made jigs and fixtures or modular fixtures are used.

- Part surfaces can be divided into location, support and clamping depending upon the function they serve.
- A maximum of six degrees of freedom need to be arrested to make the work piece stable during the machining operation. For this purpose 3-2-1 location principle is used.
- Round locators are more commonly used. They may need to be reduced for radial location in which case they are called diamond pins.
- Rest buttons could be used for supporting the part or jig.
- Clamping force is directed against the locator for stability. A variety of clamps such as strap clamps, cam clamps, and toggle clamps are used in jigs and fixtures. Their selection depends on the part geometry, machining process and the production volume.
- There are a number of types of jigs used in practice such as template jig, plate jig, channel jig, etc. The part geometry, accuracy required and production volumes dictate their individual choice.
- Complete jig design process is like an assembly operation. After making the individual choice of elements that form part of the jig, they are geometrically integrated to form the jig.
- Fixtures are more specialized and varied based on the machining operation involved. They tend to be heavier and clamped to the machine table.

Questions

15.1 Explain the functions served by a fixture.
15.2 What are the applications for which the modular fixtures are used?
15.3 Describe any two types of locators used in jigs.
15.4 Explain the three types of locators used in jigs and fixtures.
15.5 Explain the reasons for using a diamond pin.
15.6 Explain the concept with example of redundant location.
15.7 Why is fool proofing done in fixtures? Give an example.
15.8 Give any four points to be kept in mind to decide the clamping system in jigs and fixtures.
15.9 Explain the use of swing washer for clamping in jigs and fixtures.
15.10 What are the different types of clamps used in jigs and fixtures?
15.11 What is an equalizer?
15.12 What are the different types of jig bushes used?
15.13 Describe the applications of plate jigs.
15.14 Give at least four principles to be considered in the design of milling fixtures.
15.15 Give at least four principles to be considered in the design of lathe fixtures.
15.16 Give a brief note on vise fixtures.
15.17 Discuss the different parts that should form part of a milling fixture.
15.18 What are the requirements for the choice of a fixture base?
15.19 Discuss the construction methods used for jigs and fixtures.

Problems

15.1 Design a drill jig to drill a 1.6 mm hole in the clevis pin shown in Fig. 15.65 made of alloy steel. Assume large volume manufacture. All dimensions are in mm.

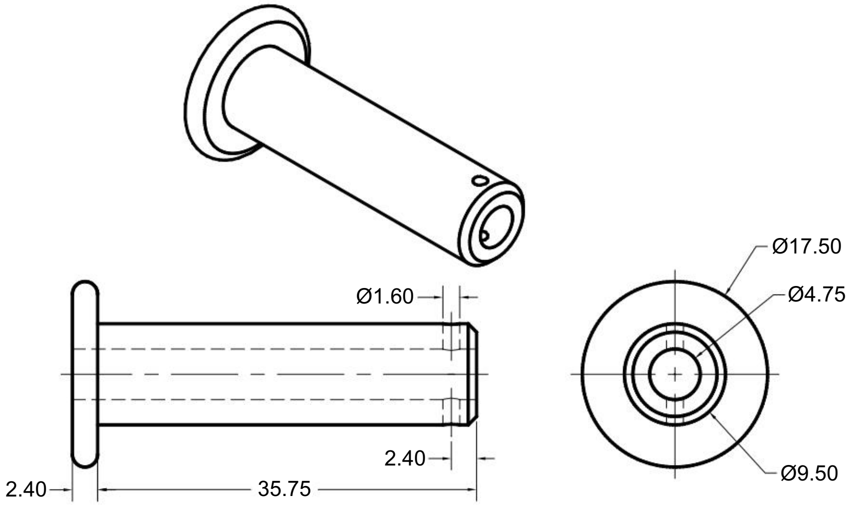


Fig. 15.65 *Clevis Pin*

15.2 Design a drill jig to drill four 6 mm holes in the gland shown in Fig. 15.66 made of steel. Assume large volume production. All dimensions are in mm.

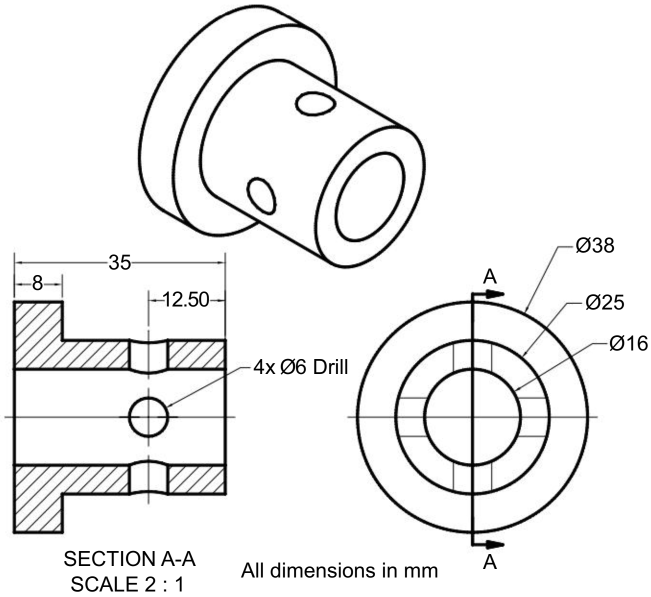


Fig. 15.66 *Gland*

15.3 Design a drill jig to drill a 6 mm hole in the pin fork made of mild steel shown in Fig. 15.67. Assume small volume production. All dimensions are in mm.

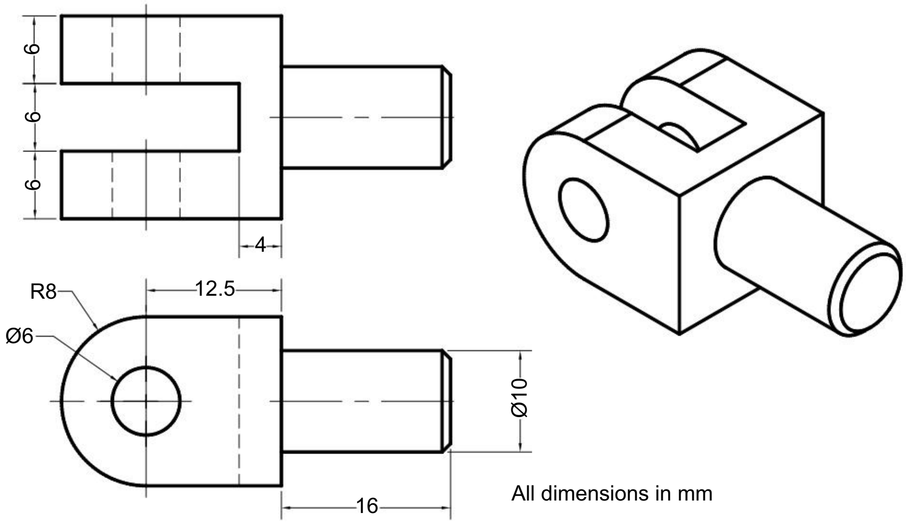


Fig. 15.67 *Pin fork*

15.4 Design a drill jig to drill six 12.5 mm holes in the flange of the adopter made of mild steel as shown in Fig. 15.68. Assume small volume production. All dimensions are in mm.

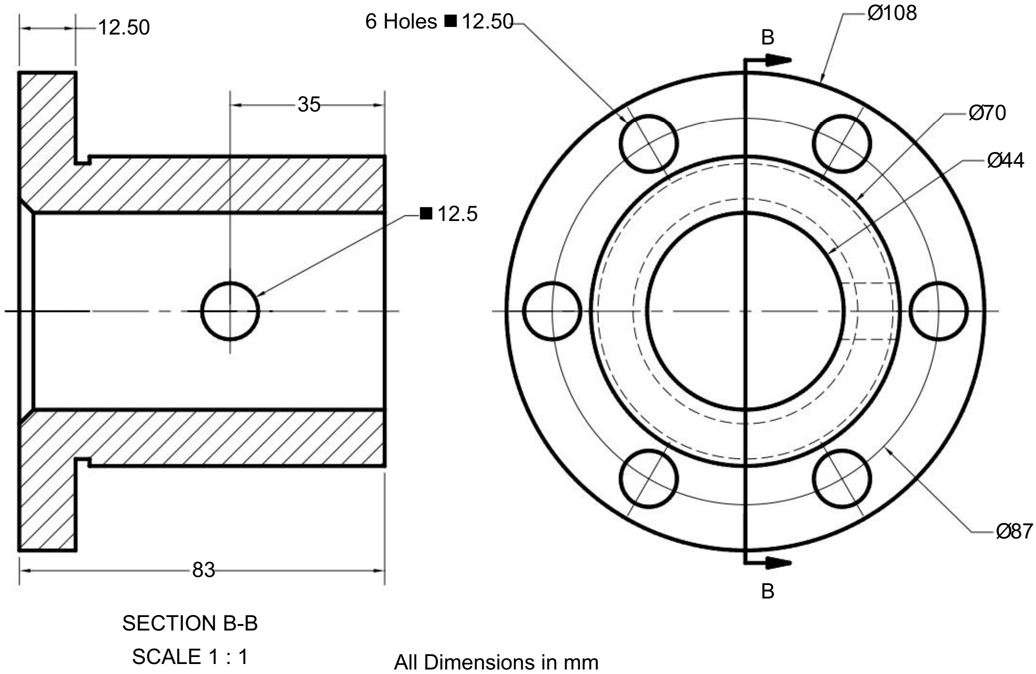


Fig. 15.68 *Adopter flange*

15.5 Five hundred guide plates must be milled to receive a locating block. The tool designer has determined three possible alternatives:
   (a) Have a toolmaker, who earns ₹240.00 per hour, mill the plates at a rate of 25 per hour.
   (b) Use limited tooling that costs ₹1750.00 in the production department. The machine operator in this department, who earns ₹140.00 per hour, can make a part every 1.2 minutes.
   (c) Use a more expensive tool that costs ₹5500.00 but is capable of producing a part every 24 seconds. This would be done in the production department, where a machine operator earns ₹140.00 per hour.

   Which alternative should the tool designer select as the most efficient and economical?

15.6 Using the listed alternatives, prepare a comparative analysis for the following tooling problem: A total of 950 flange plates require four holes accurately drilled 90 degrees apart to mate with a connector valve. Which of the listed alternatives is the most economically desirable?
   (a) Have a machinist who earns ₹200.00 per hour layout and drill each part at a rate of 2 minutes per part.
   (b) Use a template jig, capable of producing 50 parts per hour and costing ₹900.00, in the production department, where an operator earns ₹130.00 per hour.
   (c) Use a duplex jig, which costs ₹1875.00 and can produce a part every 26 seconds, in the production department, where an operator earns ₹130.00 per hour.

Multiple Choice Questions

15.1 A diamond pin is used in conjunction with a round locator for radial location in a jig because
   (a) Diamond is harder material and hence has long life
   (b) Reduces jamming possibility if two round locators are used
   (c) Diamond pin is less expensive to make
   (d) None of the above

15.2 For locating an external cylindrical surface which of the following types of locator is used
   (a) V-block
   (b) Round locator
   (c) Round locator with a hole whose diameter corresponds to the diameter of the cylindrical surface to be located
   (d) Conical locator

15.3 For locating a part in a fixture with an already existing through hole, which of the following type of locators is used
   (a) V-block
   (b) Round locator
   (c) Dowel pin
   (d) Spring loaded plunger

15.4 Redundant locators in fixturing should be avoided because
   (a) It reduces the cost of fixture
   (b) Since the sizes of parts can vary, within their tolerances, the likelihood of all parts resting simultaneously on all surfaces is remote
   (c) It is easier to manufacture
   (d) None of the above

15.5 Fool proofing of a fixture helps in
   (a) The operator will be able to place the part in the fixture in the correct orientation only.
   (b) It reduces the cost of fixture
   (c) It increases the cost of fixture
   (d) It looks aesthetically better

15.6 Principle to be followed while planning the clamping arrangement in a fixture
   (a) Always use simple clamps since complicated ones may lose effectiveness as they wear.
   (b) Rough work pieces call for a longer travel of the clamp in the clamping range,

but clamps may be made to dig into rough surfaces to hold them firmly.
(c) Clamps should not make loading and unloading of the work difficult, nor should they interfere with the use of hoists and lifting devices for heavy work.
(d) All of the above

15.7 The simplest and low cost clamp used in jigs and fixtures is
(a) Strap Clamp (b) Cam Clamp
(c) Toggle Clamp (d) Equalizer

15.8 The main difference between jig and fixture is
(a) Jigs are simpler compared to fixtures
(b) A jig guides the cutting tool during the machining operation
(c) Types of locators used are different
(d) Types of clamping arrangement used is different

15.9 The jig bush most commonly used in jigs is
(a) Liner bush
(b) Headless drill bush
(c) Headed drill bush
(d) Shaped drill bush

15.10 The following statements relate to the choice of jig bushes. Give the correct statement.
(a) Headed jig bushes require less interference (to be assembled into the jig plate) to resist drilling thrust
(b) Less-ductile jig-plate materials require more interference (to be assembled into the jig plate).
(c) Longer jig bushes in thick plates require more interference for them to be assembled into the jig plate
(d) Jig bushes with thinner walls are preferable for accuracy.

15.11 The main disadvantage of a template jig is
(a) Expensive
(b) Complex clamping arrangement
(c) Orientation of the hole pattern to work piece datums may not be as accurate as other types
(d) It is more complex than other types

Answers to MCQs

15.1 (b)	15.2 (a)	15.3 (b)	15.4 (b)	15.5 (a)
15.6 (d)	15.7 (a)	15.8 (b)	15.9 (c)	15.10 (a)
15.11 (c)				

CASE STUDY

WELDING FIXTURE

Figure 1 shows two parts- a flange and a tube made of mild steel that are to be fillet welded along the interface between the two. The welding fixture to be designed will have to hold the two parts in proper alignment for completing the arc welding. In the parts all the machining is completed.

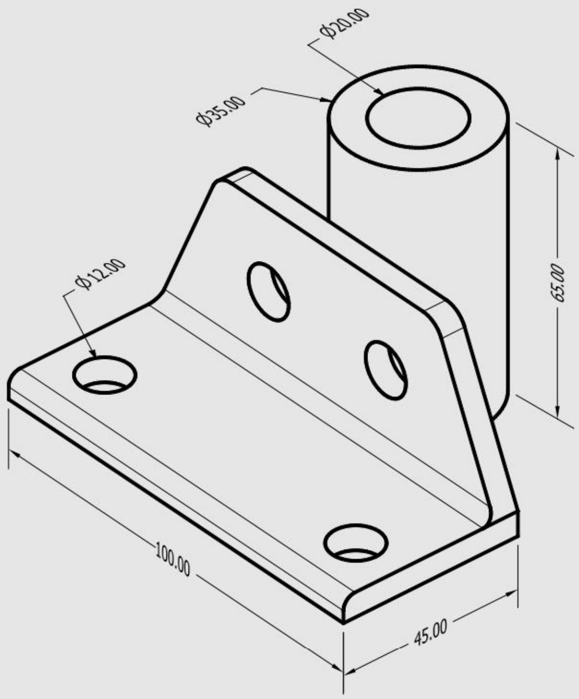


FIG. 1 *Two parts that need to be welded in the exact alignment*

The location element available for the tube is either the central hole or outside surface. Locating an internal surface is easier because of the availability of standard cylindrical locators. In the case of flange the two holes present will act as the locations, and is a typical case of radial location. Since there are two holes, one cylindrical locator will be placed in one whole that will remove 5 degrees of freedom, which leaves one degree of freedom to be removed. This can be done with the help of a diamond locator so that the tolerance on the dimension between the two holes will be accommodated. After identifying all the locators, the next step is to decide their relative placement in space to accommodate the parts. Arrange the three locators on a flat surface as shown in Fig. 2. Allocate appropriate tolerances between the locators relative to each other to satisfy the part tolerances.

Fig. 3 shows all the components along with the parts that are to be welded in the rightful place. Notice that there is no clamping arrangement provided in the fixture.

Based on the welding fixture discussion above, deliberate the following:

- What happens when the diamond locator is replaced by a cylindrical locator?
- Is a clamp necessary for the welding fixture?

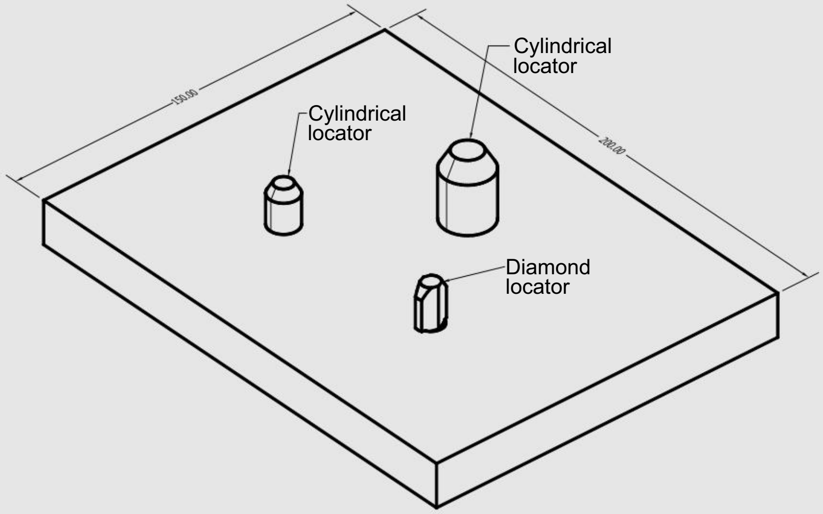


Fig. 2 *The locating elements along with the fixture plate*

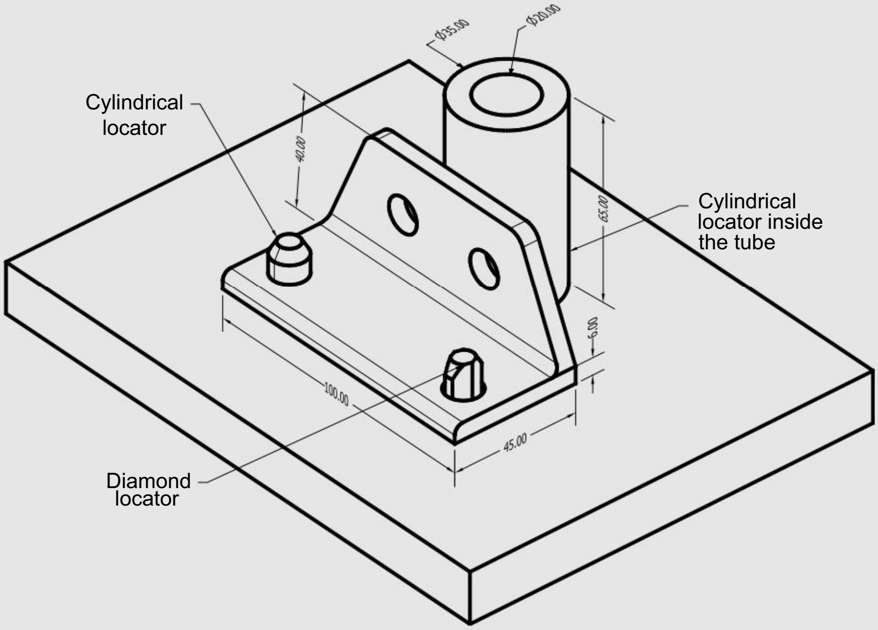


Fig. 3 *Welding fixture complete with the parts that are aligned as required*

CHAPTER 16

Metrology

Objectives

Metrology is the science of measurement of dimensions. This chapter provides a brief review of some of the concepts and instruments required for the measurement of dimensions and tolerances. After completing this chapter, the reader will be able to

- Understand the concept of tolerances, limits and fits as they are relevant for industrial manufacture
- Learn various common instruments used for making linear measurements
- Learn various common instruments used for making angular measurements
- Learn various common methods of measuring threads
- Understand the parameters of surface texture and their measurement
- Understand various gauges used for dimensional acceptance and design simple gauges

16.1 INTRODUCTION

It is important to know that when a dimension is produced by a manufacturing process, it is also important to know that the dimension can be measured accurately enough, such that it can be ascertained that the dimension was actually achieved by the process. The science of measurement is termed as metrology. Measuring can be defined as the determination of a dimension.

In order to make a measurement it is important to have a standard for the dimensions, that is universally applicable. In fact the need for measurement standards has been recognized from the beginning of civilization when construction activities have been attempted. For example when the Pyramids were built, the Egyptians had to come up with a length standard. The standard of measurement then used was known as *the royal cubit,* specified as the length of the Pharaoh's arm. The royal cubit was then subdivided into hands (the width of his hand), and digits (his finger width). This scale was then carefully inscribed on a polished strip of black granite to create a *master standard* because black granite is extremely stable and holds dimensions well.

Most of the progress achieved in industrial manufacturing is credited to the interchangeable manufacture. A Frenchman named La Blanc originally attempted the concept around 1775. However, historically Eli Whitney has been credited with introducing it in the early 1800s when he produced 10,000 muskets using interchangeable parts. The success of interchangeable manufacture depends upon the ability to specify the limits on the dimensions and a way to measure that dimensional acceptance. Only in the twentieth century interchangeable manufacture really flourished with the availability of a large number of dimensional gauging methods at low cost.

Gauging is defined as the acceptability of a given dimension, whether it lies in its specified or allowable limits or not. The cost and ease of manufacture are greatly controlled by the limits that can be imposed on dimensions at the design stage. The limits should be as wide as possible to decrease the cost of manufacture. However from the performance and maintenance point of view, the limits should be as close as possible. The designer therefore has to strike a balance between the ease of manufacture and ease of maintenance depending upon the product requirements.

A few of the terminologies that need to be understood in the learning of metrology are:

Accuracy It is the agreement of the result of a measurement with the true value of the measured quantity. It refers to whether a particular dimension is within its stated size.

Precision It refers to the exactness of the dimension or the repeatability of a measuring process. It depends upon the overall size that is being measured. If a dimension is being measured in m, then a precision of mm may be sufficient. However if the dimension is being measured in mm, then a precision in µm is suitable. But a precision in µm for measurement in m will be meaningless.

Reliability It is the ability to obtain the desired result to the degree of precision required.

Discrimination Discrimination refers to the degree to which a measuring instrument divides the basic unit.

16.1.1 Surface Plate

Surface plate is an important tool for precision measuring applications in a shop. It provides an accurate flat reference plane for many inspection requirements. Metallic surface plates are made of either steel or cast iron castings. The castings will be allowed to age sufficiently and then stress relieved so that all the residual stresses are relieved to ensure the desired degree of flatness. These are then ground and scraped to get the necessary flatness. Then they are mounted on heavy stands. Granite surface plates are superior since it is harder, denser, and impervious to water. It possesses greater temperature stability than the metallic counterparts.

16.2 TOLERANCES, LIMITS AND FITS

The basic or nominal size of a component is chosen as a convenient size based on the design process. However, it is almost impossible to produce any components to the exact dimension through any of the known manufacturing processes. Even if a component is perceived to be made to the exact dimension by the manual processes, the actual measurement with a high resolution measuring device will show that it is not so.

It is therefore customary in engineering practice to allow a tolerance on dimension, which is defined as the permissible deviation from the nominal size. Tolerance on a dimension can be specified in any of the two forms; unilateral or bilateral. In the unilateral tolerance, the variation of the size will be wholly on one side. For example,

$$30^{+0}_{-0.01}$$

is a unilateral tolerance. Here, the nominal dimension 30 is allowed to vary between 30 and 29.99 mm.

In bilateral tolerance, the variation will be to both the sides. For example,

$$30.00 \pm 0.02 \quad \text{or} \quad 30.00^{+0.05}_{-0.10}$$

In bilateral tolerance, the variation of the limits can be uniform as shown in the formal case. The dimension varies from 30.02 to 29.98. Alternatively the allowed deviation can be different as shown in the second case. Here the dimension varies from 30.05 to 29.90.

Sometimes the nominal size may be outside the allowable limits. For example a given dimension is to vary from 29.95 to 29.85. It can be written as

$$29.95^{+0}_{-0.10} \quad \text{or} \quad 30.00^{-0.05}_{-0.15}$$

The second form is preferred since it contains the nominal size as 30.

The dimensioning can also be specified in terms of the limits. For example, in the case of $30.00^{+0.05}_{-0.08}$ the upper limit is 30.05 while the lower limit is 29.92. In drafting practice, it is customary to show the dimensions in any of the forms. Some examples are shown in Fig. 16.1.

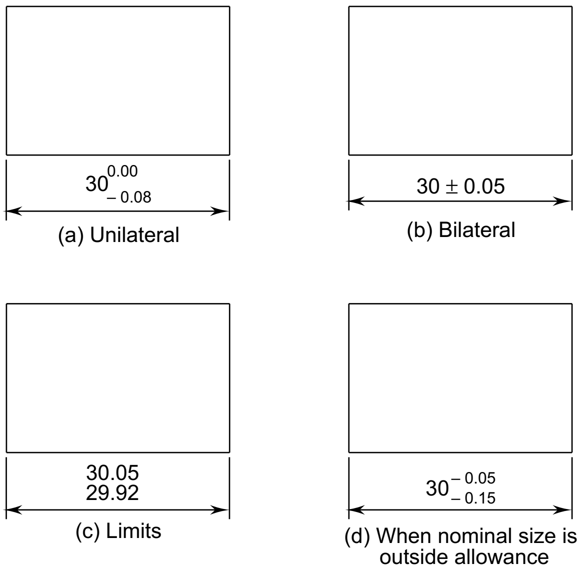


Fig. 16.1 *Typical tolerance specifications*

In engineering when a product is designed it consists of a number of parts and these parts will be mating with each other in some form. In the assembly it is important to consider the type of mating or fit between two parts which will actually define the way the parts will behave during the working of the assembly.

Take for example a shaft and hole, which will have to fit together. In the simplest case if the dimension of the shaft is lower than the dimension of the hole, then there will be clearance. Such a fit is termed as clearance fit. Alternatively, if the dimension of the shaft is more than that of the hole, then it is termed as interference fit. These are illustrated in Fig. 16.2.

The situation will change further by adding the tolerances on the dimensions of the shaft and hole as shown in Fig. 16.3.

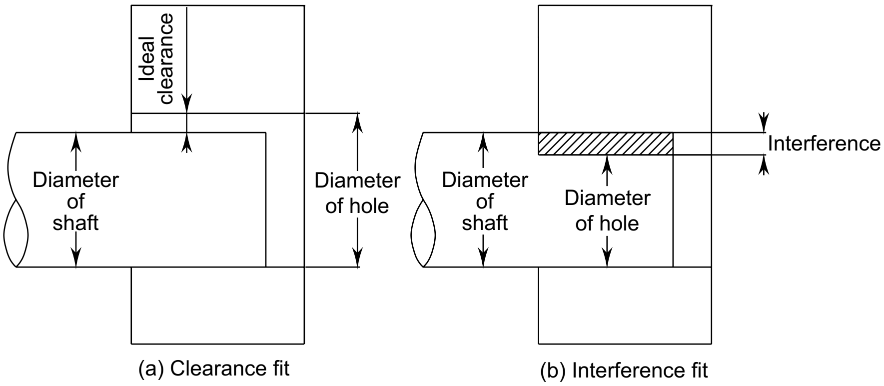


Fig. 16.2 *Typical fits possible in engineering assemblies, (A) Clearance fit, (B) Interference fit*

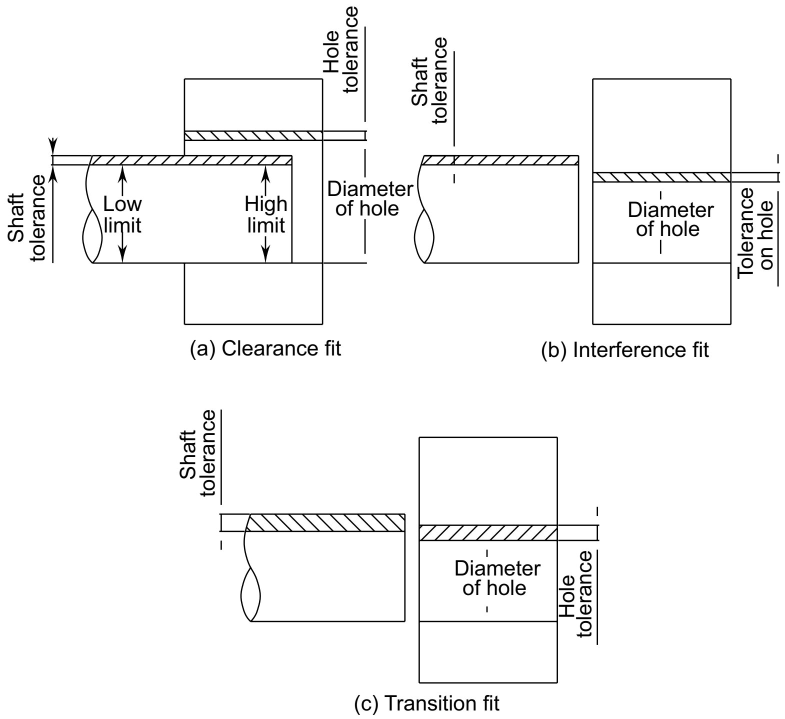


Fig. 16.3 *Typical fits possible in engineering assemblies*

In the case of Fig. 16.3(a), the maximum size of the shaft is smaller than the minimum hole and as a result, there will always be clearance, varying depending upon the actual sizes of the shaft and hole.

Maximum clearance = Maximum limit size of hole – Minimum limit size of shaft

Minimum clearance = Minimum limit size of hole – Maximum limit size of shaft

Such a fit is termed as clearance fit. Similarly in the case of (b), there will be interference for all sizes. However, in (c), depending upon the possibilities of dimensions, at times there will be clearance and other times there will be interference. Such a fit is termed as transition fit.

Maximum clearance = Maximum limit size of hole – Minimum limit size of shaft

Maximum interference = Minimum limit size of hole – Maximum limit size of shaft

Theoretically the above are the three types of fits possible. However, in actual practice, it is necessary to define a large variety of fits within the same type to account for all the possible engineering situations. To this extent ISO, in association with various national standards organisations, has established uniform standards of limits and fits.

In the ISO system of limits and fits, for any given size, a range of tolerances and deviations can be specified with reference to a line of zero deviation called the zero line. The tolerance being a function of the basic size is designated by a number symbol called the tolerance grade. Eighteen standard grades are identified as IT01, IT0, IT1, … IT16. The value of tolerance unit, *i*, is identified as

$$i = 0.45 \times \sqrt[3]{D} + 0.001 \times D$$

Where D is the dimension in mm.

The ISO system identifies the holes by capital letters A, B, C, D, … and the shafts by the lower case letters a, b, c, d, … These letters define the position of the tolerance zone relative to the nominal size. The eighteen grades of tolerances described earlier can be applied to each of these holes and shafts. Though automatic calculations are possible for some of these fits, it is tedious and generally the tables provided in the standards are referred for actual deviations.

For obtaining the required fit, the organisation can choose any one of the following two possible systems:

Hole Basis System

In this system, the nominal size and the limits on the hole are maintained constant and the shaft limits are varied to obtain the requisite fit as shown in Fig. 16.4. For example,

Let the hole size is $30.00^{+0.03}_{-0}$

Shaft of $30.00^{+0.08}_{+0.04}$ gives the interference fit

Shaft of $30.00^{+0.04}_{-0}$ gives the transition fit

Shaft of $30.00^{-0.02}_{-0.08}$ gives the clearance fit

Shaft Basis System

This is the reverse of the above system. In this system the shaft size and limits are maintained constant while varying the limits of hole to obtain any fit.

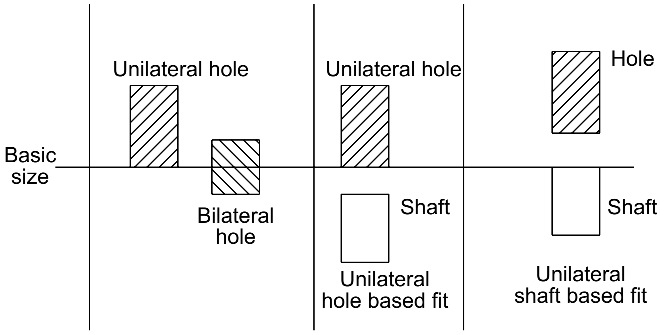


Fig. 16.4 *Typical fits possible in engineering assemblies*

Though there is not much to choose between the two systems, mostly the hole basis system is used because standard tools such as drills and reamers are used for producing the holes whose size is not generally adjustable. Generally it has been found that four classes of holes identified by the type of production method employed will suffice most of the requirements. These are given in Table 16.1. Similarly a total of nine classes of shafts will be able to provide all the necessary fits with the above holes. These are given in Table 16.2.

Table 16.1 **Deviation in µm of Basic Hole system**

Diameter Steps in mm	H6		H7		H8		H11	
	ul	ll	ul	ll	ul	ll	ul	ll
0–3	+6	0	+10	0	+14	0	+60	0
3–6	+8	0	+12	0	+18	0	+75	0
6–10	+9	0	+15	0	+22	0	+90	0
10–14	+11	0	+18	0	+27	0	+110	0
14–18	+11	0	+18	0	+27	0	+110	0
18–24	+13	0	+21	0	+33	0	+130	0
24–30	+13	0	+21	0	+33	0	+130	0
30–40	+16	0	+25	0	+39	0	+160	0
40–50	+16	0	+25	0	+39	0	+160	0
50–65	+19	0	+30	0	+46	0	+190	0
65–80	+19	0	+30	0	+46	0	+190	0
80–100	+22	0	+35	0	+54	0	+220	0
100–120	+22	0	+35	0	+54	0	+220	0
120–140	+25	0	+40	0	+60	0	+250	0
140–160	+25	0	+40	0	+60	0	+250	0
160–180	+25	0	+40	0	+60	0	+250	0
180–200	+29	0	+46	0	+70	0	+290	0
200–225	+29	0	+46	0	+70	0	+290	0
225–250	+29	0	+46	0	+70	0	+290	0

Table 16.2 Deviation in µm of shafts

Diameter Steps in mm	d8		e8		f8		h8		j7	
	ul	ll	ul	ll	ul	ll	ul	Ll	ul	ll
0–3	–20	–34	–14	–28	–6	–20	0	–14	+6	–4
3–6	–30	–48	–20	–38	–10	–28	0	–18	+8	–4
6–10	–40	–62	–25	–47	–13	–35	0	–22	+10	–5
10–14	–50	–77	–32	–59	–16	–43	0	–27	+12	–6
14–18	–50	–77	–32	–59	–16	–43	0	–27	+12	–6
18–24	–65	–98	–40	–73	–20	–53	0	–33	+13	–8
24–30	–65	–98	–40	–73	–20	–53	0	–33	+13	–8
30–40	–80	–119	–50	–89	–25	–64	0	–39	+15	–10
40–50	–80	–119	–50	–89	–25	–64	0	–39	+15	–10
50–65	–100	–146	–60	–106	–30	–76	0	–46	+18	–12
65–80	–100	–146	–60	–106	–30	–76	0	–46	+18	–12
80–100	–120	–174	–72	–126	–36	–90	0	–54	+20	–15
100–120	–120	–174	–72	–126	–36	–90	0	–54	+20	–15
120–140	–145	–208	–85	–148	–43	–106	0	–63	+22	–18
140–160	–145	–208	–85	–148	–43	–106	0	–63	+22	–18
160–180	–145	–208	–85	–148	–43	–106	0	–63	+22	–18
180–200	–170	–242	–100	–172	–50	–122	0	–72	+25	–21
200–225	–170	–242	–100	–172	–50	–122	0	–72	+25	–21
225–250	–170	–242	–100	–172	–50	–122	0	–72	+25	–21

ul = upper limit; ll = lower limit

Typical fits that would be useful for most of the engineering situations are presented in Table 16.3.

Table 16.3 Typical fits that can be obtained in Hole based system

Type of Fit	Shaft Tolerance	Hole Tolerance			
		H7	H8	H9	H11
Clearance	c11				
	d10				
	e9				
	f7				
	g6				
	h6				
Transition	k6				
	n6				
Interference	p6				
	s6				

Selective Assembly

So far, the discussion is centred on the formation of fits based on the full interchangeability between the mating parts. Sometimes, it becomes necessary to produce assemblies with tighter fits but at lower cost. The cost of production increases with a decrease in the tolerance, which is necessary for tight fits. In such cases selective assembly is used.

In selective assembly all the parts produced are measured and graded into a range of dimensions within the tolerance groups. This procedure is followed for the holes as well as shafts as follows:

Shafts	A (Small)	B (Medium)	C (Large)
Holes	A (Small)	B (Medium)	C (Large)

Here the small, medium and large refer to the size range of the parts within the total tolerance zone. By mating the shafts from group A with that of holes from group A ensures that the fit is much tighter than that by choosing any hole from A, B or C with any of the shafts from A, B or C. This is convenient for reduction of the cost of production, but creates problems for maintaining such assemblies in terms of the replacement parts.

16.3 LINEAR MEASUREMENT

The standard unit of length as per the SI units is the metre. In 1791 the original standard for meter was decided as equal to 10^{-7} of the meridian through Paris from pole to the equator. Because of errors noted later, in 1889 a new international prototype was made of 10 percent iridium alloy of platinum to within 0.0001 to be measured at 0°C. This was again modified in 1927 as the distance at 0°C between the axes of the two central lines marked on the bar of platinum-iridium kept at the BIPM (Bureau International des Poids et Mesures, France), and declared Prototype of the meter by the 1st CGPM (Conferénce Générale des Poids et Mesures - General Conference on Weights and Measures).

The 1889 definition of the meter was replaced by the CGPM in 1960 using a definition based upon a wavelength of krypton-86 radiation. The metre is defined as 1 650 763.73 times the wavelength in vacuum of transition between energy levels $2p_{10}$ and $5d_5$ of krypton-86 atoms excited at triple point of nitrogen (–210°C). This definition was adopted in order to reduce the uncertainty with which the meter may be realized.

To further reduce the uncertainty, in 1983 the CGPM replaced this latter definition by the following definition, "The meter is the length of the path travelled by light in vacuum during a time interval of 1/299 792 458 of a second."

Though this is the actual standard, the length measurements in the industry will be done and maintained using the instruments available in the shop. Some of the instruments used for this purpose are:

- Rules
- Vernier
- Micro meter
- Height gauge
- Bore gauge
- Dial indicator
- Slip gauges or Gauge blocks

Rules

The terms scale and rule are used in the machine shop interchangeably, which is incorrect. Scale is graduated in imaginary units, while rule has graduations that represent real units of length such as mm or inch. The maximum discrimination on the rule helps in identifying the smallest possible measurement. A good quality steel rule is engraved with graduated divisions.

There are a variety of rules that are available for specific applications such as flexible rules (useful for checking lengths on relatively gentle curving surfaces), narrow rules or a standard hook rule that has a hook attached at one end to help in reaching a slot or depth for the purpose of measurement.

Vernier

The vernier is the most common length measurement used in the shop. It makes use of a main scale and auxiliary (vernier) scale as shown in Fig. 16.5. The main scale is graduated to convenient divisions, which can be easily read. For example in Fig. 16.5 shows the main scale units, further divided into smaller divisions of 0.025 units each. The auxiliary scale divides this smallest unit of 0.025 units into 25 divisions as shown in Fig. 16.5. Thus

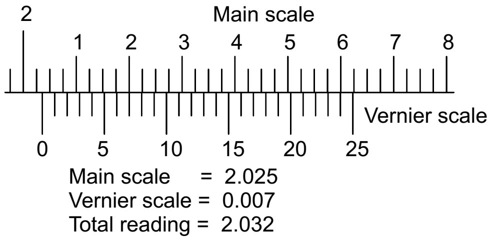


Fig. 16.5 *Vernier principle*

$$\text{Least count} = \frac{0.025}{25} = 0.001$$

To measure the value, note down the main scale division and add to it the value measured by the vernier scale or auxiliary scale. The measurement of the vernier is the divisions that are matching or aligned.

In Fig. 16.5 it is noted that the main scale division is 2.025 while the auxiliary scale reads 7 divisions. Hence,

$$\text{The reading} = 2.025 + 7 \times 0.001 = 2.032$$

The vernier calliper is provided with jaws which can do the end to end measurement on a component relatively easily and accurately. In addition, some typical vernier callipers are also provided with a set of auxiliary jaws for the purpose of measuring internal dimensions as well as the heights and depths.

With the advent of electronic measuring systems, many of the new measuring units are provided with direct digital read-outs to reduce the possible inaccuracies likely in reading the mechanical scales and the associated computing to obtain the final results. An example is shown in Fig. 16.6 of a digital vernier height gauge that has a single measuring jaw to measure the height from the base.

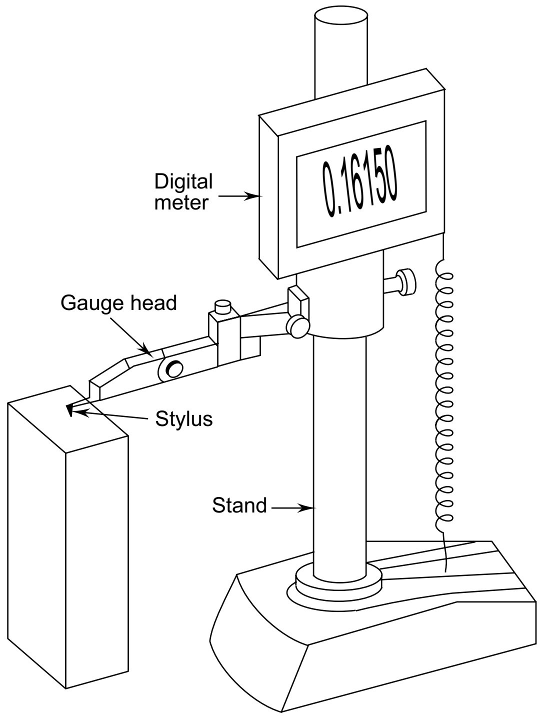


Fig. 16.6 *Digital Vernier height gauge*

Micro Meter

The micro meter is another end to end measuring device used for more accurate measurements. In this the auxiliary scale is made on the thimble, which rotates about the main scale as shown in Fig. 16.7. The thimble rotates about the main scale unlike the linear motion of the vernier scale there by improving the measurement accuracy.

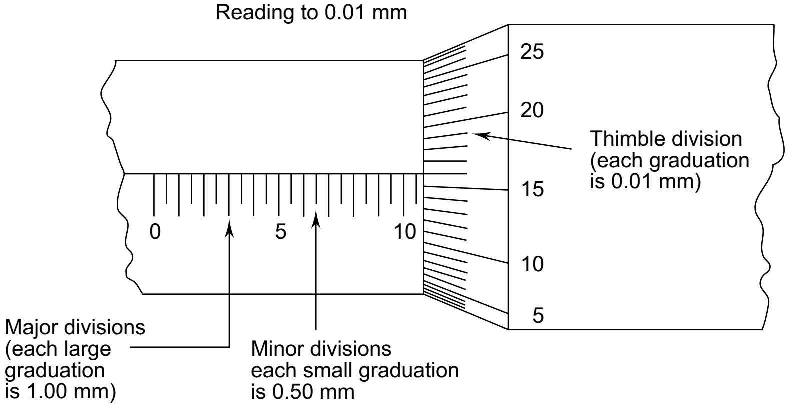


Fig. 16.7 *Micro meter principle*

In this case, the smallest division is on the main scale is 0.5 mm. The thimble is divided into 50 divisions. Thus

$$\text{Least count} = \frac{0.5}{50} = 0.01\ \text{mm}$$

The reading = 10.50 + 16 × 0.01 = 10.66 mm

The constructional features of an outside micro meter are shown in Fig. 16.8. It is provided with a fixed anvil and a rotating spindle between which the part to be measured is placed. The spindle and the anvil are connected by means of a U-shaped frame. The gap indicates the maximum measurement that can be made using the micro meter. The spindle can be rotated by means of the thimble till it touches the part surface. The ratchet at the end ensures that only the spindle does not apply too much pressure on the part.

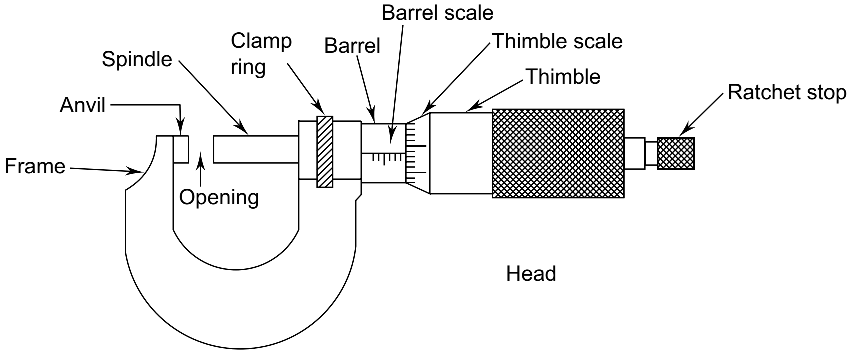


Fig. 16.8 *Construction features of Micro meter*

A number of variations are available in the micro meters to suit the measuring function being contemplated. The following are a few of the varieties:

- inside micro meters (Fig. 16.9) for measuring the inside diameters
- depth micro meters (Fig. 16.10) for measuring the depth of slots and blind holes
- screw thread micro meters (Fig. 16.11) for measuring the screw thread parameters discussed later.
- V-anvil micro meter (Fig. 16.12) that can be used for measuring the diameter of objects that have odd number of symmetrical or evenly spaced features, for example for the diameter of a 3-fluted end mill.

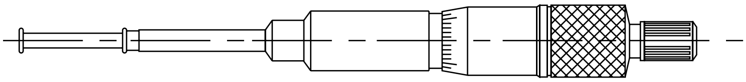


Fig. 16.9 *Inside Micro meter*

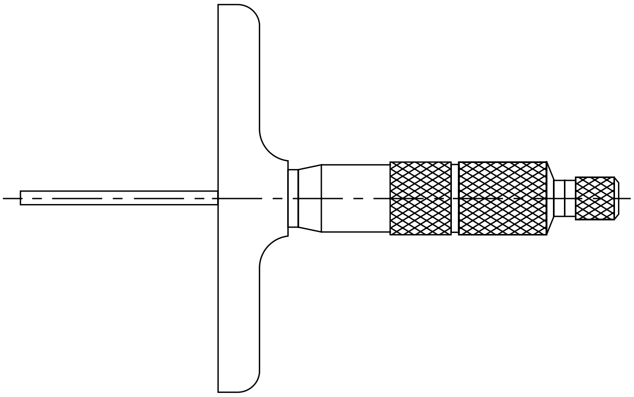


Fig. 16.10 *Micro meter to measure the depth*

Fig. 16.11 *Screw thread Micro meter*

Fig. 16.12 *V-anvil Micro meter*

Height Gauge

Height gages are some of the most important instruments used in the shop for precision layout purposes. These have a flat base which will be kept on a surface plate while the tip can be moved to measure the actual height of the part from its base as shown in Fig. 16.6.

Bore Gauge

These are used for measuring bores of different sizes ranging from small to large sizes as shown in Fig. 16.13. These are provided with various extension arms that can be added for different sizes. An indicator will be attached at the top, which will provide the deviation from the value of the extension rod.

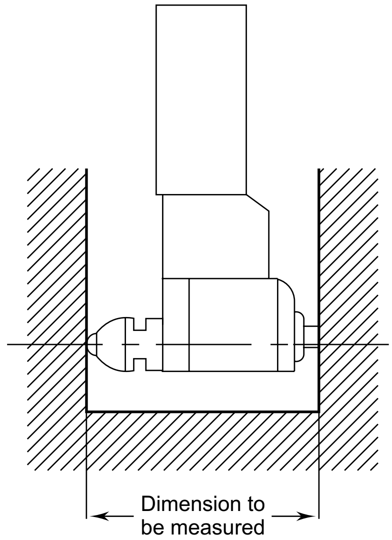


Fig. 16.13 *Bore gauge measuring an inside dimension*

Dial Indicator

This is another of the more common measuring instruments in the arsenal of a machinist. These consist of a spring loaded plunger, A as shown in Fig. 16.14 whose tip is used for measuring or gauging a surface. The movement of the plunger A is magnified through the intermediate gearing B and C to show with the pointer. The typical least count that can be obtained with suitable gearing in dial indicators is 0.01 mm to 0.001 mm.

It is possible to use the dial indicator as a comparator by mounting it on a stand at any suitable height. The use of dial indicators in machine tool alignment is discussed in Chapter 13.

Slip Gauges or Gauge Blocks

Slip gauges are used in the manufacturing shops as length standards. They are not to be used for regular and continuous measurement. Slip gauges are rectangular blocks with the thickness representing the dimension of the block. The cross-section of the block is usually 32 mm × 9 mm. The slip gauges are hardened and finished to size. The measuring surfaces of the gauge blocks are finished to a very high degree of finish, flatness and accuracy. When two slip gauges are 'wrung' together, considerable force is required to separate them in view of the high finish and flatness of the surfaces in contact.

Slip gauges come in a number of grades of accuracies depending upon the measurement accuracy requirement. They come in sets with different number of pieces in a given set to suit the requirements of measurements. A typical set consisting of 88 pieces for metric units is shown in Table 16.4.

To build any given dimension, it is necessary to identify a set of blocks, which are to be put together. One of the principles to be remembered is that the number of blocks used should always be the smallest.

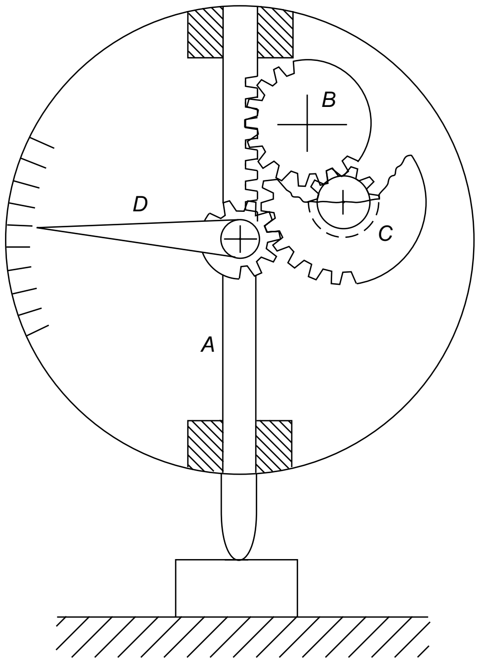


Fig. 16.14 *Principle of a dial indicator*

Table 16.4 Metric slip gauge set

Slip Gauge Size or Range, mm	Increment, mm	Number of Pieces
1.005	—	1
1.001 to 1.009	0.001	9
1.010 to 1.490	0.010	49
0.500 to 9.500	0.500	19
10 to 100	10.000	10

To assemble a dimension of 62.225 mm the following combination will be used:

$$\begin{array}{r} 1.005 \\ +\ \ 1.220 \\ +\ 60.000 \\ \hline 62.225 \\ \hline \end{array}$$

The principle to be employed in the selection of the slip gauge to be used is first choosing a block that satisfies the lowest decimal point. Continue with the next by moving towards the left of the given dimension. A few examples will help illustrate the point.

To assemble a dimension of 74.315 mm the following combination will be used:

$$\begin{array}{r} 1.005 \\ +\ \ 1.310 \\ +\ \ 2.000 \\ +\ 70.000 \\ \hline 74.315 \\ \hline \end{array}$$

To assemble a dimension of 37.936 mm the following combination will be used:

$$\begin{array}{r} 1.006 \\ +\ \ 1.430 \\ +\ \ 5.500 \\ +\ 30.000 \\ \hline 37.936 \\ \hline \end{array}$$

Comparators

Besides the length measuring devices described above, comparator is another form of linear measuring method, which is quick and more convenient for checking a large number of identical dimensions. The basic concept used is that the comparator is set for any given dimension. During the measurement a comparator is able to give the deviation of the dimension from the set dimension. This cannot be used as an absolute measuring device but can only compare two dimensions.

Initially the comparator is set with the help of a known dimension; e.g. a set of slip gauges as shown in Fig. 16.15(a). Then the indicator reading is adjusted to zero as shown. When the part to be measured is kept under the pointer, then the comparator will display the deviation of this dimension either in the + or – side of the set dimension as shown in fig. 16.15(b).

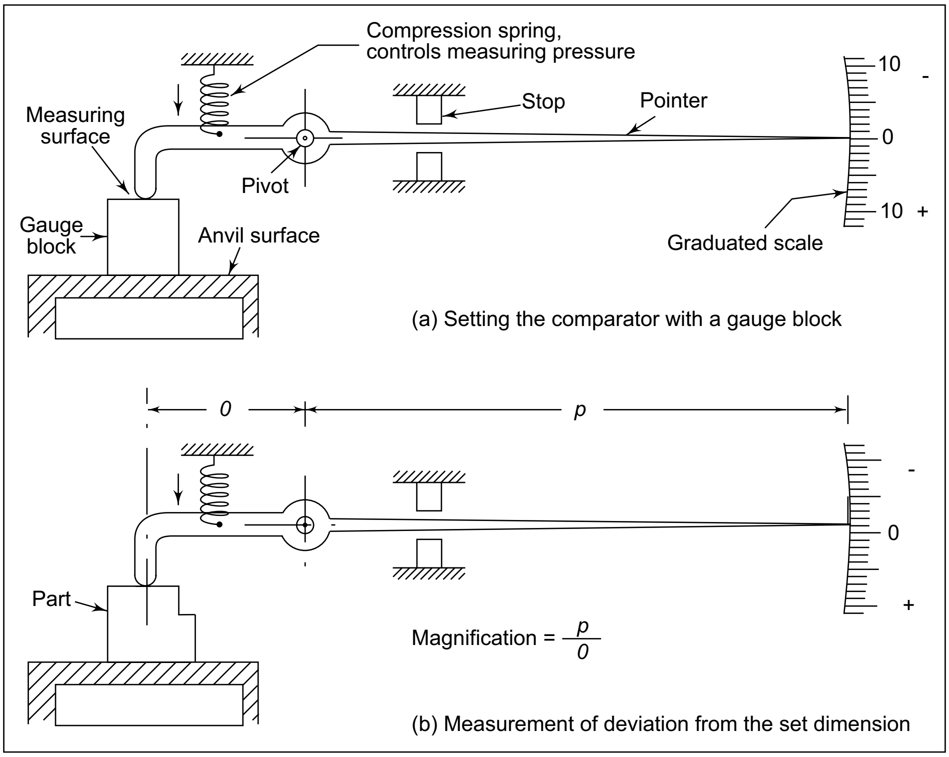


Fig. 16.15 *Principle of a comparator*

The reliability of the dimension measured using a comparator is high because of the inherent accuracy built into the magnifying system, and that the pressure used in the measurement is constant. To magnify the deviation from the set dimension a number of principles are used such as mechanical, optical, pneumatic and electrical, the details of which can be found in books on metrology.

16.4 ANGULAR MEASUREMENT

This is another important element in measuring. This involves the measurement of angles of tapers and similar surfaces. The most common angular measuring tools are:

- Bevel protractor
- Sine bar
- Clinometer

Bevel Protractor

The bevel protractor is part of the machinist's combination square as shown in Fig. 16.16. The protractor can be moved on the steel rule along the central groove and can be locked in any position required. The flat base of the protractor helps in setting it firmly on the work piece and then by rotating the rule, it is possible to measure the angle. It will typically have a discrimination of one degree.

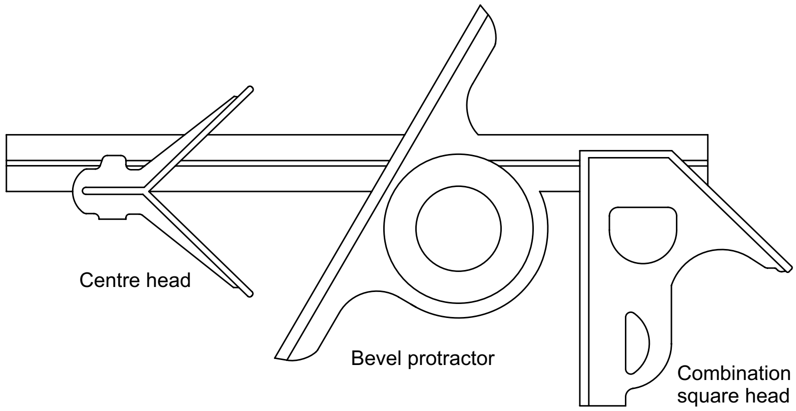


Fig. 16.16 *Combination square that includes a bevel protractor*

Sine Bar

The most common one used is the sine bar. A sine bar is bar consisting of two precision ground rollers at the two ends whose centre distance is fixed such as 125 mm or 250 mm as shown in Fig. 16.17.

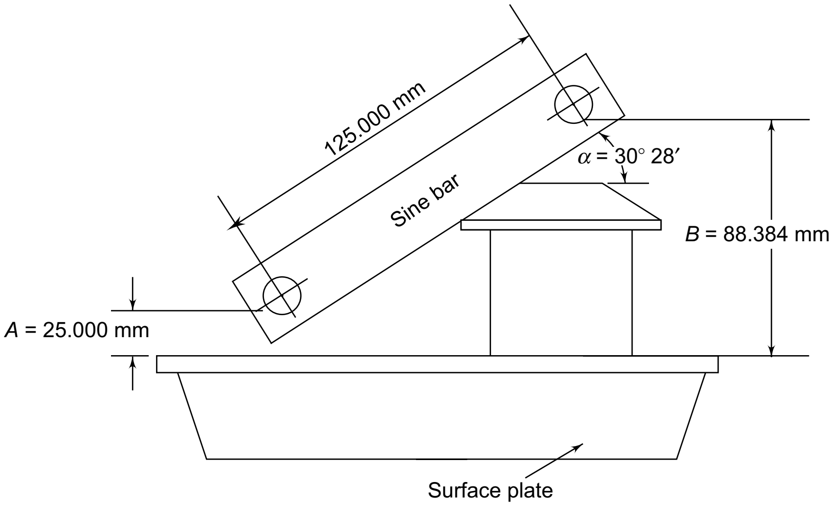


Fig. 16.17 *Principle of angular measurement with a sine bar*

The sine bar surface can be aligned with the surface to be measured such that the distance between the rollers forms the hypotenuse. The height differential of the two rollers in alignment with the work piece is shown in Fig. 16.17. The angle can be calculated using the sine formula. The angle, α in the Fig. 16.17 is given by

$$\sin\alpha = \frac{88.384 - 25}{125} = 0.5071$$

or $$\alpha = 30^\circ\ 28'$$

The way the sine bar is to be used depends upon the nature of the taper angle to be measured. For example, in Fig. 16.17 the sine bar is aligned with the work piece kept on the surface plate and the centre heights are measured with the help of a height gauge. Alternatively as shown in Fig. 16.18, the part is aligned on the sine bar with one roller on the surface plate and the other roller on a set of gauge blocks till the work piece surface is horizontal. This can be ensured with the help of a dial indicator as shown.

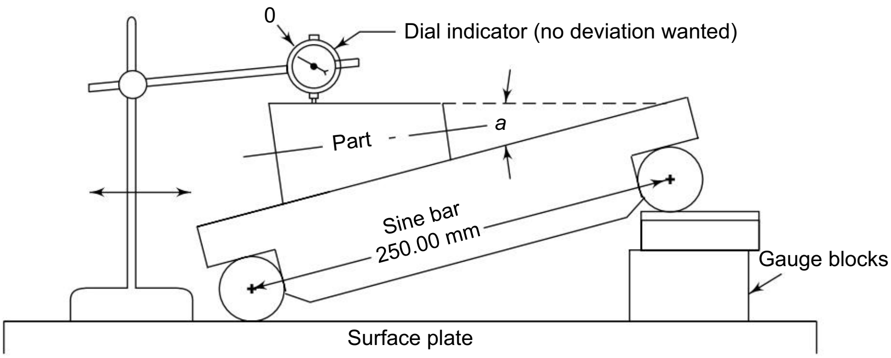


Fig. 16.18 *Principle of angular measurement with a sine bar and dial indicator*

Clinometer

Another angular measure instrument used is the clinometer, which is a clever adoption of the spirit level for measuring the inclination of a surface relative to the horizontal plane by levelling the spirit level. The main measuring unit is the spirit level that is mounted on a rotary member that is pivoted as shown Fig. 16.19(a). One face of the housing forms the base of the instrument. A circular scale is provided in the housing which is used to measure the angle of inclination of the rotary member with the spirit level relative to the horizontal plane. The bubble of the spirit level is in its centre position, when the clinometer is placed on a horizontal surface and the scale of the rotatable disc is at zero position. The base of the instrument is placed on the surface whose angle is to be measured. Since the surface is inclined, the bubble deviates from the centre. The bubble is brought to the centre by rotating the rotary member with the spirit level till the bubble is at the centre as shown in Fig. 16.19(b). The angle of rotation is then noted on the circular scale against an index.

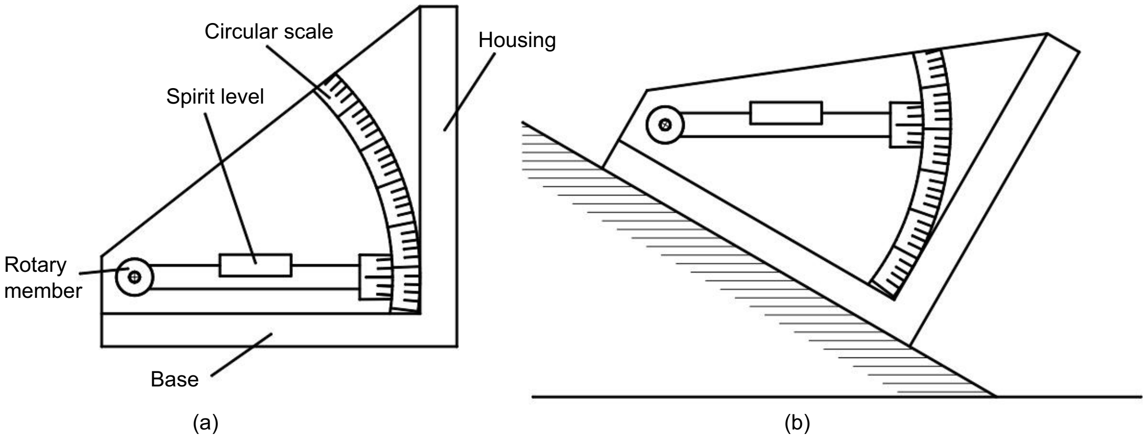


Fig. 16.19 *Clinometer*

Variations in clinometers is essentially in the method of measurement of the angle and the measuring accuracy. Measuring method could be using a vernier, a micro meter or a dial. This can be used to measure the angle of a surface from the horizontal plane or the angular separation of two surfaces by measuring each one of them with reference to the horizontal plane and then getting the result as the difference of the two readings.

16.5 THREAD MEASUREMENT

The screw thread nomenclature is shown in Fig. 16.20. Threads are normally specified by the major diameter. Though there are a large variety of threads used in engineering, the most common thread encountered is the metric V-thread shown in Fig. 16.20.

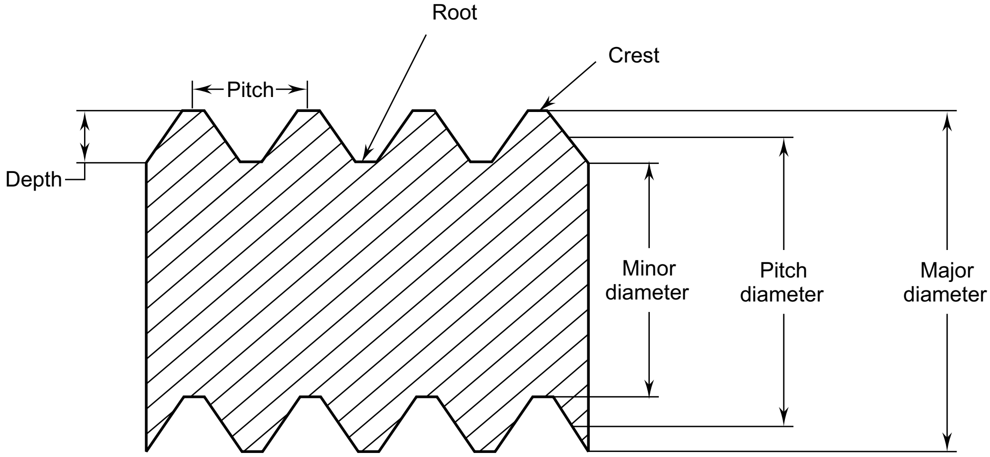


Fig. 16.20 *Nomenclature of external screw thread*

The parameters that are normally measured are:

- Major diameter
  - Micro meter
- Pitch diameter
  - Screw thread micro meter
  - Wire method
- Pitch
  - Screw pitch gauge
  - Pitch measuring machine
- Thread form
  - Optical projector

Major diameter can be measured using an external micro meter with the anvil and spindle touching the thread surface. Care has to be taken to see that thread surface is touched with minimum pressure. Since the crest surface is very small that will be in contact with the micro meter, any excess pressure is likely to deform the thread and cause error in measurement.

To measure the pitch diameter, screw thread micro meters have a specially designed spindle and anvil as shown in Fig. 16.11. The end of the spindle of this type of micro meter is pointed to form a 60° cone, and the anvil has the form of a vee to fit over the thread. The sharp tip of the spindle is ground off to make sure that only the pitch diameter is measured rather than the root or minor diameter. The swivel of the V-shaped anvil compensates for the thread lead. A screw thread micro meter is generally designed to measure threads within a narrow range of pitches and it is marked on the micro meter.

16.5.1 Measurement Over Wires

Another method used to measure precision threads is using the wires. Small hardened steel wires are of precision size and accurately ground. Two wires are placed over the thread grooves in the opposed directions of the thread as shown in Fig. 16.21. Then the size is measured using a micro meter with the spindle and anvil touching the wire surface. Then the pitch diameter is calculated as

$$\text{Pitch diameter, } D_P = M + P$$

Where P = 0.866p – G for metric threads
p = pitch of the thread
G = diameter of the wire used

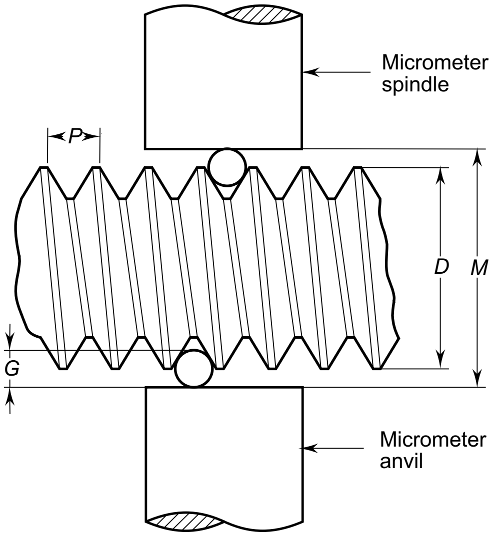


Fig. 16.21 *Measuring screw thread using a micro meter*

16.5.2 Three-Wire System

In the three-wire method, two of the wires are placed in the thread groove on one side and one wire is diametrically opposed as shown in Fig. 16.22. The measurement is done using a micro meter or preferably a diameter measuring machine for more accuracy. The accuracy of the pitch diameter measurement depends on the measuring instrument, the wire diameter, and the contact force applied.

$$\text{Pitch diameter, } D_P = M - \left[G(1 + \operatorname{cosec} \theta) - \frac{p \cot \theta}{2} \right]$$

Where P = 0.866p – G for metric threads
p = pitch of the thread
G = diameter of the wire used
θ = thread angle

Ideally the wire diameter G is to be selected such that it makes contact with the flanks of the thread on the pitch line. As a result for each pitch, there is a best size.

$$\text{Best wire diameter, } G = \frac{p}{2} \sec\left(\frac{\theta}{2}\right)$$

The method normally used for quick checking threads, particularly on production lines is the use of 'Thread Ring Gage' for external threads and 'Thread Plug Gage' for internal threads. When using a thread ring or plug gage, a "GO" and "NO GO" gage is used. This type of gage requires no extra training or skill to use.

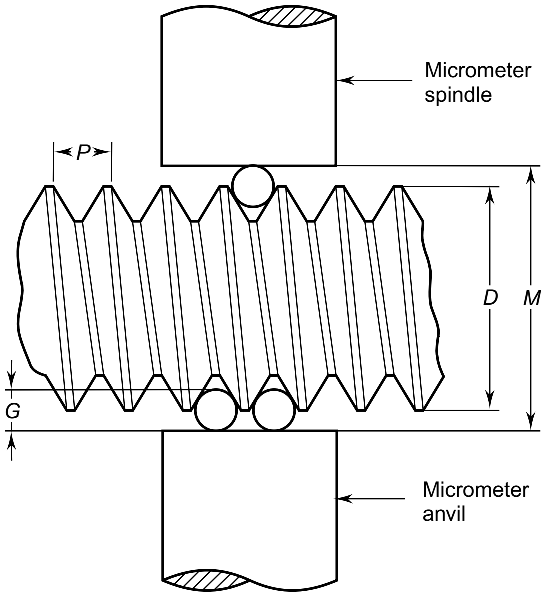


Fig. 16.22 *Nomenclature of external screw thread*

16.6 SURFACE TEXTURE

It is a well-known fact that the actual surface after machining may look smooth but in reality it is not. All surfaces have some degree of roughness and inaccuracy. When magnified the surface of a part resembles a series of jagged peaks and valleys. Hence, the surface finish of any given part is described in terms of average heights and depths of these "peaks and valleys" on the surface of the work piece.

Typical surface texture characterisation is shown in Fig. 16.23. These are some of the parameters that need to be understood to fully characterise the surface texture.

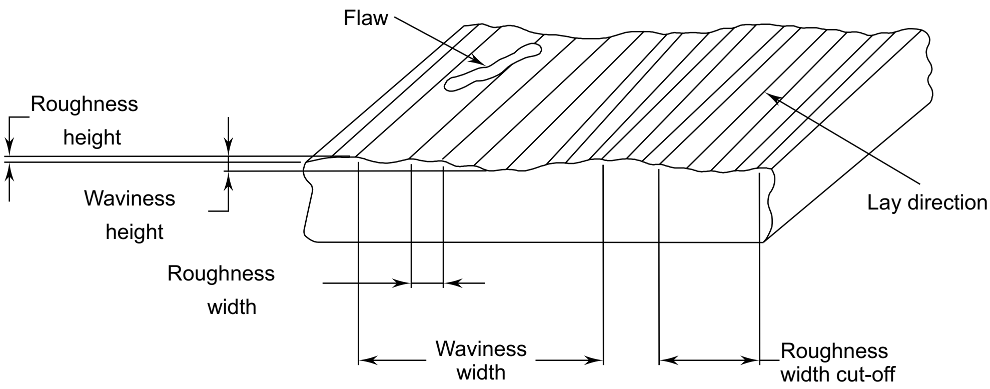


Fig. 16.23 *Elements of surface texture*

Roughness Height

This is the parameter with which generally the surface finish is indicated. It is specified either as arithmetic average value or the root mean square value.

Roughness Width

It is the distance parallel to the nominal part surface, within which the peaks and valleys are shown, which constitute the predominant pattern of the roughness.

Roughness Width Cut-off

This is the maximum width of the surface that is included in the calculation of the roughness height.

Arithmetical Average

An imaginary centre line is imposed at a point representing the average midpoint or centre of the distance between the peaks and the valleys of the surface profile as shown in Fig. 16.24. These are measured for a specified area, the figures are added together and the total is then divided by the number of measurements taken to obtain the mean or arithmetical average (AA). It is also sometimes called as the centre line average or CLA value. This in equation form is given by

$$R_a = \frac{1}{L}\int_0^L |y(x)|\,dx \cong \frac{1}{N}\sum |y_i|$$

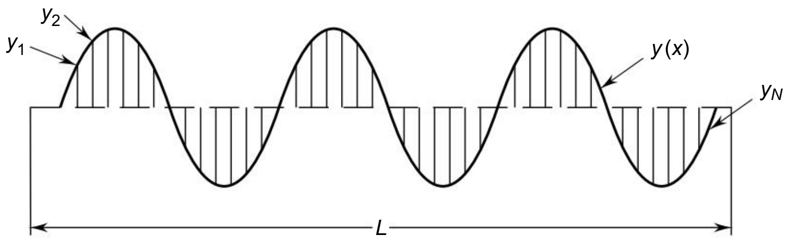


Fig. 16.24 *Surface roughness parameters*

The other parameter that is sometimes used is the root mean square value of the deviation in place of the arithmetic average, $R_{RMS.}$ This in expression form is

$$R_{\text{RMS}} \cong \sqrt{\frac{1}{N}\sum y_i^2}$$

Waviness Waviness refers to those surface irregularities that have a greater spacing than that of roughness width. It is determined by the height of the waviness and its width. The greater the width, the smoother is the surface and thus is more desirable. Also the greater the width, the greater is the difference between the size of the measurement units required to measure height (roughness) and those needed to measure the waviness width.

Lay direction It is the direction of the predominant surface pattern produced on the work piece by the tool marks.

Flaw These are those surface irregularities which are random and therefore will not be considered.

Generally the surface roughness is measured by a stylus type of instrument as shown in Fig. 16.25. The stylus moves over the sample length of the surface and records the peaks and valleys of the surface as a set of digitised points of the surface. These will then be fed into the computer, where various types of parameters that are relevant for analysing the surface texture can be calculated.

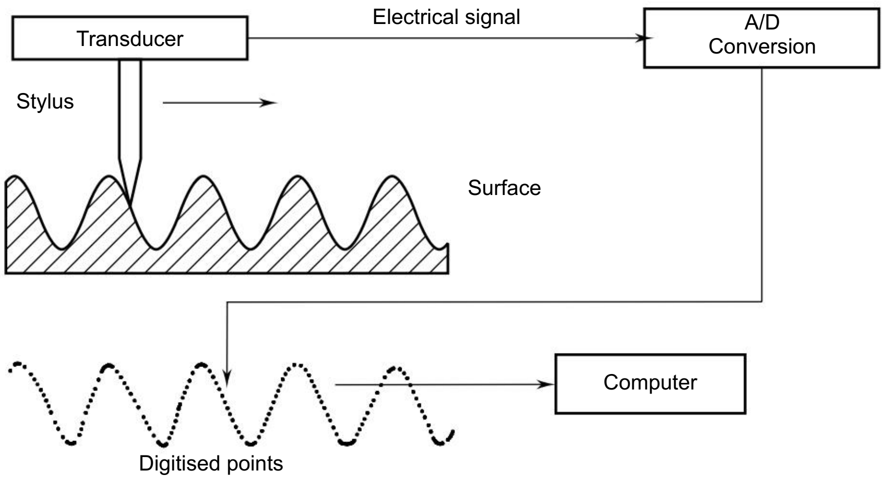


Fig. 16.25 *Principle of Surface roughness measurement*

16.7 GAUGES AND GAUGE DESIGN

Adoption of limits and fits as described above requires that the production process be designed to provide the necessary dimensions within the specified tolerances. It is also necessary to inspect all the parts to see whether they confirm to the requirements or not. Gauges or limit gauges are used for such purpose. Gauges are used for inspecting a given dimension, to check whether it is within the tolerance limits specified or not. Thus a gauge is used for accepting a dimension rather than measuring it.

Consider a simple snap gauge used to inspect a shaft as shown in Fig. 16.26. The gauge consists of two elements. A 'GO' section represents the maximum limit on the possible dimension of the shaft. A 'NOT GO' section represents the minimum limit on the dimension of the shaft. If the shaft size is within the tolerance limit specified, then the condition shown in Fig. 16.26 (b) will be satisfied, i.e., the part will go past the GO section and will not go beyond the NOT GO section as shown.

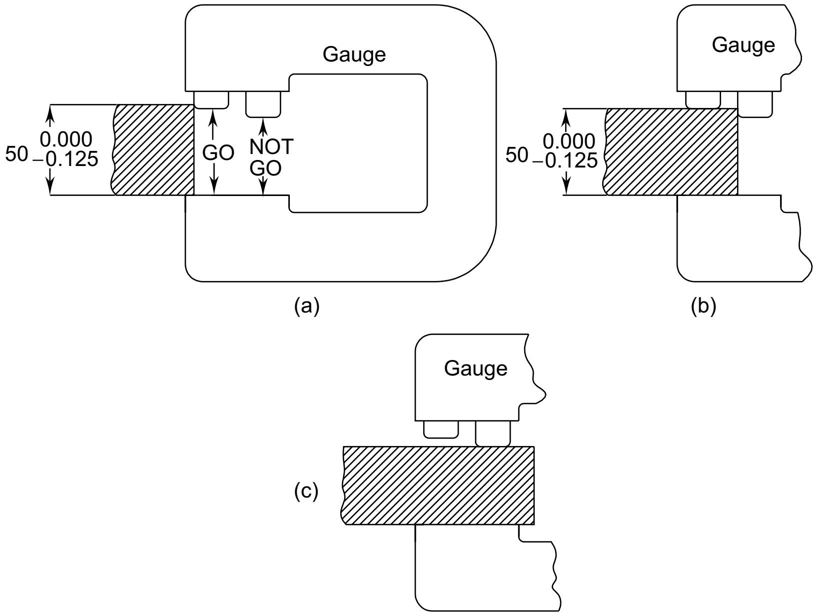


Fig. 16.26 *Principle of a snap gauge*

The condition shown in Fig. 16.26(a) is that the part dimension is larger than the maximum limit allowed and as a result the part will not go into the GO section. Similarly the condition shown in Fig. 16.26(c) is that the part dimension is smaller than the lower limit of the part resulting in the part going through the NOT GO section.

All the gauges will have similar arrangement, however the arrangement of the GO and NOT GO segments will have to be decided based on the profile dimension to be gauged. A variety of limit gauges are used industry and a few of them are mentioned below:

Snap gauge—For gauging external dimensions

Plug gauge—For gauging internal dimensions

Taper plug gauge—For gauging taper holes

Ring gauge—For gauging external diameters

Gap gauge—For gauging gaps and grooves

Radius gauge—For gauging radii

Thread pitch gauge—For gauging external threads

Special gauges—For specific applications such as alignment gauge, valve seat gauge, etc.

As shown above the gauges are high precision instruments used for acceptability of a given dimension of a part. However, the gauges themselves have to be manufactured using any of the manufacturing process. Whatever be the manufacturing process adopted and whatever care is exercised during the manufacture, it is impossible to manufacture the gauges to the exact dimensions. Hence it is necessary to allocate tolerances on the gauge dimensions, particularly the GO and NOT GO sections. The tolerance if applied outside the acceptable limits will make the gauge to allow the part with higher tolerance, hence the tolerance is generally applied inside the part tolerance. The components allowed by such a gauge will all be within the tolerance limits. Generally a total of 10% of work piece tolerance is assumed to be for the gauge tolerance. This then is equally distributed between the GO and NOT GO sections as shown in Fig. 16.27. This value is generally very small in view of the small tolerance of the parts and generally is rounded off to the nearest 0.001 mm. With such an arrangement some good parts which are within the tolerance specified in the part print, but outside the limits of the gauge, and are likely to be rejected.

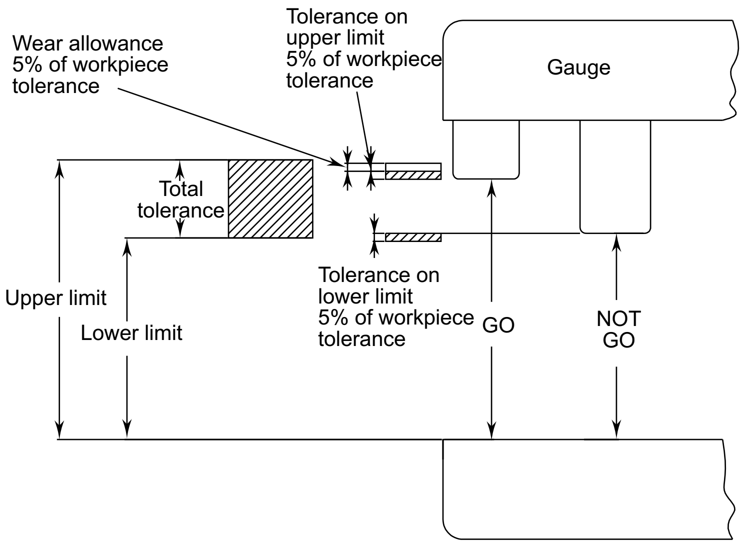


Fig. 16.27 *Disposition of wear in a snap gauge*

Another aspect to be considered in the design of the gauges is that the contact surfaces of the gauge are likely to wear out. The gauges are made with hard surfaces, which resist abrasion. But with continued usage over long periods, the GO section which experiences the relative movement with the part, should initially be made bigger to allow for the wear. About 5% of the work tolerance is normally allowed for this as shown in Fig. 16.27.

SUMMARY

Metrology is the science of measurements. Industrial production becomes a success only when it is possible to ascertain that the parts are produced within the dimensional limits specified during the design stage, at low costs.

- Standards are important to maintain the accuracy of measurements that are uniformly followed by all nations.
- Tolerance is the permissible deviation in a dimension that satisfies the operational requirements for the parts. Limits and fits allow for satisfactory functioning of the parts while lowering the manufacturing costs
- A number of different instruments of varying accuracy are used for linear measurement such as vernier, micro meter and slip gauges.
- Comparators are used to check quickly a linear dimension for the amount by which it deviates from the set dimension.
- Angular measurements are carried out with bevel protractor and sine bar.
- A number of parameters are to be measured to check the accuracy of threads. A number of methods are available for this purpose.
- Surface texture is a complex parameter that needs to be evaluated taking all the describing features into account. A simplified parameter such as arithmetic average is often used that can be measured with a stylus type instrument.
- Gauges are used to check the acceptability of a dimension within the specified tolerance values. This is normally used in mass production to quickly provide the measurement.

Questions

16.1 What do you understand by the term ‘Interchangeable Assembly’?

16.2 Define the terms tolerance, limits and fit with reference to the dimensional measurement.

16.3 Define the different types of tolerance specification methods. Compare their specific applications with examples.

16.4 Define a fit in connection with an assembly. What are the different types of fits possible?

16.5 Explain the concept of clearance, interference and transition fits. Give examples where these can be applied.

16.6 Briefly explain the ISO system of limits and fits. Explain about the tolerance grades as standardised by ISO.

16.7 Write a short note on the selective assembly.

16.8 What do you understand by the terms ‘hole basis’ and ‘shaft basis’ in terms of assembly fit specifications? Which is preferred? Give reasons supporting your answer.

16.9 What is the standard for linear measurement?

16.10 Explain the vernier principle as used in linear measurements.

16.11 What are the differences in the vernier and micro meter as used for linear measurements?

16.12 Write a short note on gauge blocks.

16.13 Explain the principle of sine bar for measuring angles.

16.14 Differentiate between measurement and gauging with reference to the application and method of use.

16.15 Explain how the gauge tolerance and gauge wear are allocated in the design of limit gauges.

Multiple Choice Questions

16.1 For an interference fit
- (a) The lower limit of the shaft should be greater than the lower limit of the hole
- (b) The lower limit of the shaft should be greater than the higher limit of the hole
- (c) The higher limit of the shaft should be greater than the higher limit of the hole
- (d) The higher limit of the shaft should be greater than the lower limit of the hole

16.2 For a clearance fit
- (a) The lower limit of the shaft should be smaller than the lower limit of the hole
- (b) The lower limit of the shaft should be smaller than the higher limit of the hole
- (c) The higher limit of the shaft should be smaller than the higher limit of the hole
- (d) The higher limit of the shaft should be smaller than the lower limit of the hole

16.3 Selective assembly of parts utilizes
- (a) An interference fit
- (b) A clearance fit
- (c) All the parts produced are measured and graded into a range of dimensions within the tolerance groups and then assembled with tighter tolerances
- (d) A transition fit

16.4 The accuracy of linear measurement is more with this instrument
- (a) Steel rule
- (b) Vernier callipers
- (c) Micro meter
- (d) Scale

16.5 V-anvil micro meter is used for measuring
- (a) Screw thread pitch
- (b) Screw thread minor diameter
- (c) The diameter of objects that have odd number of symmetrical or evenly spaced features, for example for the diameter of a 3-fluted end mill.
- (d) Chip thickness

16.6 The length standard that is most commonly used in the machine shops is
- (a) Meter rod
- (b) Slip gauge
- (c) Precision scale
- (d) None of the above

Answers to MCQs

16.1 (b) 16.2 (d) 16.3 (c) 16.4 (c) 16.5 (c)
16.6 (b)

CHAPTER 17

Numerical Control of Machine Tools

Objectives

The rapid developments taking place in electronics and computers is utilised by the machine tool industry to bring affordable automation to a large range of manufacturers. After completing this chapter, the reader will be able to

- Understand the need for soft automation
- Learn the historical development of numerical control
- Appreciate the various advantages and applications of numerical control machine tools
- Understand the principles of numerical control and various elements that form part of numerical control system
- Designate the different axes of the machine tool based on standards
- Learn part programming procedures using the ISO coding system
- Learn computer aided part programming system, in particular APT

17.1 INTRODUCTION

Competition between manufacturing firms is increasingly dictated by quality, cost, variety and servicing. Each one of these attributes of a successful product can only be produced by achieving the highest possible efficiency in manufacturing.

The variety being demanded in view of the varying tastes of the consumer calls for very small batch sizes. Small batch sizes will not be able to take advantage of the mass production techniques such as special purpose machines or transfer lines. Hence the need for flexible automation where you get the benefits of rigid automation but also be able to vary the products manufactured thus bringing in flexibility. Numerical control fits the bill perfectly, and we will see that future manufacturing would increasingly be dependent on 'Numerical Control' or NC to be short.

Numerical Control (NC) or control by numbers is the concept which has revolutionised the manufacturing scene, which is partially due to the rapid advancement in microelectronics that has taken place since the late 1960's. The key factor responsible for the popularity of the numerical control is the flexibility it offers in manufacturing.

Towards the end of the Second World War, there was increased activity in aerospace manufacturing in U.S.A. Mr John Parsons of Parsons Corporation who is one of the sub-contractors to USAF (United States Air Force), was toying with the idea of utilising the digital computers which were just then becoming popular to reduce the drudgery of computation. Machining (milling) of complex curvature is a highly skilled job. He proposed that the co-ordinate points of a complex three dimensional profile may be utilised for controlling the milling machine table so that accurate jobs could be produced. The USAF accepted his proposal and a contract was awarded to him to develop such a machine. The project was then awarded to the Servomechanism Laboratory of Massachusetts Institute of Technology in 1951, who had finally demonstrated a working milling machine in 1952. This is a Cincinnati Hydrotel Vertical Spindle milling machine with a controller built using valves (transistors were not available yet!).

Though the concept was demonstrated, the actual availability of such a machine for the aerospace industry was around 1955, after a very large number of refinements to the basic controller demonstrated in 1952. Later on, machine tool builders serving a variety of applications introduced several commercial NC units into the market. Since then rapid strides have taken place in NC technology, parallel with the developments in electronics and microelectronics.

17.2 NUMERICAL CONTROL

Numerical control of machine tools may be defined as a method of automation in which various functions of machine tools are controlled by letters, numbers and symbols. Basically a NC machine runs on a program fed to it. The program consists of precise instructions about the methodology of manufacture as well as the movements. For example what tool to be used, at what speed, at what feed and to move from which point to which point in what path. Since the program is the controlling point for product manufacture, the machine becomes versatile and can be used for any part. All the functions of a NC machine tool are therefore controlled electronically, hydraulically or pneumatically.

In NC machine tools one or more of the following functions may be automatic:

(a) starting and stopping of machine tool spindle
(b) controlling the spindle speed
(c) positioning the tool tip at desired locations and guiding it along desired paths by automatic control of the motion of slides
(d) controlling the rate of movement of the tool tip (i.e. feed rate)
(e) changing of tools in the spindle.

Initially the need of NC machines was felt for machining complex shaped small batch components as those belonging to an aircraft. However, currently this spectrum encompasses practically all activities of manufacturing, in particular the capital goods and white goods. Thus the range covered is very wide. Besides machining, with which we are concerned in this book, NC has been used in a variety of manufacturing situations. The majority of applications of NC are still in metal cutting machine tools such as milling machines, lathes, drilling machines, grinding machines and gear generating machines. Besides a number of metal forming machine tools such as presses, flame cutting machines, pipe bending and forming machines, folding and shearing machines also use NC for their programme control. The inspection machines called Co-ordinate Measuring Machines (CMM) are also based on NC. Lastly the robots basically may be material handling units, but their control principles are very close to the NC. Besides these applications listed for manufacturing, other applications such as filament winding or assembly machines based on the NC principles could also be widely seen in the industry.

NC machines have been found quite suitable in industries such as

1. For the parts having complex contours, that cannot be manufactured by conventional machine tools.
2. For small lot production, often for even single (one off) job production, such as for prototyping, tool manufacturing, etc.
3. For jobs requiring very high accuracy and repeatability.
4. For jobs requiring many set-ups and/or the set-ups are very expensive.
5. The parts that are subjected to frequent design changes and consequently require more expensive manufacturing methods.
6. The inspection cost is a significant portion of the total manufacturing cost.

One or more of the above considerations would justify the processing of a part by a NC machine tool.

NC is superior to conventional manufacturing in a number of ways. The superiority comes because of the programmability. These are as follows:

1. Parts can be produced in less time and therefore are likely to be less expensive. The idle (non-cutting) time is reduced to absolute minimum. This of course depends on the way the program for the part is written. The endeavour of the machine tool builder is to provide facility where by the non-cutting time can be brought to the barest minimum possible. It is possible to reduce the non-productive time in NC machine tools in the following ways:
   - by reducing the number of set-ups,
   - by reducing set-up time,
   - by reducing work piece-handling time, and
   - by reducing tool-changing time.

   These make NC machines highly productive.
2. Parts can be produced more accurately even for smaller batches. In the conventional machine tools, precision is largely determined by human skills. NC machines, because of automation and the absence of interrelated human factors, provide much higher precision and thereby promise a product of consistent quality for the whole of its batch.
3. The operator involvement in part manufacture is reduced to a minimum and as a result less scrap is generated due to operator errors. No operator skill is needed except in setting up of the tools and the work. Even here the setup has been simplified to a very great extent.
4. Since the part program takes care of the geometry generated, the need for expensive jigs and fixtures is reduced or eliminated, depending upon the part geometry. Even when the fixture is to be used, it would be very simple compared to a conventional machine tool. It is far easier to make and store part programs (tapes).
5. Inspection time is reduced, since all the parts in a batch would be identical, provided proper care is taken about the tool compensations and tool wear in part program preparation and operation. With the use of inspection probes in the case of some advanced CNC controllers, the measurement function also becomes part of the program.
6. The need for certain types of form tools is completely eliminated in NC machines. This is because the profile to be generated can be programmed, even if it involves 3 dimensions.
7. Lead times needed before the job can be put on the machine tool may be reduced to a great extent depending upon the complexity of the job. More complex jobs may require fixtures or templates if they are to be machined in the conventional machine tools, which can be reduced to a large extent.
8. CNC machining centres can perform a variety of machining operations that have to be carried out on several conventional machine tools, thus reducing the number of machine tools on the shop floor.

This saves the floor space and also results in less lead-time in manufacture. This results in the overall reduction in production costs.

9. Set-up times are reduced in a number of situations, since the set-up involves simple location of the datum surface and position. Further the required number of setups can also be reduced. All this translates into lower processing times. Many a times, a component could be fully machined in a single machining centre or turning centre, each of which has wider machining capabilities. In conventional manufacture if the part has to be processed through a number of machine tools which are located in different departments, the time involved in completion and the resultant process inventory, would be large. This would be greatly eliminated by the use of NC machine tools.
10. Machining times and costs are predictable to a greater accuracy, since all the elements involved in manufacturing would have to be thoroughly analysed before a part program is prepared.
11. Operator fatigue does not come into picture in the manufacturing of a part. The NC machine tool can be utilised continuously since these are more rigid than the conventional machine tools.
12. Tools can be utilised at optimum feeds and speeds that can be programmed.
13. The modification to part design can be very easily translated into manufacture by the simple changes in part programs without expensive and time consuming changes in jigs, fixtures and tooling. This adds to the flexibility of manufacture.
14. The capability (metal removal) of NC machines is generally high because of the very rigid construction employed in machine tool design compared to the conventional machine tools.

Though the NC machines have a range of advantages, there are certain limitations one should take care of while making a choice in favour of them.

1. The cost of NC machine tool is much high compared to an equivalent conventional machine tool. The cost is often 5 to 10 times higher. Also the cost of tooling is high. This is a very high initial investment. All this makes the machine hourly rate high. As a result, it is necessary to utilise the machine tool for a large percentage of time.
2. Cost and skill of the people required to operate a NC machine is generally high in view of the complex and sophisticated technology involved. The need is for part programmers, tool setters, punch operators and maintenance staff (electronics and hydraulics) who have to be more educated and trained compared to the conventional machine operators.
3. Special training is needed for the personnel manning the NC machine tools. NC manufacturing requires training of personnel both for software as well as hardware. Part programmers are trained to write instructions in desired languages for the machines on the shop floor. They also need to be acquainted with the manufacturing process. Similarly, machine operators have to be prepared for the new NC culture. These factors are important for the successful adoption and growth of NC technology.
4. As NC is a complex and sophisticated technology, it also requires higher investments for maintenance in terms of wages of highly skilled personnel and expensive spares. The need for maintenance engineers trained in all the sub systems present such as mechanical, hydraulic, pneumatic and electronics makes the job more difficult. Though the latest machines are equipped with a large number of diagnostic facilities, maintenance is still one of the major limitations.
5. The automatic operation of NC machines implies relatively higher running costs. Moreover, the requirement of a conditioned environment for operating NC technology adds further to the running costs.

17.3 NC MACHINE TOOLS

The CNC machining centre, at the moment appears to be the most capable and versatile automatic machine tool that can perform drilling, milling, boring, reaming and tapping operations. The general objective behind the development of NC machine tools continues to remain the reduction of cost of production by reducing the production time. This in turn is directed towards the avoidance of non-productive time which is mainly due to the number of set-ups, set-up time, work piece handling time, tool change time and lead time. The performance of a variety of machining operations on the same machining centre eliminates the non-productive waiting time, that occurs if such operations are performed on several different machines. Provision of automatic tool changing, indexing of tables and several pallets further add to the productivity of the machining centres.

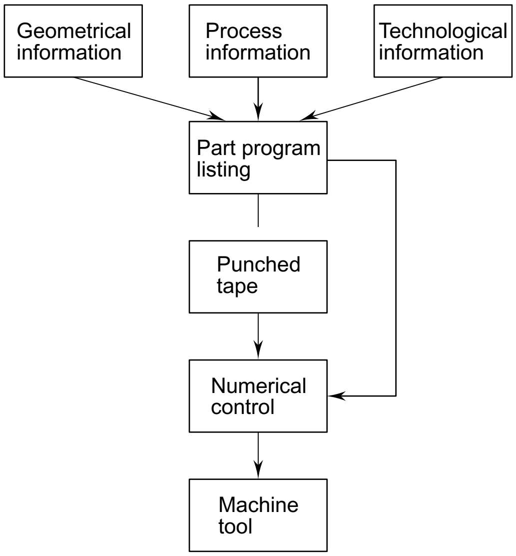


Fig. 17.1 *Principle of operation of a NC machine tool*

The principle of operation of a NC machine tool is shown in Fig. 17.1. The basic information that has to be input into the system consists of the part geometry, cutting process parameters followed by the cutting tools used. This part program is then entered into the controller of the machine, which in turn runs the machine tool to make the part.

The command received from the operator is communicated to the corresponding axis driving system for execution. The axis motion control system operates in a feedback loop with suitable transducers such as linear scales and/or rotary encoders to get the appropriate position or velocity feedback as shown in Fig. 17.2. Most of these systems have a very high response with good resolution of the order of 1 μm (micron) or less.

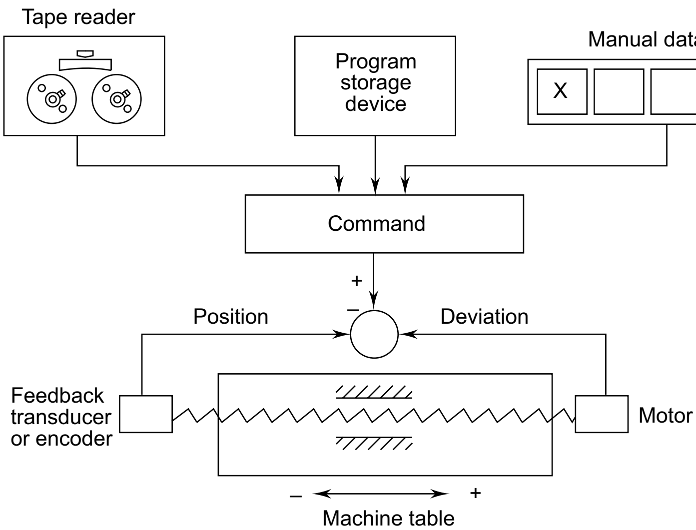


Fig. 17.2 *The data processing in a CNC machine tool in closed loop control*

The controllers have a number of modes in which to operate. There could be 4 possible modes in which the controller can function as shown in Fig. 17.3 in relation to a machining centre. The first shows a typical drilling machine operation, termed as point to point mode. In this, the control has the capability to operate all the 3 axes, but not necessarily simultaneously. As a result, it would be possible to move the tool to any point (in X and Y-axes) in the fastest possible speed and carry out the machining operation in one axis (Z-axis) at that point. This would be useful for drilling and punching machines. The second type is an improvement over this, in which in addition to the point to point mode, the machine tool has the capability to carry out a continuous motion in each of the axis direction. This would help in obtaining the milling in a straight line along any of the axes.

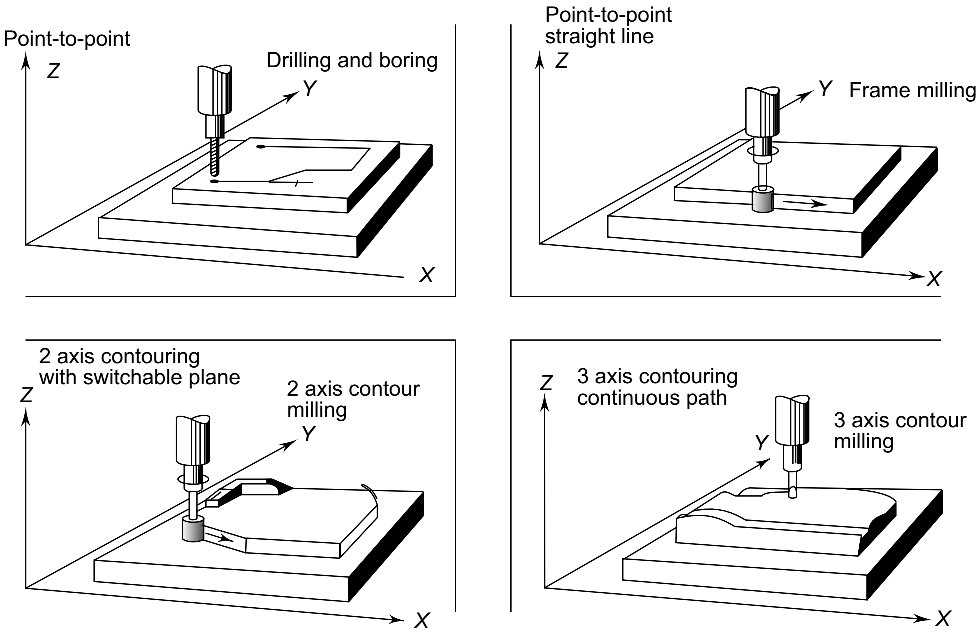


Fig. 17.3 *Types of control systems possible in CNC operation*

In the third type is shown a control system, which improves the previous type by adding the simultaneous motion capability in any 2 axes. This is what is required in most of the cases. Any 3D profiles to be machined can be completed using the concept of 2.5D mode, in view of the limitation of the machine.

The last one is the highest form of control that is generally found in most of the current day control systems. This gives the capability of simultaneous 3 or more axes motion. This would be useful for machining most of the complex 3D profiles encountered in industrial practice such as aerospace components, moulds and dies.

17.3.1 Machine Control Unit

Every NC machine tool is fitted with a machine control unit (MCU) which performs the various controlling functions under the program control. The MCU may be generally housed in a separate cabinet-like body or may be mounted on the machine itself. When separately mounted, it may sometimes be like a pendant which could swing around (Fig. 17.4) for convenient handling by the operator. Appearance wise it looks like a computer with a display panel generally of small size (9 inches), and a number of buttons to control the machine tool along with a keyboard. This control unit controls the motion of the cutting tool, spindle speeds, feed rate, tool changes, cutting fluid application and several other functions of the machine tool.

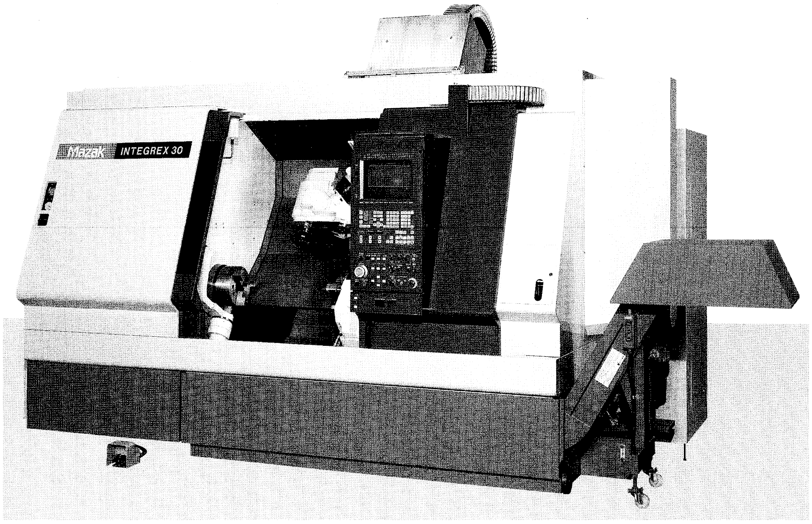


Fig. 17.4 *Machine tool and its control unit*

17.3.2 Part Program

This is a very important software element in the NC manufacturing system. It looks like a computer program containing a number of lines/ statements/ instructions (called NC blocks). It is, therefore, the detailed plan of manufacturing instructions proposed for machining the part. It is written keeping in view the vocabulary understood by the MCU in terms of various standard words, codes and symbols. The format and the language are dependent on the machine tool hardware and the MCU. Some typical NC blocks written in the word address format as per ISO are shown below:

```
N30 G00 X120.0 Y 45.0 Z-85.0
N40 G90
N50 G03 X200.0 Y200.0 I-100.0 J0 F200
N60 G01 X120.0 Y110.0
```

The program can also be written in higher level languages such as APT, UNIAPT, COMPACT II, etc. These programs have to be converted into the earlier mentioned machine tool level program with the help of processors. It is similar to the practice by which computer programs written in high level languages such as FORTRAN are converted into the relevant computer machine language with the aid of a suitable compiler. Programming done in a language such as APT is mostly processed with the help of a computer and so is also known as computer aided part programming, discussed later.

The programs can also be developed directly using the CAD/CAM systems such as Unigraphics, Pro Engineer, Euclid, and SDRC I-DEAS or CAM systems such as MasterCAM, SmartCAM, SurfCAM, Duct, etc. These also would require a post processor like the earlier discussed computer aided part programming systems.

17.3.3 NC Tooling

The operator gathers, or is supplied with, the relevant tooling for the part to be machined. A distinctive deviation of the NC tooling from the conventional one is that each cutting tool is set in a different adapter

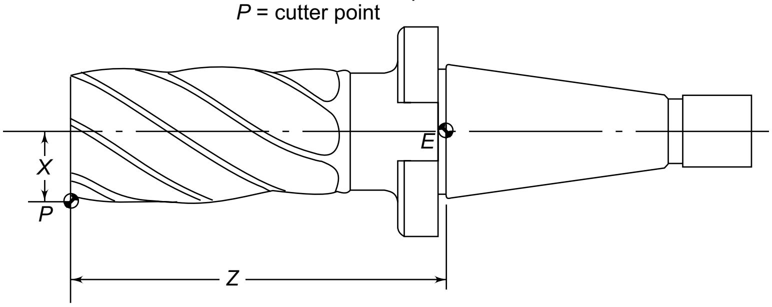


Fig. 17.5 *Typical spindle tooling holding an End Mill*

(Fig. 17.5). The configuration suggested by ISO is now generally followed. A power-operated drawbar may be employed to pull the tooling at the retention knob. This helps eliminate any clearance between the mating surfaces of spindle and tooling shank. It is not uncommon to set apart an allocation of 20 to 30% of total budget for tooling during the buying of new NC machine tools.

A pre-set tool has adjustable locating faces. It enables the dimensions between the tool cutting edges and location faces to be pre-set to a close tolerance using a pre-setting device. The pre-set tool usually needs to be removed from the machine for adjustments required during batch production. The tools may be stored on a drum, which is operationally an integral part of the machine itself. In the latter case, the tools are automatically replaced or changed in the spindle.

These inform the operator about the deviation of the tool tip the supplied tool has, with the one taken into account by the part programmer. The programmer gets the information from the tool files that are updated periodically. In spite of the "updating", the position of the tool tip when supplied to the operator may be different (from what is mentioned in the tool file) because of wear and tear, re-sharpening or setting of a new cutting tool due to breakage.

17.3.4 Axes of NC Machines

The major component of a NC program involves the input of co-ordinates of the tool end point to produce any machining profile, it is necessary to follow a proper co-ordinate system. To this extent the axes designation was standardised by EIA (Electronics Industry Association, USA) and ISO. Most of the NC machines' builders follow the International Standard ISO/R841 to designate the axes of their machines. The principles followed in this standard are explained below.

Co-ordinate System

All the machine tools make use of the Cartesian co-ordinate system for the sake of simplicity. The guiding co-ordinate system followed for designating the axes is the familiar right hand co-ordinate system. The main axes to be designated are the rectangular axes and the rotary axes. Typical right handed co-ordinate system is shown in Fig. 17.6. One could use his right hand (as shown in Fig. 17.7) to arrive at these alternate variable positions of the same right hand co-ordinate system.

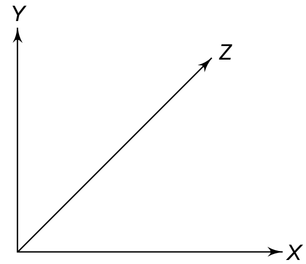
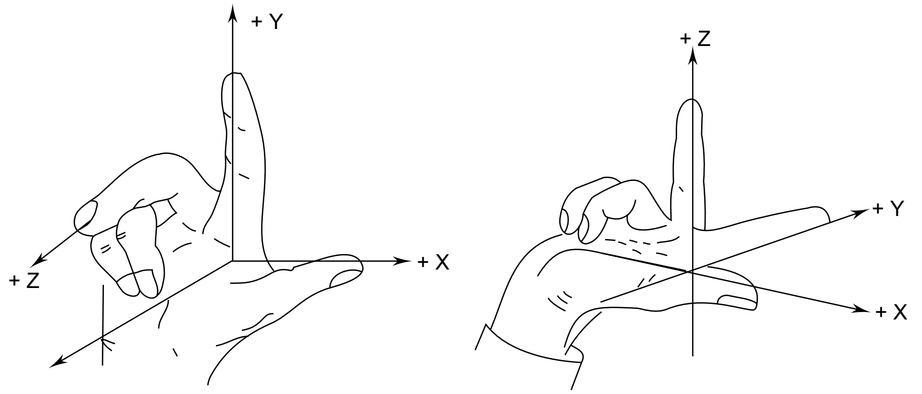


Fig. 17.6 *Right hand co-ordinate systems*



(a) Axis designation for horizontal *Z*   (b) Axis designation for vertical *Z*

Fig. 17.7 *Finding directions in a Right Hand Co-ordinate System and also the positive directions for rotary motions*

Designating the Axes

First axis to be identified is the Z-axis. This is followed by the X and Y axes respectively.

Z-Axis and Motion

Location The Z-axis motion is either along the spindle axis or parallel to the spindle axis. In the case of machine without a spindle such as shapers and planers, it is identified as the one perpendicular to the work-holding surface, which may or may not be passing through the controlled point (e.g. the cutting tool tip in case of shaper).

Direction The tool moving away from the work holding surface towards the cutting tool is designated as the positive Z direction. This means in a drilling machine the drill moving into the work piece is the negative (–) Z direction. This helps in reducing the possible accidents because of wrong part program entry in the co-ordinate signs.

When there are several spindles and slide ways: In such cases, one of the spindles, preferably the one perpendicular to the work-holding surface may be chosen as the principal spindle. The primary Z motion is then close to the primary spindle. The tool motions of other spindle quills or other slides, which are termed as secondary and tertiary motions, may be designated as U, V, W and P, Q, R respectively.

For other machines the positive (+) Z motion increases the clearance between the work surface and the toolholder. The designation of Z-axis is demonstrated in Fig. 17.8.

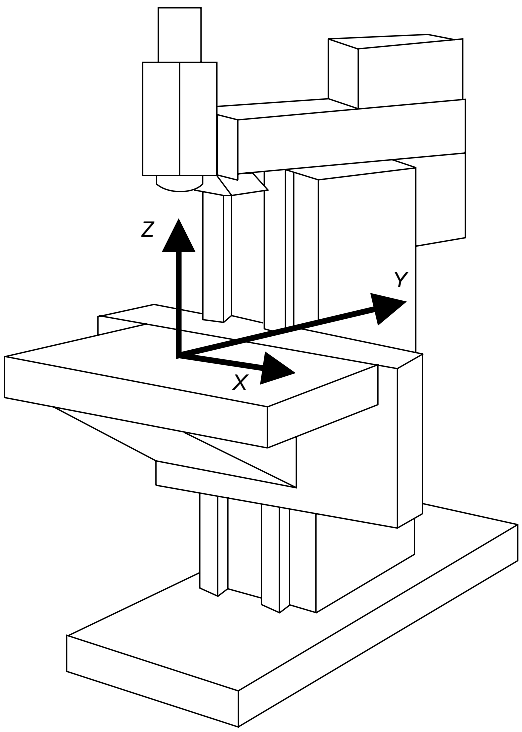


Fig. 17.8 *Vertical axis Milling Machine or Machining centre*

X-Axis The X-axis is the principal motion direction in the positioning plane of the cutting tool or the work piece.

Location It is perpendicular to the Z-axis and should be horizontal and parallel to the work- holding surface wherever possible.

Direction When looking from the principal spindle to the column, the positive (+) X is to the RIGHT. For turning machines, it is radial and parallel to the cross slide. X is positive when the tool recedes from the axis of rotation of the work piece. For other machine tools, the X-axis is parallel to and positive along the principal direction of movement of the cutting or the guided point.

Y-Axis It is perpendicular to both X- and Z- axes and the direction is identified by the right hand Cartesian co-ordinate system.

Rotary Motions

A, B and C define the primary rotary motions.

Location These motions are located about the axis parallel to X, Y and Z respectively. In addition to the primary rotary motions, if there are secondary rotary motions, those should be designated as D or E regardless of whether they are parallel or not to A, B and C.

Direction Positive A, B and C are in the directions which advance right hand screws in the positive X, Y, and Z directions respectively. In Fig. 17.7, the fingers of the right hand point towards the positive direction of the rotary motions. All the above-mentioned motions, viz. X, Y, Z; U, V, W; P, Q, R; A, B, C and D, E are with reference to a point, the movement of which is sought to be controlled. This point is generally the tip of the cutting tool. Often the tool point may not move in some directions, e.g. the quill of the spindle of a vertical milling machine moves in the Z direction but not in the X and Y directions. In such cases, the work surface is generally moved in a direction opposite to the one intended for the tool, e.g. the table of the milling machine holding the work piece may be moved in –X and –Y directions. Such movements of machine elements, say –X or –Y, are denoted as +X' or +Y' respectively. Primed letters can thus be used for all the above-mentioned motions to indicate the corresponding reversed directions for moving work surfaces.

As already discussed, most of the machine tool manufacturers adhere to the standards to a very great extent. However, some deviations may be present in some cases because of historical reasons or specific convenience in operations or programming of the machine tool. Some examples of the axes designation as suggested above and applied to practical machine tools is described below. In Fig. 17.9 is shown a typical horizontal axis boring mill in 3 and 4 axes configuration. In the 4-axes version, a complimentary motion parallel to the spindle movement (Z-axis) is designated as W-axis.

A 5-axes machining centre with a horizontal spindle is shown in Fig. 17.10. In addition to the normal 3-axes (X, Y and Z), two rotary axes A and B are added. In one case, the spindle originally horizontal is swivelling about the X-axis. A rotary table is added on the table to give a rotary motion about the Y-axis.

17.3.5 CNC Machining Centres

CNC machine tools have grown from the conventional machine tools. For example, the successful tool room milling machines were converted to CNC by simply replacing the motion elements by automated devices. Later the CNC machine tools were redesigned with greater emphasis on the structural rigidity, power

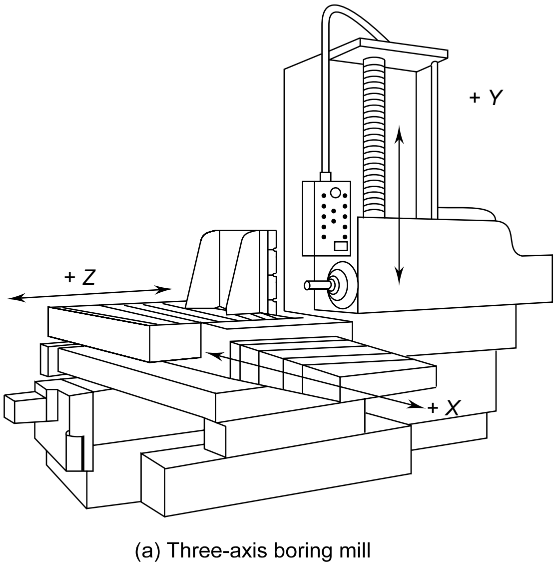


Fig. 17.9 *CNC Horizontal axis boring mills in 3 and 4 axes versions*

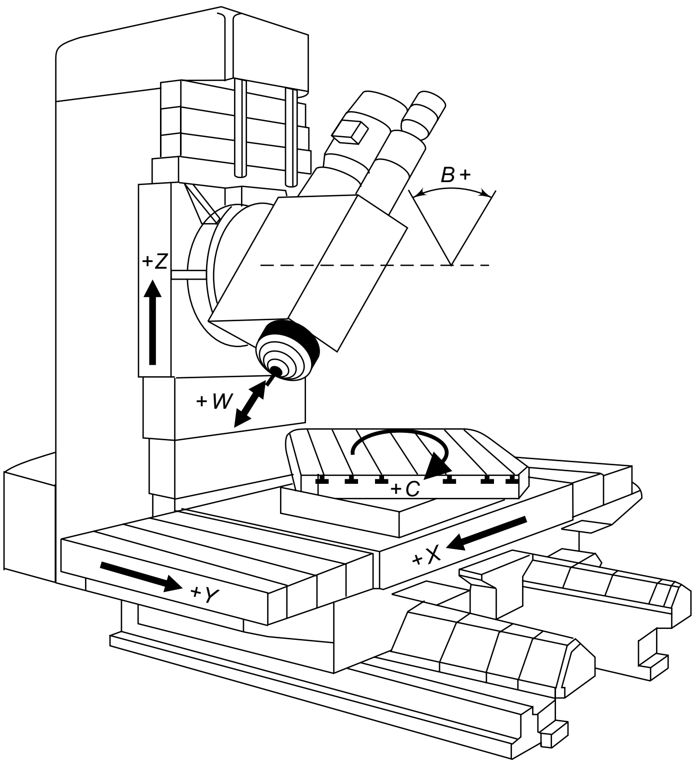


Fig. 17.10 *5 axes CNC Vertical axis machining centre configuration*

available as well as the ability to perform a variety of functions. Also, most of the modern CNC milling machines have expanded machining capabilities by the addition of accessory devices, making them more versatile. That is why these are now called 'machining centres' rather than milling machines.

The CNC machining centres can be broadly categorised into two varieties:

- Vertical axis machining centre, and
- Horizontal axis machining centre.

Vertical Axis Machining Centre

The vertical axis machining centres or VMC as is popularly abbreviated, are generally more versatile in terms of the tool being able to generate more complex surfaces compared to the horizontal axis. Most of the early CNC machine tools therefore are of this category. A typical example is shown in Fig. 17.11. However, the subsequent developments have improvised the machines in such a way that the current machines have a much rigid construction. Most of the general machines come with 3 axes. Additional axes will be added to cater to the machining of more complex geometries. For example the spindle head can be swivelled in one or two axes (about X or Y axis) to provide A or B axis motion. These are required for machining sculptured surfaces.

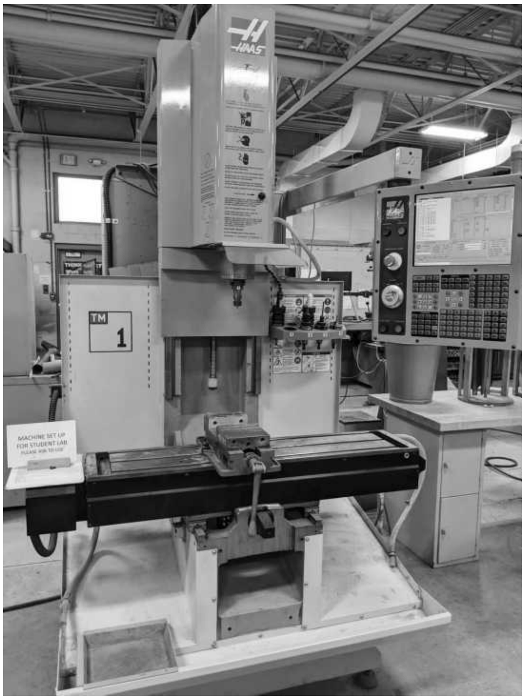


Fig. 17.11 *CNC Vertical axis tool room mill*

Horizontal Axis Machining Centre

By its very configuration, the horizontal axis machining centre or popularly called HMC is sturdier than the vertical configuration and hence is used for heavier work pieces with large metal removal rates. Since these machines provide for heavier metal removal rates, the cutting tools used would normally be big. As a result, the tool magazine will have to provide larger place for each tool. This results in the tool magazines for HMC to become heavier. Also, they are normally provided with tool magazines having higher capacity.

CNC Turning Centres

Majority of the components machined in the industry are of the cylindrical shape. Hence the CNC lathes, more appropriately called turning centres, are also important machine tools. The evolution of the CNC turning centres follows the developments in the CNC machining centres closely.

The major change to be noticed in the turning centres is the early adoption of the slant bed (Fig. 17.4) to allow for a better view of the machining plane as well as for easy placement of the various devices involved in the machining zone. Most of the turning centres are also provided with a tool turret which may have a capacity of 8 to 12 tools of various types (Fig. 17.12).

Fig. 17.12 *CNC Turning centre with tool turret*

17.3.6 Coordinate Measuring Centres

Fig. 17.13 *CNC Coordinate measuring machine*

Coordinate measuring machines (CMMs) are similar to any metal cutting machine tool except that the tool is replaced by means of a touch trigger probe (Fig. 17.13). This makes it an extremely powerful metrological instrument. When the touch trigger probe touches the part surface it generates the coordinates of the point it contacted. Thus it can be used for measuring the geometrical characteristics of the part. The CMM can be manually operated by an operator using a joystick type control or can have full CNC control. Programs can be written using the programming language similar to the CNC part program, for the probe to contact the part surface to get the dimensions as required.

17.4 PART PROGRAMMING FUNDAMENTALS

Part programs for simple components can be carried out manually. However, if the component has complex features which require too many repetitive and/or tedious calculations for preparing its program for cutter path description, then it is recommended that computer aided part programming be resorted to.

To be a good CNC programmer, one should have a fair knowledge about the machine tools, cutting tools and fixtures to be used and the manufacturing processes. He also should have a good understanding of geometry, algebra and trigonometry. In fact, machine shop experience is the prerequisite for a good programmer, as only careful process planning can lead to efficient and practical programs.

The following are the steps to be followed while developing the CNC part programs.

- Process planning
- Axes selection
- Tool selection
- Cutting process parameters planning
- Job and tool setup planning
- Machining path planning
- Part program writing
- Part program proving

Process Planning

Process plan is a detailed plan of the steps involved in manufacturing (machining) a given part. The following are the contents of a process plan:

- Machine tool used
- Fixture(s) required
- Sequence of operations
- For each of operation
- Cutting tools required
- Process parameters

A programmer is supposed to carry out a careful study of the part drawing to prepare the process plan. The choice of the machine tool used depends upon the operations required, accuracy requirements, machine tool capability and availability, cutting tool availability and the shop practices. A careful choice of various options at this stage would decide on the final cost of manufacture of the part.

Axes Selection

All the CNC machine tools rely on the axes system for describing the axes motion. To correctly describe the motion, it is therefore necessary to establish the axes system to be followed with the particular part. The ISO designation of axes was discussed earlier. In tune with that axes system, one has to choose the axes. However, it is also necessary for one to choose the axes system as appropriate to the machine tool co-ordinate system in question.

The axes system of all the CNC machine tools would generally have a fixed datum position as designated by the machine tool manufacturer. It may be called the reference position, fixed datum or home position. This absolute datum position of the CNC machine tool may not be very convenient for setting the job. Hence most of the CNC machine tools come with 'Floating Datum'. In this case the programmer can select the part datum anywhere in the machining limits of the machine tool, based on the geometry of the part being machined.

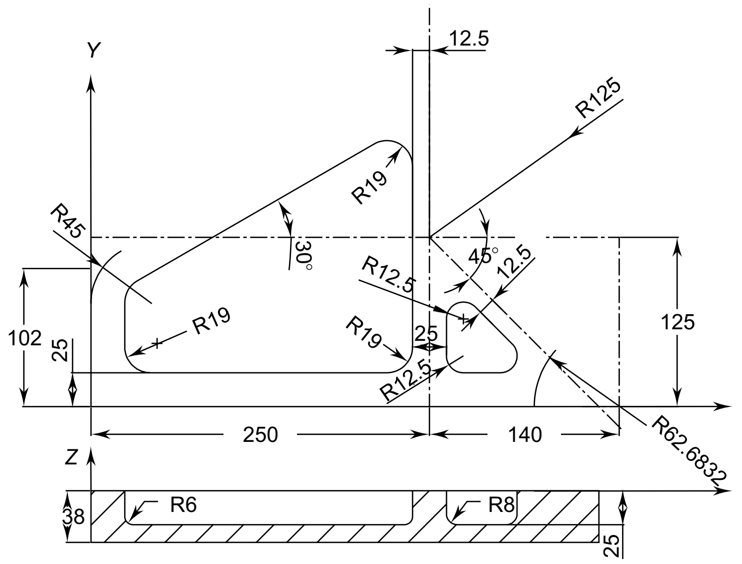
The reference axes should be chosen so that co-ordinates for various features can be determined Fig. 17.14(a). Here X and Y are the reference axes. For the sake of convenience, the orthographic views of the component are shown.

The basis for choosing the axes system has more to do with the part geometry and the type of machine tool being used. When the operator is developing the program, it becomes extremely important to choose the right type of datum, since a careful selection eliminates a large amount of calculation process. Also the part program becomes simple, being able to make use of the advanced software facilities such as mirror imaging, etc.

The first principle to be used while arriving at the datum is to keep all the parts in the first quadrant of the co-ordinate system. This would help in having all the co-ordinate values as positive. It helps the first time programmer in eliminating as many errors as possible. Once enough experience is gained, it would be possible for the programmer to carefully adjust the values. Similarly touching the two sides of the pre-machined work piece can easily do the setting of the tool.

Sometimes the datum could be chosen as the geometric centre of the work piece if all the geometry is symmetrical as shown in Fig. 17.14(b). In such a choice, the geometry calculation effort reduces to a minimum. Also the mirror image facility available in the controller can be effectively exploited.

The Z-axis datum is kept generally to match with the top surface of the work piece. This helps in two ways. First, all positive values of Z co-ordinate would keep the tool away from the work piece, so that the collision of tool with the work is avoided. Secondly when the tool is to be set, the tool tip can be easily matched with the work piece top surface.



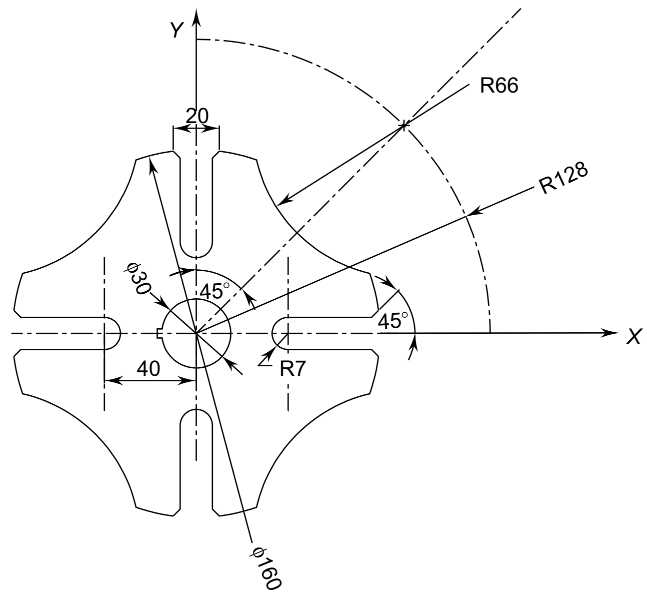


Fig. 17.14 *A typical component with axes system designated*

Tool Selection

The choice of cutting tools is a very important function. For a given operation many tools are feasible, but some of them are more economical than others. Therefore for the economy of manufacture, it is essential to choose the right tool for the job. As a rule, we will only select the regular cutting tools for use in CNC machine tools. No special tooling is generally suggested, since the geometry can very well be generated by the CNC control.

As an example when a contour is being milled, the choice can be an end mill or a slot drill (end cutting end mill). End mill is stronger and can take deeper cuts than a corresponding slot drill. However, slot drill can enter into a solid material, but an end mill cannot in view of the fact that the cutting edge in the bottom does not extend to the centre of the tool. As a result, an end mill should always approach the work piece from the side while the slot drill can approach from the side or from the top.

Also the size of an end mill or slot drill depends on the contour to be machined. You should choose the largest size of end mill available for better surface finish and higher material removal rate. However, the tool radius may be limited often by the radius of curvature being generated.

For example, sometimes a tapered end mill or slot drill would make the machining very simple in generating the draft surfaces of dies and moulds. Otherwise generating a draft angle may take a large amount of programming.

Cutting Process Parameters Planning

For a given tool and the operation selected, the appropriate process parameters are to be selected. These are to be generally taken from the handbooks supplied by the cutting tool manufacturers or based on the shop experience. It is important in the context of CNC manufacture that the feeds and speeds selected should be as high as possible to reduce the machining time consistent with the product quality achieved.

Job and Tool Setup Planning

This aspect would be covered in greater details later. This basically is aimed at setting the job on the machine tool and adjusting the cutting tool to the correct position. This is important since the accuracy of the geometry generated by the CNC machine tool is dependent on the initial position carefully defined.

Machining Path Planning

It is a very important aspect of programming wherein the knowledge of machining operations plays a vital role. A careful planning of the tool path ensures that the requisite manufacturing specifications are achieved at the lowest cost.

Part Program Writing

This aspect deal with the actual writing of the part programs, taking the format and syntax restrictions into account.

Part Program Proving

This is another aspect, which the programmer should very carefully do before the part program is released to the shop. Once the program is made, it should be verified before it can be loaded on the machine tool controller for the manufacture of the component. It is obvious that a faulty program can cause damage to the tool, work piece and the machine tool itself. Sometimes, these accidents can prove grave for the operator and others around. One of the preliminary ways of avoiding such possibilities is to carry out a visual check of the program manuscript and of the displayed program on the VDU of the controller.

But this is understandably not enough in itself. A trial run can be carried out with or without the tool or work piece to enable visualisation of movements taking place and of any collisions possible between the tool, the work piece and the clamping device. At this stage, it is worthwhile to stress the point that while the program is being prepared, the positions of the clamps should be taken into account and they should be clearly indicated in instructions to the operator. This is vital for eliminating the possibility of collisions occurring during machining.

During trial runs, feed and speed override control should be used so that the operator works at such values which enable him to exercise manual control comfortably and operate the emergency switch well in time. The program is run block by block, i.e. after execution of each block the machine waits till the operator manually presses the switch on the machine console for execution of the next block.

With the job and tool in position, dry runs are made, i.e. keeping a safe distance in between the tool and the job; the motions can be visualised for correctness. If during these trials, any mistakes are noticed, the program is examined and the necessary corrections made. After this, one component is made and checked. Based on this, speeds and feeds are modified and further corrections carried out so that correct profiles are obtained. Sometimes only a single complex and precise job may be made using numerical control. This could even be of an expensive material. In such cases, the program is tried first on a cheap material, say wood, Perspex etc. Only when the first trials are approved, the updated program is permitted to be used for further production.

Nowadays, graphical simulation packages are available on CNC systems, which make possible a graphical output on the VDU screen. This output shows the work piece and the tool, the motion of the tool and the progressive removal of material as the program proceeds. Visualisation of this animation of the machining process helps to prove the part program before any actual machining is carried out. These verifications are carried out at a fast speed and thus the proving of the part program is done without much loss of time.

It is also possible to carry out the verification on a microcomputer screen. Through these, it is possible to see how the tool path is programmed. A more advanced version is the simulation program, which can show how the material is being removed, so that the actual geometry generated can also be seen in these systems. This enables a fast detection of mistakes and their correction without loss of production time of the CNC machine tools. Many of these systems have the capability of showing the clamps and other elements, which are likely to interfere with the tool movement. Also some systems have the capability of dynamically simulating the actual sized tool through the work material to make the simulation more realistic.

Another simple method of verifying the program is plotting. However, it should be understood that this would give only a two dimensional picture. The plotter is connected through an interface for obtaining the plot. The plot can be examined and compared with the component drawing for any errors in the tool path.

Documentation for NC

It would now be amply clear that documentation is the most essential aspect of the CNC manufacturing practice. Therefore, it is worthwhile to list these as a checklist.

1. Component drawing.
2. Process planning sheet: As discussed earlier, this should contain details of the sequence of the operations, the machine tool used, the tools used with their numbers, speeds, feeds etc.
3. Tool cards: These should show each tool in assembled form with dimensions and identification numbers for each element (tool, collet, chuck etc.).
4. Setting card: This would show all tools, as in position on the machine tool, with their identification numbers, and the corresponding compensation values.
5. Programming sheets.
6. Punched paper tape, if this is the input form used.

The originals of these documents are kept in the programming room records cabinet while copies are sent to the shop floor as per the production planning. Whenever any changes are to be made, all issued copies are recalled and destroyed. The originals are updated (or made afresh) and copies are released accordingly.

17.5 MANUAL PART PROGRAMMING METHODS

In the earlier days, a number of formats for NC part programs were used such as fixed sequence or tab sequential. These systems required giving a large number of unwanted or duplicate information in each block of a part program. Now these have been replaced by means of a system called 'Word Address Format' in which each of the information or data to be input in the form of numerical digits is preceded by a word address in the form of an English alphabet. For example N105 means that N is the address for the numerical data 105. Thus the controller can very easily and quickly process all the data entered in this format. A typical block of word address format may look as follows:

N115 G81 X120.5 Y55.0 Z-12.0 R2.0 F150 M3

17.5.1 ISO Standards for Coding

In the early years of development of Numerical Control standardisation has been given due importance. As a result many of the things that we use in NC are standardised and many of the manufacturers follow the standards to a great extent. One of the first things to be standardised is the word addresses to be used in programming. All the 26 letters of the English alphabet were standardised and given meaning as follows:

Character	**Address For**
A	Angular dimension around X axis
B	Angular dimension around Y axis
C	Angular dimension around Z axis
D	Angular dimension around special axis or third feed function*
E	Angular dimension around special axis or second feed function*
F	Feed function
G	Preparatory function
H	Unassigned
I	Distance to arc centre or thread lead parallel to X
J	Distance to arc centre or thread lead parallel to Y
K	Distance to arc centre or thread lead parallel to Z
L	Do not use
M	Miscellaneous function
N	Sequence number
O	Reference rewind stop
P	Third rapid traverse dimension or tertiary motion dimension parallel to X*
Q	Second rapid traverse dimension or tertiary motion dimension parallel to Y*
R	First rapid traverse dimension or tertiary motion dimension parallel to Z*
S	Spindle speed function
T	Tool function
U	Secondary motion dimension parallel to X*
V	Secondary motion dimension parallel to Y*
W	Secondary motion dimension parallel to Z*
X	Primary X motion dimension
Y	Primary Y motion dimension
Z	Primary Z motion dimension

* Where D, E, P, Q, R, U, V, and W are not used as indicated, they may be used elsewhere.

The complete part program for a given component consists of a beginning code of % which signifies the start of the tape (in case of paper tapes) or beginning of a program if direct computer communication is involved such as in DNC mode. A part program consists of large number of blocks (similar to sentences in a letter) each representing an operation to be carried out in the machining of a part.

Each block always starts with a block number used as identification and is programmed with a ‘N’ word address. This must be programmed at the beginning of every block. As per ISO 2539, it has a minimum of three digits, e.g. N009, N028. However, some control manufacturers, notably Fanuc dispense with this requirement. In their case, only those blocks which are to be specifically addressed as per the requirement of program flow would be given a block number. Other blocks can do away with this requirement. This saves valuable RAM space in the controller where the part programs are stored.

Each block can have one or more of the word addresses as explained above in a sequence. A typical ISO format for block is shown below:

N5 G2 X±53 Y±53 Z±53 U..V..W..I..J..K..F5 S4 T4 M2 *

This shows a typical sequence in which the word addresses should occur in the block. However, it is not necessary that all these addresses should be present in each of the block nor is the sequence important. The word addresses can occur in any sequence.

The numerical values immediately after word address indicate the maximum number of digits that are allowed for that particular address character. For example, G, the preparatory function is followed by two-digit information, say G00 to G99. The unsigned numbers indicate that the numerical value given will be without any sign. Also a single digit indicates that the numerical value to be given is an integer.

When real values are to be given, two digits indicate them, the first one representing the number of digits before the decimal place while the latter is for those after the decimal place. For example, X ± 53 indicates that five digits before the decimal and three after it are needed to describe the word address X. The ± indicates that this address can be given with a sign. The + sign need not be given, since it is automatically assumed.

As per the standards followed, decimal sign should not be given, its position being defined by the format specifications. However, many controllers allow the decimal point as it is better for easy understanding of the program. It is also easier to program directly the numbers with decimal point.

Since each function is indicated by its address character, the order of writing words in a block is not important except that the letter N should come right in the beginning and the end of block should be placed where the information for that block is completed. Fanuc uses the end of block character as ";". Others treat the "Carriage Return" and "Line Feed" as End of Block (EOB). In this book we will use (*) as the end of block for easy understanding, though this is not required to be punched in the actual part program.

In the variable block format the number of words and characters are variable, i.e. if any word is not required in any block, then it need not be written and also if the value of any function remains the same in the next operation, then it need not be repeated in the block. The following examples will clarify these details.

N110 G01 X-312.55 Y14.5 Z12.565 F200 S1500 T1103 M03 *
N115 Y187.0 Z0 *

In the block for operation number 110, the value of functions G, X, B, F, S, T and M will be the same as in operation number 115; only the values for the functions Y and Z will change to 187.0 and zero respectively. This feature of the word address format, and also since the order of words is not important, it makes the writing of programs very convenient.

Since the programming format for various control systems are not identical and may differ from ISO recommendations, it is important that the relevant programming manual should be consulted while preparing the program.

17.5.2 Co-ordinate Function

As discussed above, the co-ordinates of the tool tip are programmed for generating a given component geometry. The co-ordinate values are specified using the word address such as X, Y, Z, U, V, W, I, J, K, etc. All these word addresses are normally signed along with the decimal point depending upon the resolution (at least 1 μm or less for precision CNC machine tools) available in the machine tool. Some examples are:

X123.405 Y-34.450

17.5.3 Feed Function

Generally the feed is designated in velocity units using the 'F' word address. For example, F150 means that the feed rate is specified as 150 mm per minute. This is the actual speed with which the tool moves along the programmed path. However, depending upon the programmed path, there could be some deviations in the actual feed followed by the controller. Also the controller calculates the actual feed rate of each of the axis.

Once the feed rate is programmed in a block, it remains in force in all the subsequent blocks till it is replaced by another F value, i.e. it is modal.

The feed rate programmed can be overridden by a setting on the controller console, in steps of 10% between 0 and 150%. However, in some situations this override will not work, for example, in case of thread cutting or thread milling.

By using an appropriate 'G' code, it is also possible to change the feed rate units from mm per minute to mm per revolution or vice versa.

17.5.4 Speed Function

In some of the CNC machine tools, spindle speeds are set manually and so are not to be programmed. However, most of the CNC machines that are coming now would have the capability for the step less variation of spindle speeds. Hence they need to be programmed using spindle speed word 'S'.

The speed can be set directly in the revolutions per minute or RPM mode using the S word address as follows:

S1500 means that spindle speed is to be set at 1500 rpm.

However, in some cases such as in turning centres, when the work surface controls the actual cutting speed, then direct RPM program would make the cutting speed vary whenever the work diameter changes. This is harmful from the surface finish as well as the tool life point of view and hence another option for spindle speed function is the constant surface speed. When this option is exercised, the spindle speed is specified not in RPM but in meters/minute or feet/minute depending upon the units chosen.

17.5.5 Tool Function

All NC machines are generally equipped with turrets or tool magazines with automatic tool changers (ATC) which enable the positioning of the pre-set tools in a few seconds. Thus the ratio of cutting time to total machine time is considerably increased.

The tool function is normally indicated by the word address 'T'. This may have 2 or more digits depending upon the tool magazine capacity. Most common is the 2 digits such as T15. This causes the tool magazine position 15 or tool number 15 to be brought into the spindle replacing the already present tool in the spindle. The tool replaced from the spindle will be brought back to the empty position created when the tool 15 was loaded.

In machines where tool change is carried out manually, the word "T" will cause the stopping of the machine spindle and a light signal will appear indicating to the operator that he has to carry out the tool change, the order of which he must already have been instructed about.

Tool offset, to be discussed later, can also be programmed by using the same T word, e.g. T1513 which means tool No. 15 (i.e., tool located in the position 15 in the magazine) is to be loaded in the spindle and the value in offset register 13 is to be taken into account when this tool carries out the operation.

17.5.6 Preparatory Functions

This is denoted by '*G*'. It is a pre-set function associated with the movement of machine axes and the associated geometry. As discusses earlier, it has two digits, e.g. G01, G42, and G90 as per ISO specifications. However, some of the current day controllers accept up to 3 or 4 digits. In this lecture we will only discuss some of the regular functions. ISO has standardised a number of these preparatory functions also popularly called G codes. The standardised codes are shown below:

Code	**Function**
G00	Point-to-point positioning, rapid traverse
G01	Line interpolation
G02	Circular interpolation, clockwise (WC)
G03	Circular interpolation, anti-clockwise (CCW)
G04	Dwell
G05	Hold/Delay
G06	Parabolic interpolation
G07	Unassigned
G08	Acceleration of feed rate
G09	Deceleration of feed rate
G10	Linear interpolation for "long dimensions" (10 inches-100 inches)
G11	Linear interpolation for "short dimensions" (up to 10 inches)
G12	Unassigned
G13-G16	Axis designation
G17	XY plane designation
G18	ZX plane designation
G19	YZ plane designation
G20	Circular interpolation, CW for "long dimensions"
G21	Circular interpolation, CW for "short dimensions"
G22-G29	Unassigned
G30	Circular interpolation, CCW for "long dimensions"
G31	Circular interpolation, CCW for "short dimensions"
G32	Unassigned
G33	Thread cutting, constant lead
G34	Thread cutting, linearly increasing lead
G35	Thread cutting, linearly decreasing lead
G36-G39	Unassigned
G40	Cutter compensation-cancels to zero
G41	Cutter radius compensation-offset left

G42	Cutter radius compensation-offset right
G43	Cutter compensation-positive
G44	Cutter compensation-negative
G45-G52	Unassigned
G53	Deletion of zero offset
G54-G59	Datum point/zero shift
G60	Target value, positioning tolerance 1
G61	Target value, positioning tolerance 2, or loop cycle
G62	Rapid traverse positioning
G63	Tapping cycle
G64	Change in feed rate or speed
G65-G69	Unassigned
G70	Dimensioning in inch units
G71	Dimensioning in metric units
G72-G79	Unassigned
G80	Canned cycle cancelled
G81-G89	Canned drilling and boring cycles
G90	Specifies absolute input dimensions
G91	Specifies incremental input dimensions
G92	Programmed reference point shift
G93	Unassigned
G94	Feed rate/min (inch units when combined with G70)
G95	Feed rate/rev (metric units when combined with G71)
G96	Spindle feed rate for constant surface feed
G97	Spindle speed in revolutions per minute
G98-G99	Unassigned

Many of the control manufacturers follow these standard codes without altering the meaning. However, some manufacturers do change them to suit their way of programming.

It is generally possible to include more than one G address in one block, provided these functions are not mutually exclusive. For example, G02 and G03 (see details given later in this section) together in one block are normally not permissible. If they are given, the latter G code will become operational overriding the earlier from the same category. In Fanuc controls up to 5 G codes can be given in one block. However, in MAHO Philips 532 system only one G code needs to be given. Though this makes the reading of the program easier, it unnecessarily increases the number of blocks in a program and the subsequent increase in the size of the part program.

Another aspect that one should normally remember is that some of the G codes are modal, which means that they behave as settings to the control. Once given they remain operational till cancelled by another G code from the same group. A few other G codes are non-modal, which means that they remain operational in the block in which they are programmed. When we are describing the G codes, it will be made clear which is modal and which is not.

Also some of the G codes are default or turn on codes. This means that they are operational when the controller is started. Also when the program is completed, generally the controller is reset back to the original default settings. Hence care has to be taken by the programmer to understand the default codes in operation. This would also be mentioned along with the G code description.

A few of the usual preparatory functions which are generally present in all machining centres and are uniformly followed by all controller manufacturers are given below:

Motion group

*G00	Rapid Positioning
G01	Linear Interpolation
G02	Circular interpolation Clockwise
G03	Circular interpolation Counter clockwise

Dwell

G04	Dwell

Active plane selection group

*G17	XY Plane selection
G18	XZ Plane selection
G19	YZ Plane selection

Cutter compensation group

*G40	Cutter compensation, Cancel
G41	Cutter radius Compensation left
G42	Cutter radius Compensation right

Units group

*G70	Inch units
G71	Metric units

Hole making canned cycle group

*G80	Canned Cycle Cancel
G81-G89	Canned Cycles definition and ON

Co-ordinate system group

*G90	Absolute co-ordinate system
G91	Incremental co-ordinate system

Preset

G92	Absolute pre-set, Change the datum position

The * sign indicates the generally accepted default or turn on code in operation. However, some control manufacturers allow this to be modified to whatever suits them. The above is only a possible indication but not in any way standardised by ISO.

In the following we will give a description of the way to use these G codes.

Co-ordinate System Group, G90 and G91

The input of dimensional information can be done either in the absolute or in the incremental system. The preparatory function G90 is used for absolute programming. In absolute system, the dimensions are given with respect to a common datum chosen by the programmer. It must be programmed and can be cancelled by function G91 (and also when the program statement has the word M02 or M30). In Fig. 17.15, OX and OY are the datum.

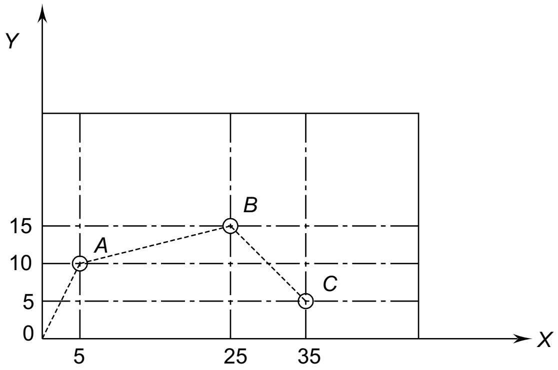


Fig. 17.15 *Absolute (G90) and incremental (G91) systems*

Whatever may be the route of the move, the programmed X and Y values of each position remain the same. Suppose the route to be 0-A-B-C, then

N007 G90 G01 X5.0 Y10.0 F...
N008 X25.0 Y15.0
N009 X35.0 Y5.0

This system is generally advisable for programming, as there are few chances of errors. When the tool is in a particular quadrant, such as the first, then all the co-ordinate values would be positive. In the third quadrant all values will be negative. In the second and fourth quadrants both positive and negative signs will be present. Thus it is suggested that the new programmers should always make use of the absolute system till they become familiar with the programming system.

The incremental type, denoted in the program by the word G91, is also available on all NC equipment. This is generally "Turn on mode" and can be cancelled by the word G90. The end of the program words M02 or M30 also sets it. In the incremental system, the data is incremental to the previous block. Unlike as with G90, the programmed data changes only if the route of the move is altered. Referring to the same figure (Fig. 17.15), the program for route 0-A-B-C would be

N007 G91 X5.0 Y10.0 F...
N008 X20.0 Y5.0
N009 X10.0 Y-10.0

If the route is changed, then the program changes accordingly. Incremental programming is useful when the features are dimensioned in a continuous chain, e.g., 5 holes, 31.250 mm apart along X axis would simply be programmed for each feature as X31.25; while in absolute programming, one would have to calculate the value for each position, i.e., X31.25, X62.5, X93.75,... It is also important when one follows the incremental programming system, to take care of the direction in which the movement is taking place, irrespective of the quadrant in which the tool is moving.

In a program, both the systems may be followed but it should be done carefully. In the incremental system any error done in a single block will carry forward and no correction can be done. Also the errors in the transmission system will result in error accumulation, while that does not happen in the case of absolute co-ordinate system.

Units Group, G70, G71

This group of codes specifies the units in which the program is to be interpreted. G70 stands for programming in inch units while G71 stands for programming in mm units. Any one of these can be made as turn on code depending upon the default units likely to be used. Most of the controls destined for areas other than North America would generally have default G71. This can be easily changed when necessary. In any case, it would be a better practice to make the habit of giving this code as the very first code in the part program.

A given program should be written only in either inch or mm units, but not both. Hence, only one of the two codes should be present in one program.

Active Plane Selection Group, G17, G18, G19

Some of the functions in NC control can only work in a plane rather than in all the three possible co-ordinate axes. This therefore requires the selection of active plane. This can be done by using these codes. The typical co-ordinate system and the corresponding plane labelling are shown in Fig. 17.16.

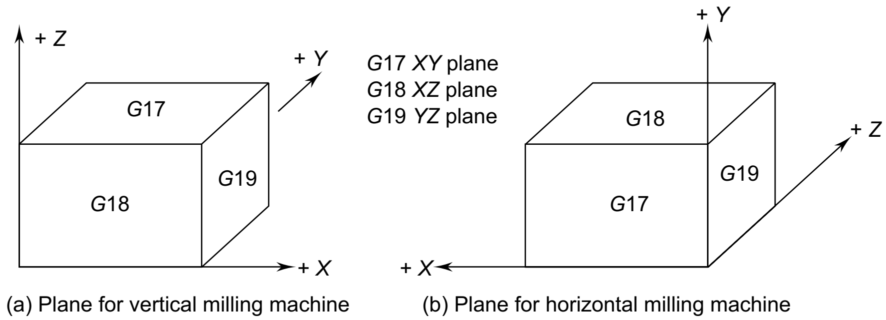


Fig. 17.16 *Plane selection for milling machines*

G17 XY Plane selection
This is the default turn on code. This allows for the working to be carried out in the horizontal plane in case of vertical axis milling machines as shown in Fig. 17.16(a). In the case of 2.5 axes machines, in a given block only X and Y co-ordinates are to be specified while the Z co-ordinates are to be specified in a separate block. For a horizontal axis machine, the working plane is the vertical plane perpendicular to the spindle axis.

G18 XZ Plane selection
This allows the working to be carried out in XZ plane. In the case of 2.5 axes machines, in a given block only X and Z co-ordinates are to be specified while the Y co-ordinates are to be specified in a separate block.

G19 YZ Plane selection
This allows the working to be carried out in YZ plane. In the case of 2.5 axes machines, in a given block only Y and Z co-ordinates are to be specified while the X co-ordinates are to be specified in a separate block.

Preset, G92

As described earlier, each of the machine tool has a separate machine reference point. However, this point is not very convenient to use as a co-ordinate datum for the part. Most of the NC machine tools allow for a 'Floating Datum' to be fixed anywhere in the machining envelope of the machine tool. As a result, the programmer can choose a convenient position on the part as datum, which may be referred to as 'Program

Zero Point'. The same will have to be communicated to the NC controller as datum. The choice of the datum as explained earlier is to suit either the setting of the component or to simplify the co-ordinate calculations.

It is necessary in the beginning to make the system understand the co-ordinate datum position of the part, which is different from the machine reference point. To do this, we make use of the G92 code. The part, which was pre-machined, is clamped at a suitable position on the machine table. A known tool or a setting mandrel of known diameter is kept in the machine spindle. This tool is then brought to a known position near or on the work piece blank, called the set point. The same is then programmed in the part program using the G92 code. For example, in Fig. 17.17, the work piece of dimensions 200 × 170 × 30 mm is located on machine bed with the longer edge along X-axis. The tip of the setting tool, held in the spindle, is made to touch the point B, i.e., the setting position. At that instant, the program block entered is

N015 G92 X-150.0 Y200.0 Z50.0

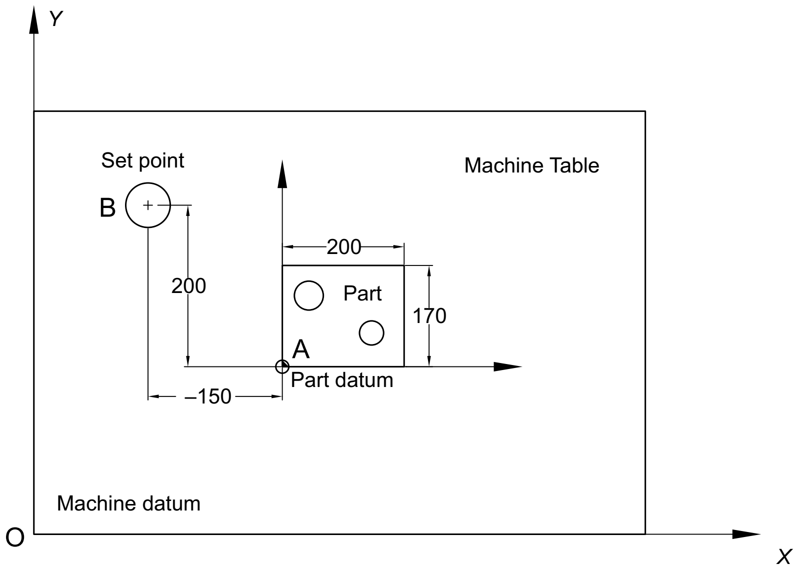


Fig. 17.17 *Setting the work piece on the machine table*

Depending upon the point being touched, the co-ordinate can be specified, taking the diameter of the probe touching the work piece. The tool tip is to be set at a distance of 50 from the top surface by means of a suitable gage.

The system automatically understands that point A is 0, 0, 0. It is obvious that it would be difficult to locate the point A otherwise. If the top surface is taken as Z = 0, and the setting is done at the point A, so N...G92 X0 Y0 Z0 would mean point A is –150, 200, –30. While using the function G92, no axis motion takes place. Using this function, the program zero point and the work co-ordinate system are set.

G53 to G56 are the other codes used for setting the programmable datum positions. These would allow for fixing a number of positions on the machine table whose co-ordinates can be entered into the controller as a permanent memory. When required their positions can be simply called by giving the particular G code in the program. This would also be useful for machining a batch of components all of which are located on the machine table each at the positions indicated by G53, G54, etc.

Motion Group, G00, G01, G02, G03

This is the most important group of codes used in part programming. G00 is the turn on code from this list. More explanation is given below:

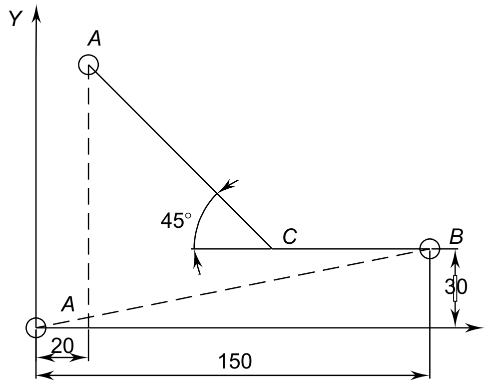


Fig. 17.18 *Positioning, preparatory function G00*

Rapid Positioning, G00 This is used for moving the tool at a rapid rate (normally the maximum available feed rate such as 8000 or 20 000 mm/min) along the axes involved for achieving the position programmed. The path taken by the tool to reach the programmed point is not important for this code.

This is a modal (stays active till cancelled by any other function of its family, i.e., G01, G02, G03) function and is also the 'turn on mode' (available as soon as the system is switched on or when a new program starts). Referring to Fig. 17.18, from position A, it is required to achieve position B.

This is typical to all machining situations when the tool has to be brought close to the component before any cutting commences. It is obvious that this movement is in the air (cutting air) and so to minimise the idle time, it should take place at the maximum feed rate of various slides involved. For this, the program block would be

N105 G90 G00 X150.0 Y30.0

It will be noticed that the initial path is at 45 degrees because both the X and Y slides move at the same feed rate (assuming the motors are of the same rating) till the required Y ordinate value is achieved, after which only the X slide moves till position B is achieved. This is one way of achieving the final position. There could be other possible methods implemented by different controllers.

Linear or Straight line Interpolation, G01 This is code is generally used when the material is to be cut using a feed rate. When the motion is desired along a straight line at a given feed rate, this function is used. It is modal. If a cut has to be made from D to E (Fig. 17.19) at a feed rate of 250 mm per minute, then the block would be

N115 G01 X110.0 Y30.0 F250

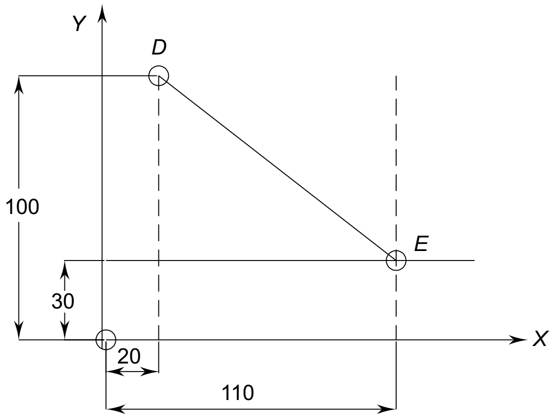


Fig. 17.19 *Linear interpolation, preparatory function G01*

In this case the controller moves all the three axes at a rate such that the resultant velocity along the line matches the programmed feed rate.

Circular Interpolation, G02/G03 When an arc is to be traversed in a plane, the function G02 or G03 is used if the direction of the motion is clockwise or anti-clockwise respectively, looking in the negative direction of the axis perpendicular to the plane. Referring to Fig. 17.20, when the motion is from F to G in XY plane, the program block would be, as per ISO,

N125 G02 X65.0 Y60.0 I35.0 J-10.0 F250

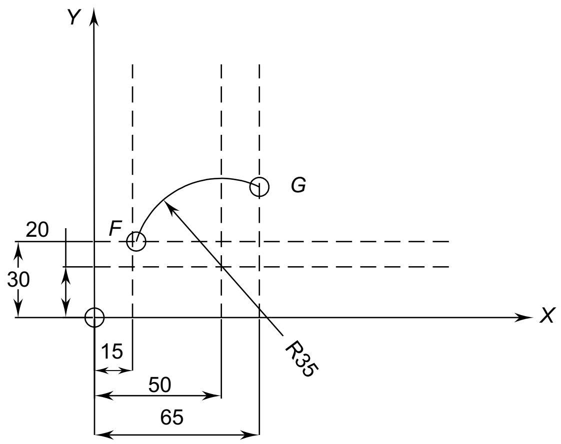


Fig. 17.20 *Circular interpolation, preparatory function G02/G03*

But, if the motion were from G to F, then it would be

N130 G03 X15.0 Y30.0 I-15.0 J-40.0 F250

Here, (X, Y) are the co-ordinates of the destination and (I, J) the distances, along the reference axes of the centre of the arc from the starting point of the arc. It is essential that the co-ordinates of the destination should be correct and within the prescribed limits. In some systems, (I, J) are the co-ordinates of the centre of the arc.

Some systems carry out circular interpolation when the value of the arc radius is given. The value is positive if the angle subtended by the arc at the centre is less than 180 degrees, and negative if otherwise. Assuming the radius and the angle subtended to be 40 mm and 100 degrees respectively, then

N130 G02 X65.0 Y60.0 R40.0 F250

It is possible to draw a complete circle, which would mean that the destination coincides with the starting point. Then

N310 G02 I35.0 J-10.0 F250

It will be noted that the destination need not be stated, since its co-ordinates are the same as that of the starting point already entered in the previous block. One should check up from the controller programming manual as to which procedure is to be followed. It may be noted that a full circle cannot be obtained with R-value.

Other G functions will be discussed later in this chapter.

Dwell, G04

This is to give a delay in the program. When the G04 code is encountered, the controller stops at that particular point for a specified time mentioned in the block. After that time the controller continues to execute the next block in the program. The delay time is normally mentioned in seconds using the X word address. In some controls, other than X may also be used. For example in Fanuc controls, P word address is used to specify the dwell time in milliseconds. An example is shown below:

N045 G04 X3.0

This calls for a stoppage of the control for a period of 3 seconds.

17.5.7 Miscellaneous Functions, M

These functions actually operate some controls on the machine tool and thus affect the running of the machine. Generally only one M code is supposed to be given in a single block. However, some controllers allow for two or more M codes to be given in a block, provided these are not mutually exclusive, e.g., coolant ON (M07) and OFF (M09) cannot be given in one block.

Less number of M codes have been standardised by ISO compared to G codes in view of the direct control exercised by these on the machine tool. The ISO standard M codes are shown below:

Code	**Function**
M00	Program stop, spindle and coolant off
M01	Optional programmable stop
M02	End of program-often interchangeable with M30
M03	Spindle on, CW
M04	Spindle on, CCW
M05	Spindle stop
M06	Tool change
M07	Coolant supply No. 1 on
M08	Coolant supply No. 2 on
M09	Coolant off
M10	Clamp
M11	Unclamp
M12	Unassigned
M13	Spindle on, CW + coolant on
M14	Spindle on, CCW + coolant on
M15	Rapid traverse in + direction
M16	Rapid traverse in – direction
M17–M18	Unassigned
M19	Spindle stop at specified angular position
M20–M29	Unassigned
M30	Program stop at end tape + tape rewind
M31	Interlock by-pass
M32–M35	Constant cutting velocity
M36–M39	Unassigned
M40–M45	Gear changes; otherwise unassigned

M46–M49	Unassigned
M50	Coolant supply No. 3 on
M51	Coolant supply No. 4 on
M52–M54	Unassigned
M55	Linear cutter offset No. 1 shift
M56	Linear cutter offset No. 2 shift
M57–M59	Unassigned
M60	Piece part change
M61	Linear piece part shift, location 1
M62	Linear piece part shift, location 2
M63–M67	Unassigned
M68	Clamp piece part
M69	Unclamp piece part
M70	Unassigned
M71	Angular piece part shift, location 1
M72	Angular piece part shift, location 2
M73–M77	Unassigned
M78	Clamp non-activated machine bed-ways
M79	Unclamp non-activated machine bed-ways
M80–M99	Unassigned

Some of the common miscellaneous functions often found in many controllers are:

M00

This would terminate the auto operation of the machine after completing the instructions in the block in which it has been programmed. This is called 'Program stop' and if it is required to continue with the rest of the program, the 'start' button on the console is to be pressed. This is useful for changing the clamp position or to carry out inspection of a particular dimension after a machining cut is taken. This being a pause function, it calls for the attention of the operator, delays the completion of the program and therefore should be avoided as far as possible.

M01

This is 'Optional stop' and stops the machine, as in the case of M00, only if the "Optional stop" switch on the controller console is 'ON'. This is useful when inspection is to be carried out on some components and not all in a given batch.

M02

This is 'End of program' and it causes the stopping of the machine and clearing of all the control registers. Another code M30 also does the same function.

M03

The miscellaneous function for machine spindle control for clockwise rotation. This starts the spindle to move in the clockwise direction at the speed set earlier using the S word address. When it is given in a block it is the first code to be executed before all the other codes in a block are acted upon.

M04

The miscellaneous function for machine spindle control for counter clockwise rotation. This starts the spindle to move in the counter clockwise direction at the speed set earlier using the S word address. When it is given in a block it is the first code to be executed before all the other codes in a block are acted upon.

M05

M05 is the miscellaneous function for stopping the machine spindle. When it is given in a block it is the last code to be executed after all the other codes in a block are acted upon.

M06

M06 is for tool change.

M07

M07 are for 'Coolant 1 On'

M08

M08 are for 'Coolant 2 On'

M09

M09 is for 'Coolant Off'.

M13

M13 the miscellaneous function for machine spindle control for clockwise rotation and the starting of the coolant simultaneously. This starts the spindle to move in the clockwise direction at the speed set earlier using the S word address. When it is given in a block it is the first code to be executed before all the other codes in a block are acted upon.

M30

M30 is similar to M02. It indicates 'End of tape' and 'tape rewind'. If a paper tape is used, the tape is rewind till the % sign is encountered. For machines working with RAM, the active program comes to the beginning. Many a times M02 and M30 are synonymous in operation in modern day controllers.

17.5.8 Program Number

In many of the latest CNC systems, there is a provision for labelling the program at the start itself, which facilitates searching from stored programs. The symbol used for the program number in Fanuc controls is "O" or ":", followed by its number. For example, O238 or :238. Such information does not interfere with the NC program.

Invariably, in most of the components there are a number of repetitive features, e.g., pattern of holes, profiles etc. Instead of writing blocks for each of them repeatedly in the program as per process plan, the facility exists for writing the sub programs for each feature and entering them with labels at the end of the main program. In the main program where these are required, they are called by an appropriate block e.g. in Fanuc controls, M98 P1001, i.e., miscellaneous function 98 is call for sub program (also called subroutine), the number after 'P' referring to the sub program being called. The sub programs are ended with the word 'M99' in Fanuc controls.

Example 17.1

The component to be machined is shown in Fig. 17.21. It is assumed that the pocket is through and hence only the outside is to be machined as a finish cut of the pocket. The tool to be used is a 20 mm diameter slot drill. If an end mill is to be used the program should be modified with a hole to be drilled at B first before the end mill is used. The setting is done with point A as reference (0, 0, 0) and the reference axes are along X and Y directions. A typical program, as per ISO (except the decimal point), for this would be:

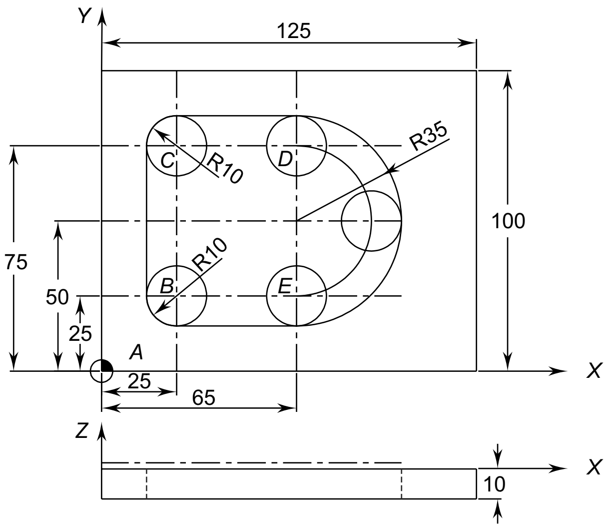


Fig. 17.21 *Example*

N001 G92 X0 Y0 Z0	absolute presetting at A
N002 G90	absolute programming
N003 G00 X25.0 Y25.0 Z2.0 T01 S3000 M03	tool brought rapidly at B, 2 mm above XY plane
N004 G01 Z-12.0 F120	tool goes down to full depth
N005 Y75.0	proceeds to C
N006 X65.0	proceeds towards right to D
N007 G02 Y25.0 I0 J-35.0	cuts curved profile till E
N008 X25.0	proceeds to B
N009 Z2.0	tool moves 2 mm above the XY plane
N010 G00 Z50.0 M05	spindle stops and rapidly moves up
N011 X0 Y0	rapid move to start position 0,0
N012 M30	end of program and tape rewind

17.5.9 Tool Length Compensation

In the programs discussed earlier, absolute pre-setting is performed with a tool and the co-ordinates for reference are registered. However, if the tool were replaced by another tool, say 20 mm shorter in length, then the tool movement would be 20 mm less along the Z axis if the same programmed values are used. So, the alternative is to modify the program every time the tool is changed.

In NC practice, all tools are measured in the assembled state and this information is always kept up to date (Fig. 17.22). For the tools being used, the difference in length, with respect to the pre-setting tool, is recorded and is manually entered and stored with the associated tool number. Whenever these tools are called into action by programmed instruction, the respective compensation values are activated and automatically taken into account in the tool motion. The following program example explains how tool length compensation is automatically taken care of.

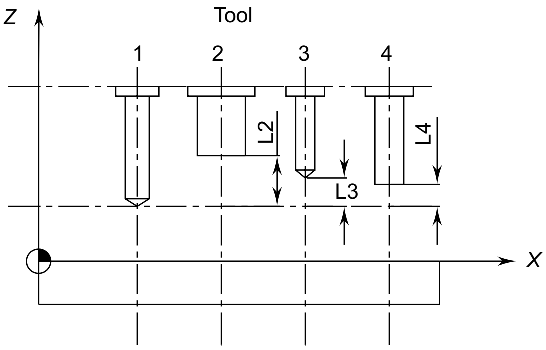


Fig. 17.22 *Tool length compensation*

```
---------------------
N003 M06 T01
---------------------
---------------------
N006 M06 T02
---------------------
---------------------
```

In these program blocks, M06 refers to tool change and T01, T02, ... refer to the tools to be loaded. Whenever the tool is ground or replaced, the new values are entered to replace the earlier ones and thus the program remains unchanged. This is an essential facility, without which the multiplicity of the programs for each job/tool combination would be enormous and futile. It will be understood that the values entered compensate for the difference in lengths and thus all tools "effectively" become independent of tool dimensions, if the dimensions of all the tools are stored. When programming, the tool dimensions are not considered since the compensation values are calculated by the control system itself during manufacturing.

What has been discussed so far are the illustrations to explain some of the general facilities available in CNC programming. These are in no way exact procedures for any control system and therefore it is imperative that the readers carefully browse through the programming manual of the system for which he has to write the programs. The number of facilities and the procedures to follow differ from system to system, as do their limitations.

All these details enable the programmer to know the operation motions, which he is to write in code form, i.e. the program. This would then be punched on a paper tape and the tape is fed into the tape reader of the machine controller. The program can also be entered directly on the machine control panel. The programs can also be prepared on microprocessor based terminals or personal computers and then be stored or transmitted directly to machine tools. The latter facility permits convenient editing. The programming section should maintain an updated list of tools available and information regarding machining parameters.

17.5.10 Cutter Radius Compensation

In profiling operations, one needs to calculate the tool path for preparing the program. This path refers to the spindle axis that is away from the profile required. Figure 17.23 shows the component and the tool path. Apart from the problem of calculating, one should realise that whenever the cutter size changes, the program

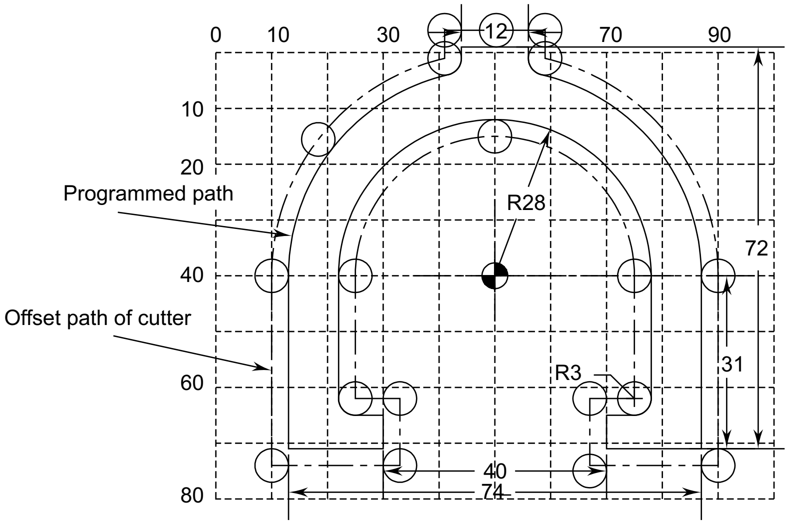


Fig. 17.23 *Cutter radius compensation*

would need editing. However, if a compensation equal to the radius of the cutter is entered and stored in the control system, then the program could be written for the component profile and thus no change in program would be required.

The preparatory functions G40, G41 and G42 are used for radius compensation and form one group. These are modal and the one programmed in any block remains active till cancelled by the other.

G40 Compensation 'off'.

G41 Used when the cutter is on the left of the programmed path when looking in the direction of the tool movement, i.e. the radius compensation is considered to the left of the programmed profile.

G42 used when the cutter is on the right of the programmed path when looking in the direction of the tool movement i.e. radius compensation is to the right of the programmed profile (In some systems, cutter diameter compensation is possible and in these cases, the value of the diameter is entered as the compensation value). The tool radius entry is always positive. If the programmed path is determined for a particular size of the cutter, the compensation value would be '+' or '–' depending on whether the cutter used for machining is oversized or undersized. The following program will illustrate the use of these preparatory functions. Fig. 17.24 shows the top view and the top of the component surface is taken as Z = 0.

N010	G92 X0 Y0 Z50.0	Absolute preset at 'A', 50 mm above work surface
N020	G90	Absolute programming
N030	M06 T1	
N040	Z2.0 M03 S600	Rapid plane
N050	G41 G01 X100.0 Y-60.0 F100	cutter compensation ON; tool moves along the path left of the programmed contour
N055 Z-5.0	Move to depth	
N060	X300.0	
N070	Y-260.0	

N080	X100.0	
N090	Y-60.0	
N100	G40 X0 Y0	cutter radius compensation OFF
N110	G00 Z50.0 T0 M05	compensation value for tool T1 cancelled
N120	M06 T02	(assuming a different tool is used for internal profile)
N130	X200.0 Y-120.0 Z2.0 S600 M03	
N140	G01 Z-5.0 F100	
N150	G42 Y-100.0	cutter compensation ON; tool path right of programmed contour
N160	G02 I0 J-60.0	
N170	G00 Z50.0 M05	
N180	G40 X0 Y0 T0 M02	

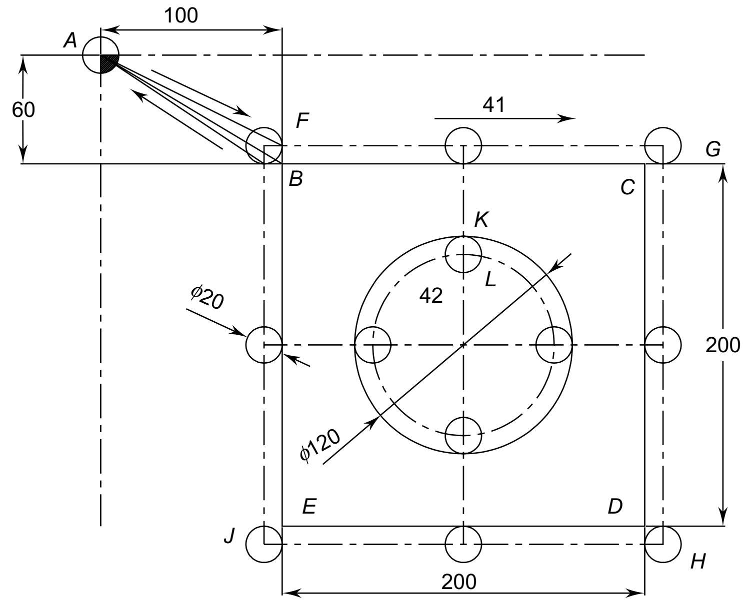


Fig. 17.24 *Example showing the cutter radius compensation using the G codes G41 and G42*

In the block N050, the programmed position is B, but the cutter centre comes to the position F such that BF is perpendicular to the next move BC (Block N060), programmed in the next block. AF is sometimes called ‘Ramp-on-move in view of its shape. Similarly, in the block N150, the programmed position is L but the cutter centre attains the position M, such that LM is perpendicular to the direction of the next move, i.e. tangent to the circle (Block N160). In the block N100, the compensation is deactivated by the word G40 which is entered after the programmed position B. The cutter stops at K, such that KB is perpendicular to the direction of the previous move i.e. EB. KA is called ‘Ramp - off - move’.

In the blocks N110 and N180, T0 causes cancellation of the compensation values for the tool in action at that time. Radius compensation is also used when similar profiles are to be cut with different depths of cut e.g. rough cut and finish cut. When a rough cut is made, a compensation equal to the thickness of the material to be left for finish is entered and during the finish cut this compensation is taken off. In view of identical programmed paths in roughing and finishing, such programming is done by using sub-routines. For the roughing cut, the sub-routine is called in the main program using the compensation and for the finish cut, the same sub-routine is called using the same tool without compensation.

Compensation for tool length and radius can be specific to a tool and if so, when the word T03 occurs in a block, it calls the tool number 3 into action with the compensation pre-registered for it. However, in many CNC systems the compensation values (MDI entry) are stored separately, irrespective of the tools being used. This helps in calling different compensation values even with the same tool when used on different occasions. For example, T0104 word would mean that tool number 01 would be used with the compensation value, entered in the register against identifier 04. Before commencing work on the machine, the operator must examine the compensation values stored and verify them with the list of values supplied to him. Negligence on this count could be very serious.

In some of the popular control systems, the pre-registered compensation values in the program block are called by the words D and H, which refer to the tool radius and length compensations respectively. For example,

```
N017 M06 T02
N018 G81 X170.0 Y100.0 Z65.0 R48.0 H07 F100 M03
```

would mean that the drilling operation will take place with tool Number 02 with a length compensation corresponding to the entry against the identifier 07. Similarly,

```
N074 M06 T06
N075 G01 X70.0 D03 F150 M03
```

would mean that milling will take place with tool number 06 with a radius compensation corresponding to the entry in the register against the identifier 03.

17.5.11 Canned Cycles

It is found many a times that a series of motions are to be repeated a number of times, many of which are fairly common to all the positions. For example, in the case of drilling operation the tool (twist drill) has to position a little above the hole in rapid position, then move to the required depth with the given feed rate and then the tool has to return to the top of the hole. The same actions are to be repeated for each of the holes. For each of the operation 3 NC blocks to be written, out of which two blocks need to be repeated without any change for each of the hole to be drilled in the same plane. It therefore is possible to define a canned cycle or fixed cycle which can repeat all these motions without having to repeat the same information for each of the hole. The most common cycles that would be useful are for the hole making operations such as drilling, reaming, tapping, etc. The advantages to be derived in using the canned cycles can be gauged by looking at the part programs below for the component shown in Fig. 17.25.

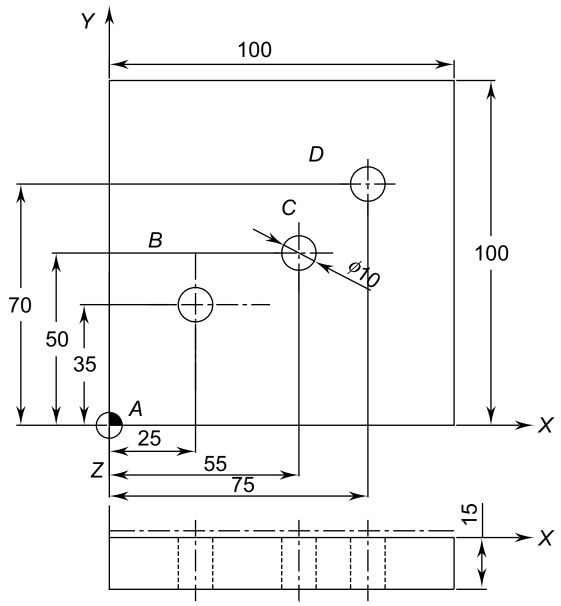


Fig. 17.25 *Example for canned cycles*

For the component shown in Fig. 17.25, the NC program for drilling the three holes without using canned cycles is shown below:

```
N010 G00 X25.0 Y35.0 Z2 *
N015 G01 Z-18.0 F125 *
N020 G00 Z2.0 *
N025 X25.0 Y35.0 *
N035 G01 X55.0 Y50.0 F125 *
N040 G00 Z2.0 *
N045 X25.0 Y35.0 *
N050 G01 X75.0 Y70.0 F125 *
N055 G00 Z2.0 *
N065 X0 Y0 Z50 *
```

For the same component the NC program using canned cycles is shown below:

```
N010 G81 X25.0 Y35.0 Z-18.0 R2.0 F125 *
N015 X55.0 Y50.0 *
N020 X75.0 Y70.0 *
N025 G80 X0 Y0 Z50 *
```

The motions embedded in various canned cycles as per ISO are shown in Table 17.1.

Table 17.1 Standard canned cycle motions

Canned Cycle Number	Feed from Surface	At Programmed Depth (End of Feed Point)			Used for
		Dwell	Spindle Speed	Spindle Return Motion	
G80	Off	—	Stop	—	Cancel canned cycle
G81	Constant	—	—	Rapid	Drilling, centre drilling
G82	Constant	Yes	—	Rapid	Counter sinking, Counter boring
G83	Intermittent	—	—	Rapid	Deep hole drilling
G84	Constant	—	Reverse	Feed	Tapping
G85	Constant	—	—	Feed	Reaming
G86	Constant	—	Stop	Rapid	Boring
G87	Constant	—	Stop	Manual	Multiple Boring
G88	Constant	Yes	Stop	Manual	Boring
G89	Constant	Yes	—	Feed	Boring

17.6 COMPUTER AIDED PART PROGRAMMING (CAP)

17.6.1 Concept of CAP

Preparing the part programs for CNC machine tools manually is a viable system for any kind of job. But the assistance of the computer would be desirable for part programming because of a variety of reasons. The first and foremost in this respect is the complexity of the work piece which makes manual part programming almost impossible. An example is the close tolerance contouring to some mathematically defined, or through a set of points other than a circular arc. This will require too many co-ordinate calculations making manual part programming too tedious to be practicable. Thus, simple repetitive and complex manual calculations involved in part programming would be taken care of by the computer, leaving the part programmer to attend to other functions to make a better part program. The reliability of the part program is enhanced as the computer makes all the calculations and thus the part programmer is less likely to make any errors. Besides, the computer would have some facilities for some error-detection to assist the part programmer in producing a better part program. The input language to the computer is a universal language akin to English and identical for all types of machine tool controllers. The part programmer is not burdened with having to learn about the idiosyncrasies and specific coding requirements of each of the machine tools, enabling him to handle the diverse array of machines and controls with ease. The part programming time is considerably reduced by as much as 75 percent, depending on the complexity of the job; in particular for components having repeated geometry in various locations. The computer, in addition to generating the valid NC codes, would also be able to provide additional useful information such as a plot of the cutter path, total machining time for a program reducing the tape proving costs.

The APT (Automatically Programmed Tools) language system originated at the Servomechanism laboratory of Massachusetts Institute of Technology, as did the first NC machine tool in 1952. This was the pilot study sponsored by the US Air Marshal Command, which resulted in the prototype system being released for the whirlwind computer in 1955. Though this version was an important step towards the computer preparation of tapes, the user still had to calculate the end points of each straight line cut to be performed by the machine tool. MIT, under the sponsorship of the Aerospace Industries Association, has released APT II for IBM 7040 wherein the complete job of part program preparation from the part drawing was undertaken by the computer. This version was continually developed until 1961, when the APT Long Range Program (ALRP) was created by the AIA and the job of keeping APT up-to-date was given to the Illinois Institute of Technology Research Institute, Chicago. In recognition of the role played by the computer in manufacture, over and above the simple guiding of the cutter tool along the work piece, the original sponsors have changed the ALRP in 1969 to Computer Aided Manufacture International or CAM-I. Now the work of CAM-I is done by IITRI as well as a large number of contractors all over the globe.

The APT NC reference language consists of a specially structured set of vocabulary, symbols, rules and conventions which are easily understood by the part programmer and would help him in faster preparation of control tapes. The vocabulary, which forms the mainstay of the reference language, is a carefully selected set of mnemonics chosen for their similarity in form and meaning with English. The computer translation of these English-like statements to the valid NC codes for any particular machine tool controller is generally carried out in two stages as shown in Fig. 17.26.

The first phase involves the conversion of input information into a generalised set of cutter location (CL) data and the relevant machine motions. At this stage, the output generated is the universally applicable cutter centre co-ordinates (called CLDATA, CLFILE, CLTAPE) which are independent of the machine tool on which the part is to be finally made. This set of programs is called processor and only one such processor is sufficient for any number of NC machine tools in the shop. The output of the processor contains information regarding the feed rates, spindle speeds, directions, coolant status, tool selection and other pieces of information which are machine tool/control unit-oriented, in addition to the cutter centre co-ordinates with respect to the work piece.

The second set of programs, called post processor, converts the generalised cutter location data into the specific control codes of the machine tool. As a result, the post processor is no more general like the processor but one each would be needed for every machine tool/ control system combination. The post processor would convert the cutter location data along with the machine tool-oriented information into the appropriate NC codes employed or the particular machine tool/ control system combination in question.

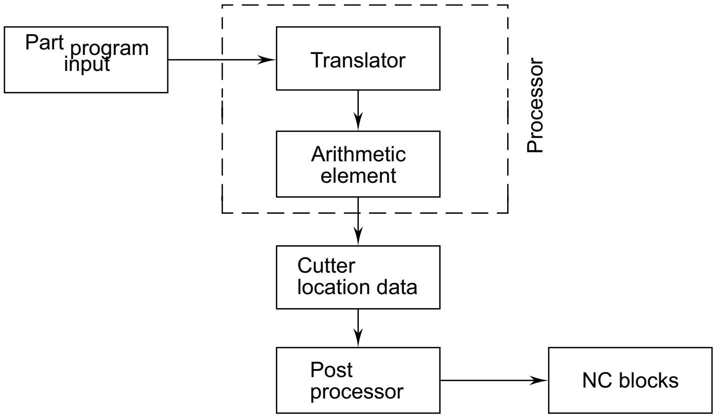


Fig. 17.26 *Computer aided part programming system (APT) configuration*

The two-pass preparation detailed above is most commonly used. The prime need for this is for making the part programming system more flexible. By taking out all the machine tool control unit-oriented information and making it a separate module, which would be far smaller compared to the main tape preparation system, one is able to achieve the desired generality. Since the machine tool oriented information is embedded in the post processor, which happens to be a much smaller segment in the overall tape preparation system, it is far more economical to duplicate for the various other machine tools which one may be willing to operate.

The various functions that can be attributed to the postprocessor are:

1. Converting the CLDATA to the machine tool co-ordinate system.
2. Converting the CLDATA to the control unit understandable NC blocks taking care of the following machine tool functions:
   - Maximum table or spindle traverses,
   - Available feeds and speeds,
   - Available preparatory, miscellaneous and other functions,
   - Straight line and circular interpolations,
   - Acceleration and decelerations of slides taking care of the overshoot of corners, and
   - Other machine tool control unit system requirements such as tape reader time, servo setting time, etc.
3. Provide output
   - Required control tape.
   - Diagnostic listing on line printer, and
   - Other operator/programmer instructions.

The modification of the main processor, which is very complex, will generally not be possible except by the people who had originally written it. Thus, it would be expedient for the users to develop expertise only in the writing or maintenance of the post processor, which is far simpler and requires much less manpower. Post processors are easier to write or modify. Further, if the same part is to be made on two or more different machines, then the first pass, i.e., the geometrical processing is to be done only once for creating the CLFILE which would be used to post process for all the machines concerned, thus saving the geometrical processing time.

A careful examination of its functions would reveal the fact that though the post processor has to be written for a particular machine tool control unit, there are many things which are common to all post processors. For example, the input and output sections are almost the same for all post processors because of the greater standardisation that has been achieved in these areas. Therefore, it seems that by modularising such common sections, it is possible to reduce a large amount of post processor writing time. Further standardisation is done by grouping classes of machine tools such as lathes, 2-axis machining centres, drills, punching machines, etc. which have a large amount of commonality between them. Thus, it is desirable to write an imaginary post-processor for a class of machine tools, which has all features possible to these machines, which is termed as the 'Universal Post-processor'.

17.6.2 Apt Language Structure

The APT language consists of many different types of statements made up of the following valid letters, numerals and punctuation marks.

Letters

ABCDEFGHIJKLMNOPQRSTUVWXYZ

Numerals

0123456789

Punctuation marks

/ A slash divides a statement into two sections. To the left of the slash are the MAJOR words, and to the right are the words, symbols and/or scalars that modify the word on the left of the slash so as to give it a complete and precise meaning or definition, e.g. GO/PAST, LN, TO, CS.

, A comma is used as a separator between the elements in a statement generally to the right of the slash.

= An equals is used for assigning an entity to a symbolic name, e.g. CI = CIRCLE/25, 50, 30.

Words

The words to be used in the statements are built up from one to six letters or numerals with the first one being a letter. No special character is allowed in the words.

Key Words

There are certain reserved names called key words in the language, which have a fixed meaning. These words cannot be used for any other purpose. A key word may be replaced by another name using a SYN statement. All key words consist of between two and six letters, without any numerals. The key words are divided into two classes, the MAJOR key words, which define the type of the statement, and the MINOR key words, which give the required parameters and modifiers.

Symbols

Symbols are the words used as substitutes for geometrical definitions and numerical values, where the first character must be a letter. A symbol must be defined before it is referenced in a subsequent part program statement.

Labels

Label names are used to reference a statement so that control can be transferred to that statement changing the usual linear execution sequence. Labels are identical to the words with the difference that all the characters in a label can be numerals. A label must be terminated by a right parenthesis.

Numbers

Numbers have their usual meaning as in algebra and are often referred to as scalars. No distinction is made between integer and real numbers.

The structure of statements used in APT part program is shown in Fig. 17.27.

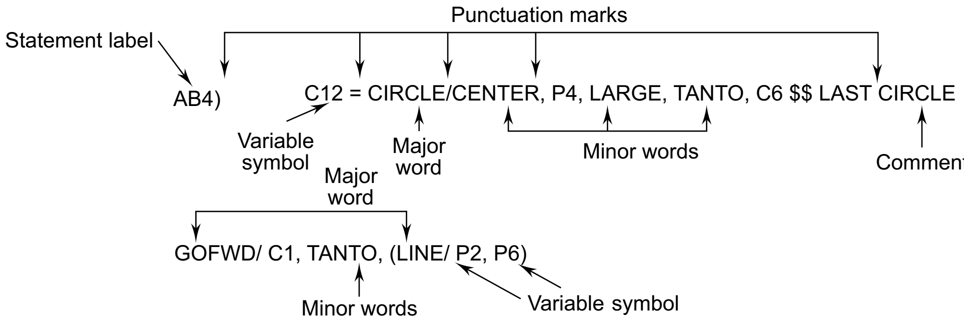


Fig. 17.27 *Structure of statements in APT*

17.6.3 APT program

The complete APT part program consists of the following four types of statements:

- Geometry
- Motion
- Post processor
- Compilation control

Geometry Commands

There are many ways in which the part geometry in APT could be specified. The part geometry is normally broken into a number of surface elements that could be defined from the data given in a part print. These are POINT, LINE, CIRCLE, PLANE, VECTOR, PATERN, SPHERE, TABCYL, etc. For each of the type of surface that can be defined, a number of alternative ways are possible for definition to simplify the definition procedure. A few examples are shown below:

Point The point has three co-ordinates along X, Y and Z-axes (Fig. 17.28). The Z co-ordinate when not specified is taken as either zero or the prevailing Z surface definition.

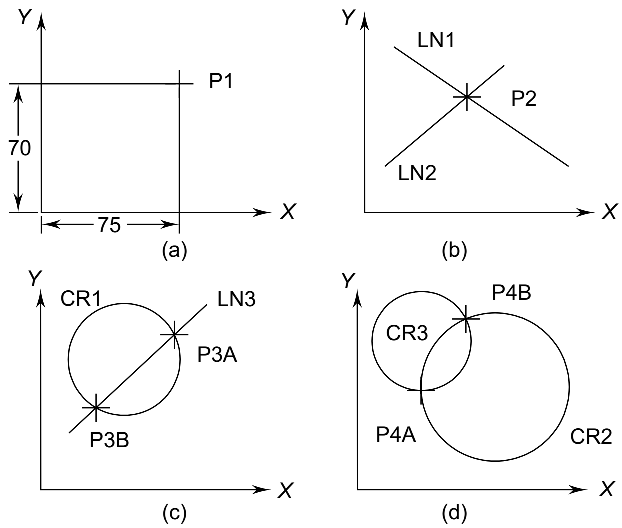


Fig. 17.28 *Point Definitions*

P1 = POINT/75.0, 70.0
P2 = POINT/ INTOF, LN1, LN2
P3B = POINT/ XSMALL, INTOF, LN3, CR1
P3A = POINT/ XLARGE, INTOF, LN3, CR1
P4A = POINT/ XSMALL, INTOF, CR2, CR3
P4B = POINT/ XLARGE, INTOF, CR2, CR3

Line Lines are considered to be of infinite length and do not have a direction. Lines must not be perpendicular to the XY plane (Fig. 17.29). Lines are considered planes perpendicular to the XY plane.

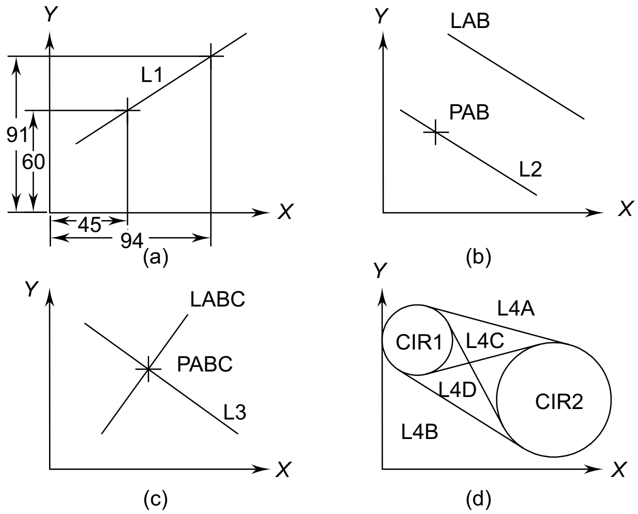


Fig. 17.29 *Line Definitions*

L1 = LINE/ 45, 60, 94, 91
L2 = LINE/ PAB, PARLEL, LAB
L3 = LINE/ PABC, PERPTO, LABC
L4A = LINE/ LEFT, TANTO, CIR1, LEFT, TANTO, CIR2
L4B = LINE/ RIGHT, TANTO, CIR1, RIGHT, TANTO, CIR2
L4C = LINE/ LEFT, TANTO, CIR1, RIGHT, TANTO, CIR2
L4D = LINE/ RIGHT, TANTO, CIR1, LEFT, TANTO, CIR2

Right and left is established by looking from the first circle specified in the definition to the second circle.

Circle A circle is always considered as a circular cylinder perpendicular to the XY plane of infinite length. The radius value when specified must not be negative (Fig. 17.30).

C1 = CIRCLE/ 61, 62, 37
C2 = CIRCLE/ CENTER, PT1, TANTO, LN1
C3 = CIRCLE/ PT4, PT2, PT3
C4A = CIRCLE/ YLARGE, LN2, YLARGE, LN3, RADIUS, 15
C4B = CIRCLE/ XSMALL, LN2, YLARGE, LN3, RADIUS, 15
C4C = CIRCLE/ YSMALL, LN2, YSMALL, LN3, RADIUS, 15
C4D = CIRCLE/ YSMALL, LN3, XLARGE, LN2, RADIUS, 15

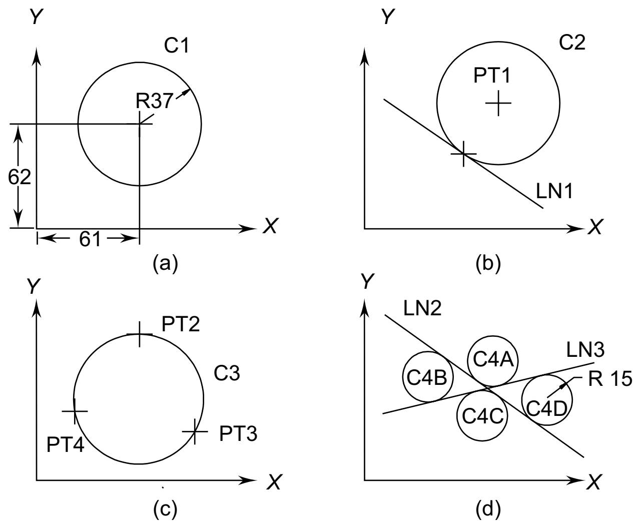


Fig. 17.30 *Circle Definitions*

The following few examples of APT geometries of components (views shown in XY plane only) have been defined:

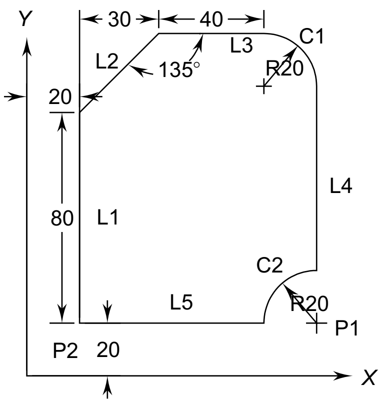


Fig. 17.31 *Example 1 for APT geometry definition*

PARTNO/ EXAMPLE 1 FIG. 17.31
P2 = POINT/ 0, 0
L1 = LINE/ 20, 20, 20, (20 + 80)
L2 = LINE/ (POINT/ 20, (20 + 80)), ATANGL, 45
P1 = POINT/ (20 + 30 + 40 + 20), 20
C2 = CIRCLE/ CENTER, P1, RADIUS, 20
L4 = LINE/ P1, PERPTO, (LINE/ XAXIS)
C1 = CIRCLE/ (20 + 30 + 40), (20 + 80 + 30 – 20), 20
L3 = LINE/(POINT/(20 + 30),(20 + 80 + 30)), PARLEL,
(LINE/XAXIS)

PARTNO/ EXAMPLE 2 FIG. 17.32
L4 = LINE/ XAXIS
L5 = LINE/ YAXIS
L3 = LINE/ PARLEL, L5, XLARGE, (20 – 5)
LM = LINE/ PARLEL, L5, XLARGE, (140/2)
C1 = CIRCLE/ (20 + 20), 22.5, 5
C2 = CIRCLE/ 20, (15 + 15), 5
C3 = CIRCLE/ 20, 15, 5
L1 = LINE/ LEFT, TANTO, C1, LEFT, TANTO, C3
L2 = LINE/ RIGHT, TANTO, C1, RIGHT, TANTO, C2
L3 = LINE/ PARLEL, L5, XLARGE, (20 – 5)
P1 = POINT/ CIRCLE, C1
P2 = POINT/ (20 + 20), (22.5 + 55)
P3 = POINT/ (20 + 100 – 20), (22.5 + 55)
P4 = POINT/ (20 + 100 – 20), 22.5
PAT= PATERN/ RANDOM, P1, P2, P3, P4

Motion Commands

This section of commands is more complex compared to the rest of the part program. The main function of these commands is to describe the actual machining sequence making use of the geometry elements defined earlier. The processor assumes that the tool moves around the work piece for the purpose of machining. If this were not true for any machine tool control unit, the post-processor would take care of the necessary conversions.

The motion commands can be broadly divided into three groups:

- Set-up commands,
- Point-to-point motion commands, and
- Continuous path motion commands.

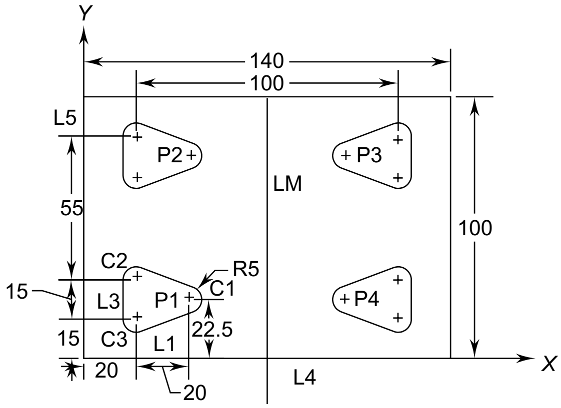


Fig. 17.32 *Example 2 for APT geometry definition*

Set-up commands These commands are used to identify the initial conditions for the specified machining. They are:

- FROM/ point
- CUTTER/ dia, radius
- INTOL/ dsval, psval, cs1val, cs2val
- OUTTOL/ dsval, psval, cs1val, cs2val
- TOLER/ dsval, psval, cs1val, cs2val

Point-to-point motion commands These are used to specify the positioning commands used for point to point applications such as drilling operation. They are:

- GODLTA/ dx, dy, dz
- GOTO/ x, y, z
- GOTO/ point
- GOTO/ patern

Continuous path motion commands These are used to specify the continuous path motion involving milling and turning operations to generate a variety of surfaces. The desired path where the cutting tool (cutter) is in continuous contact with the work piece surface is described by means of three intersecting surfaces. These surfaces are designated as drive surface (ds), part surface (ps) and check surface (cs) which are specified in that order in the motion statement. For any motion to be described, all the three surfaces are to be designated either explicitly or otherwise. The cutting tool is expected to move along the intersection of the part and drive surface till it is stopped by means of the check surface.

Part surface is in continual contact with the tool tip and helps in control of the depth of cut. Drive surface is the other surface with which the cutting tool is in continual contact during a given motion. The tool periphery or tool axis follows the drive surface. Check surface is the one which limits the given motion statement.

PSIS/ ps	Part surface is
AUTOPS	Automatic part surface (current Z level)
NOPS	No part surface
GO/ TO, ds, TO, ps, TO, cs	Start-up command for continuous path

The modifier TO can be replaced by either ON, PAST or TANTO depending upon the cutter relationship with the respective surface.

GOLFT/ ds, TO, cs	Contour Motion command Go to left
GORGT/ ds, TO, cs	Contour Motion command Go to right
GOFWD/ ds, TO, cs	Contour Motion command Go forward
GOBACK/ ds, TO, cs	Contour Motion command Go back
GOUP/ ds, TO, cs	Contour Motion command Go up
GODOWN/ ds, TO, cs	Contour Motion command Go down

In the following example, complete motion statements are presented for Fig. 17.31:

```
PARTNO/ EXAMPLE 1 FIG. 17.31
FROM/ 0, 0, 50
CUTTER/ 20
TOLER/ 0.01
GO/L1, (PL1 = PLANE/ 0, 0, 1, 3), L5
AUTOPS
GODLTA/ -8
TLLFT, GOLFT/ L1, PAST, L2
GORGT/ L2, PAST, L3
GORGT/ L3, TANTO, C1
GOFWD/ C1, TANTO, L4
GOFWD/ L4, PAST, 1, INTOF, C2
GORGT/ C2, PAST, L5
GORGT/ L5, PAST, L1
GODLTA/ 8
GOTO/ 0, 0, 50
```

Post Processor Commands

These commands are used to specify the machine tool functions and are supposed to be acted upon by the post processor identified earlier in the part program.

```
COOLNT/ ON            To specify the cutting fluid requirement
        OFF
        MIST
        FLOOD
        TAPKUL
FEDRAT/ IPM, feed
        MMPM
        IPR
        MMPR

LOADTL/ toolno, magpos
REWIND
```

SPINDL/ RPM, n, CLW
SFM CCLW
SMM

Compilation Control Commands

This segment covers statements that normally prepare the computer for accepting the part program, improve the readability of the part program and control the output of the computer.

PARTNO/ <literal string>

The PARTNO is used as identification for the part program and as such should be the very first statement in a part program. It should also be noted that in a given line PARTNO should appear in the first six character positions without any blanks. The length of <Literal string> will depend on the particular implementation, e.g. PARTNO/LEVER 138C

The FINI statement is the physical end of a part program and should always be present at the end of any part program.

MACHIN/name, <parameters>

The MACHIN statement identifies the post processor to be used for outputting the necessary NC blocks. Name signifies the name of the particular post processor identification name in literal strings, e.g. EXPOST, followed by < parameters > which initialises the various flags in the post processor program. The number of such parameters depends upon a particular implementation.

e.g. MACHIN/EXPOST, 2,2,1,OPTION, 0

When no post processing of CLDATA is required, then the following statement may be specified which would suppress the post processing.

NOPOST

To add comments in a part program, the command REMARK may be used as shown:

REMARK/GEOMETRICAL DEFINITIONS

This statement is ignored by the processor and as such is identical to a $$ in usage, the only difference being that REMARK should be present only in the first six character positions of a line.

Repetitive Programming

Just as DO loop, subroutines and macros are used in manual part programming, similarly facilities are available for repetitive programming in computer aided programming. These are described here.

Looping Normally, a part program is executed sequentially starting from a PARTNO statement to the FINI statement. But it would be possible to change this sequential execution by means of the transfer statements available in APT.

JUMPTO unconditional transfer
IF conditional transfer

The usage is

JUMPTO/ lbl1

A better option for looping is the arithmetic IF statement which allows a conditional transfer to a segment of the program depending on the value of an arithmetical expression. The general usage is

IF (< expression>) lb11, lb12, lb13

When the numerical value of the < expression > is negative, zero or positive, then control is transferred to the statement referenced by lb11, lb12, or lb13 respectively. It is always necessary to label the statement which immediately follows the IF statement in a part program. The <expression > could be a variable or an arithmetic expression.

```
         X= 0
LB0)     YVAL = 20
LB1)     GOTO/X, YVAL, 0
         GODLTA/-10
         GODLTA/10
         YVAL = YVAL+30
         IF (500 -YVAL) LB2, LB1, LB1
LB2)     X=X + 50
         IF (500 -X) LB3, LB, LB0
LB3)     -----------------
```

Macro The sequence of similar or identical statements which need to be referred more often in a part program are best referred by a MACRO statement such that the part program bulk is reduced. This statement is very similar to a FORTRAN SUBROUTINE statement. The syntax is

```
<name >= MACRO/<parameters>
-----------
-----------
TERMAC
```

All the statements that are enclosed between a MACRO statement and a TERMAC statement are to be executed whenever this macro is called by

```
CALL/name, <parameters>
```

Tracut The result of TRACUT usage in motion statements is to TRAnspose the CUTter locations only without actually altering the original geometrical definitions. This is useful particularly for repetitive geometries. The usage is

```
TRACUT/matrix
--------------
--------------
TRACUT/NOMORE
```

17.6.4 Complete Part Program in APT

In the following is shown the example of a complete part program for the part shown in Fig. 17.32. The machining of the four pockets involves the following steps:

- Centre drill four holes P1, P2, P3 and P4
- Drill of these holes through
- Rough mill of each pocket to depth of 3 mm leaving 0.5 mm for finishing
- Rough mill to entire depth
- Finish mill of complete pocket, and
- Repeat steps (iii) to (v) for the other three pockets.

The complete part program is presented below with the necessary remarks to facilitate understanding.

```
PARTNO MILLING EXERCISE ON MOOG HYDRAPOINT
REMARK PROGRAMMER P.N.RAO
MACHIN/63, 3, 2, 1, 0 OPTION, 0
MACHIN/40, OPTION, 3, 1, 1, 2, 2, 2, 2, 2, 3          $$ PLOTTER
PPLOT/ALL,LOWLFT,-60,0,0,XYPLAN, UPRGT, 120, 160, 0, SCALE, 0.04
PRINT/ON                                              $$ TO GET CANONICAL INFORMATION
CLPRNT/ON                                             $$ TO GET CENTRE LINE DATA
TOLER/0.01
REMARK GEOMETRY STATEMENTS
L4 = LINE/ XAXIS
L5 = LINE/ YAXIS
L3 = LINE/ PARLEL, L5, XLARGE, (20 - 5)
LM = LINE/ PARLEL, L5, XLARGE, (140/2)
C1 = CIRCLE/ (20 + 20), 22.5, 5
C2 = CIRCLE/ 20, (15 + 15), 5
C3 = CIRCLE/ 20, 15, 5
L1 = LINE/ LEFT, TANTO, C1, LEFT, TANTO, C3
L2 = LINE/ RIGHT, TANTO, C1, RIGHT, TANTO, C2
L3 = LINE/ PARLEL, L5, XLARGE, (20 - 5)
P1 = POINT/ CIRCLE, C1
P2 = POINT/ (20 + 20), (22.5 + 55)
P3 = POINT/ (20 + 100 – 20), (22.5 + 55)
P4 = POINT/ (20 + 100 – 20), 22.5
PAT= PATERN/ RANDOM, P1, P2, P3, P4
M1=MATRIX/ TRANSL, 0, 55, 0                           $$ FOR POCKET 2
M2=MATRIX/ MIRROR, LM                                 $$ FOR POCKET 4
M3=MATRIX/XYROT,180,TRANSL,(20+120),15+15+15 + 55)$$FOR POCKET 3
PLN = PLANE/ 0, 0, 1, 2                               $$ CLEARANCE PLANE
TPP = POINT/ –50, 150                                 $$ START POINT
REMARK MOTION STATEMENTS START HERE
CLRSRF/ (76 + 19.5)                                   $$ TOOL ABOVE WORK
FROM/ TPP
REMARK CENTRE DRILLING
TOOLNO/ 1, LENGTH, 19.5
SPINDL/ RPM, 2000, CLW
COOLNT/ MIST
CYCLE/ DRILL, 8, MMPM, 160, 3
GOTO/PAT
COOLNT/OFF
CYCLE/OFF
GOTO/TPP
TOOLNO/ 3, LENGTH, 85
SPINDL/ RPM, 1500, CLW
COOLNT/ ON
CYCLE/ DRILL, 10.5, MMPM, 200, 3
```

```
GOTO/ PAT
COOLNT/ OFF
CYCLE/ OFF
GOTO/ TPP
REMARK MILLING OF POCKETS
TOOLNO/ 4, LENGTH, 20.5              $$ MILL TOOL
SPINDL/ RPM, 1000, CLW
COOLNT/ ON
MAC1 = MACRO/                        $$ MACRO FOR GOING ROUND
FEDRAT/ MMPM, 60                     $$ THE POCKET ONCE
TLRGT, GORGT/ L1, TO, L3
GORGT/ L3, TO, L2
GORGT/ L2, TO, L1
TERMAC
MAC2 = MACRO/                        $$ MACRO FOR COMPLETE MACHINING
RAPID                                $$ OF A POCKET
CUTTER/ 11                           $$ LEAVE 0.5 FOR FINISHING
GO/ L1, PLN, PAST, L2
CUT
CYCLE/ MILL, 3, MMPM, 60             $$ MILL TO 3 DEPTH
CALL/ MAC1
CYCLE/ MILL, 8                       $$ MILL FULL DEPTH
CALL/ MAC1
CUTTER/ 10                           $$ FINISH THE POCKET
GO/ L1, PLN, L2
CALL/ MAC1
CYCLE/ OFF
DNTCUT
RAPID
GOTO/ TPP                            $$ THEORETICAL MOTION ONLY
TERMAC
CALL/ MAC2                           $$ MACHINE POCKET 1
TRACUT/ M1
CALL/ MAC2                           $$ MACHINE POCKET 2
TRACUT/ NOMORE
TRACUT/ M3
CALL/ MAC2                           $$ MACHINE POCKET 3
TRACUT/ NOMORE
TRACUT/ M2
CALL/ MAC2                           $$ MACHINE POCKET 4
TRACUT/ NOMORE
CUT                                  $$ LAST MOTION STATEMENT
REWIND
FINI
```

After entering the program, a printout of the APT processor can be obtained showing the canonical information of all the geometry defined and the cutter location data. A plot of the CLDATA is also obtained to prove the validity of the program.

SUMMARY

With changes in the machining requirements, the conventional automation procedures are no longer useful. It is necessary to provide soft automation to cater to the varying needs of the machining industry.

- Numerical control is a process of controlling the machining operations by the use of numbers and words following a strict syntax.
- Numerical control was developed to cater to the complex machining needs of the aerospace industry, but quickly adopted by all the manufacturing industries.
- Numerical control machine tools are useful for small volume, high accuracy and complex machining geometries, for reducing the processing time as well as the cost.
- Numerical control operation utilises feedback control to maintain the accuracies.
- The Z-axis of NC machine is designated as the main spindle axis while the X- and Y-axes are perpendicular to it.
- Part programming starts with the selection of the datum, preparing the coordinates through which the tool centre should be moving (CLDATA) to generate the required geometry and then translating the CLDATA to the word address format.
- Preparatory functions or G codes specify the geometric nature of motions of the tool while the miscellaneous or M codes specify the machine tool functions.
- Tool length and diameter compensation helps in simplifying the part program involving multiple tools.
- Canned cycles combine a number of motions that need to be repeated a number of times to simplify programming.
- Computer aided part programming is used for complex geometries that require a large amount of calculations to get the CLDATA points. APT is the first computer aided programming language, which is similar to FORTRAN and has geometry, motion and post processor commands in the program.

Questions

17.1 Briefly explain the principle of Computer Numerical Control for machine tools. Also mention its applications.

17.2 Describe four main features of CNC machines which distinguish them from conventional machine tools.

17.3 Explain the advantages and limitations of numerical control of machine tools.

17.4 Explain with the help of illustrations the principles of ISO designation of CNC machines.

17.5 Show the axes of a CNC horizontal Boring Machine.

17.6 Explain the concept of 'Floating Datum' and 'Set Point' with reference to CNC Part programming. What is their relationship? Explain how they are used in programming in ISO Format.

17.7 Explain with neat sketch the operation of the canned cycle G81 as per ISO.

17.8 What are the various functions embedded in the G82 canned cycle? Explain their use with examples in ISO Format.

17.9 What do you understand by the word "canned cycle" in manual part programming? Explain with neat sketches the differences between the operation of the canned cycles G81 and G83.

17.10 How is cutter compensation given in the case of a machining centre? Explain with the help of an example how it is operational. Specify any of the limitations in using this facility.

17.11 Explain the concept and need for a post processor as used in computer assisted part programming systems such as APT. Describe the functions of a post processor.

17.12 What are the basic assumptions made while programming in APT language?

Problems

17.1 For the following component (Fig. 17.33) make a part program on a machining centre equipped with ISO standard controller. The work material is AISI 1040 steel. Clearly show the set point and axes on the sketch of the part. Prepare also the planning sheet as used in the laboratory. Show all the calculations that are necessary.

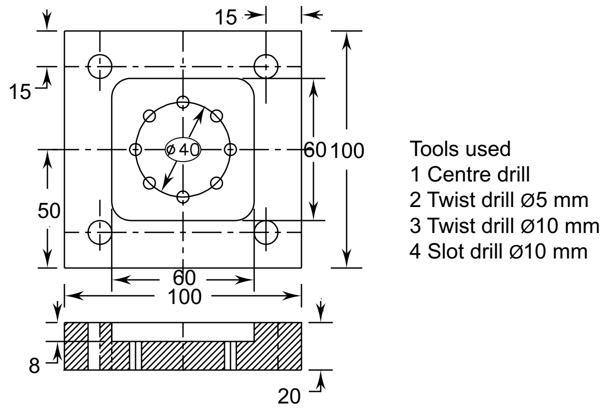


Fig. 17.33

17.2 Examine the following CNC part program for a machining centre equipped with a controller following ISO standard. Identify any errors found in the program and also explain the errors.

```
%
N7001 *
N1 G71 *
N2 G90 *
N3 T1 M6 *
N4 G0 X75.0 Y100.0 *
N5 G1 Z-3 *
N6 X175.0 F100 *
N7 Y25.0 *
N8 X75.0 *
N9 Y100.0 *
N10 M30*
```

For the above program prepare the geometry of the part generated, if the diameter of the slot drill used is 10.0 mm.

For the program shown in question 2 above, prepare the geometry of the part generated, if the diameter of the slot drill used is 5 mm.

17.3 For the following component (Fig. 17.34) make a part program on a vertical axis machining centre. Clearly show the set point and axes on the sketch of the part. Prepare also the planning sheet as used in the laboratory.

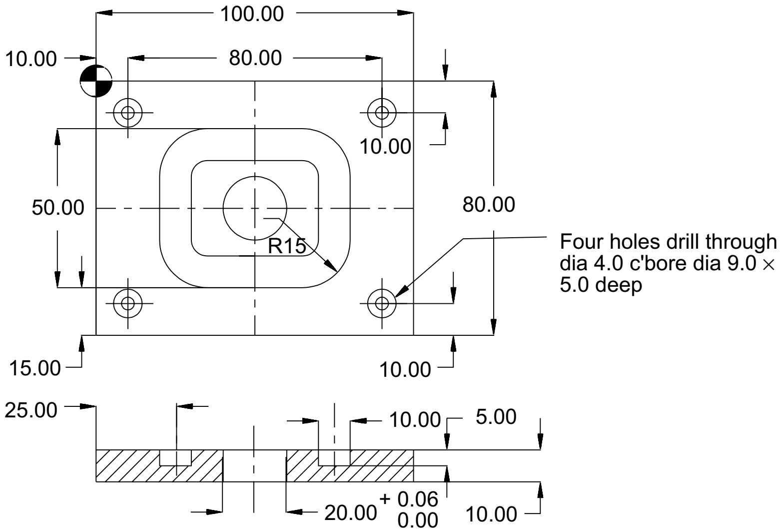


Fig. 17.34

17.4 Examine the following CNC part programs. Give the errors in the programs and also explain the errors.

(a)
```
%
O7001 *
N2 G0 X3.0 Y4.0 *
N3 G1 X7.0 F100 *
N4 Y1.0 *
N5 X3.0 *
N6 Y4.0 *
N10 M02 *
```

(b)
```
%
O9401*
N1 G17 T1 M6 *
N2 G92 X90.0 Y70.0 *
N3 G81 Y2.0 Z-10.0 F200 S500 M3*
N5 G01 X4.0 Y12.0 F150 *
N10 M02 *
```

17.5 Explain the mistakes found in the following statements.

(a) N05 G01 X12.3 Y23.0 F120 *

(b) N25 G04 X2.0 O1234 *

(c) N45 G00 T1001 S400 *

(d) N75 G03 X0 Z0 F120 *
(e) N60 M01 T1000 Y0 *

17.6 For the component shown in Fig. 17.35 make a part program on a machining centre equipped with the ISO controller. Clearly show the set point and axes on the sketch of the part. Show all the necessary calculations.

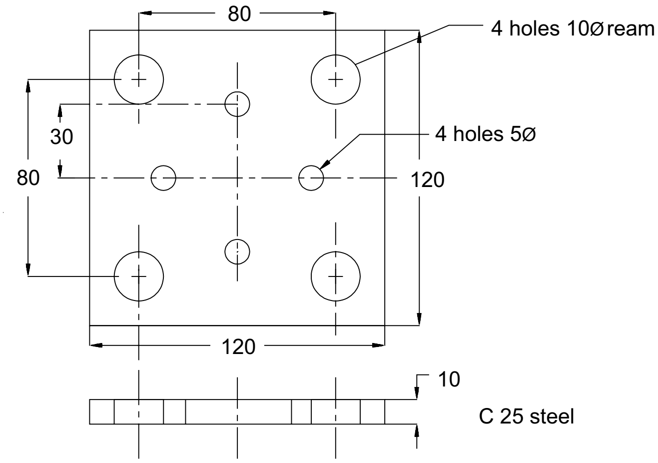


Fig. 17.35

17.7 For the following component (Fig. 17.36) write the geometry statements using the APT language. As far as possible make use of appropriate and simple APT geometry statements.

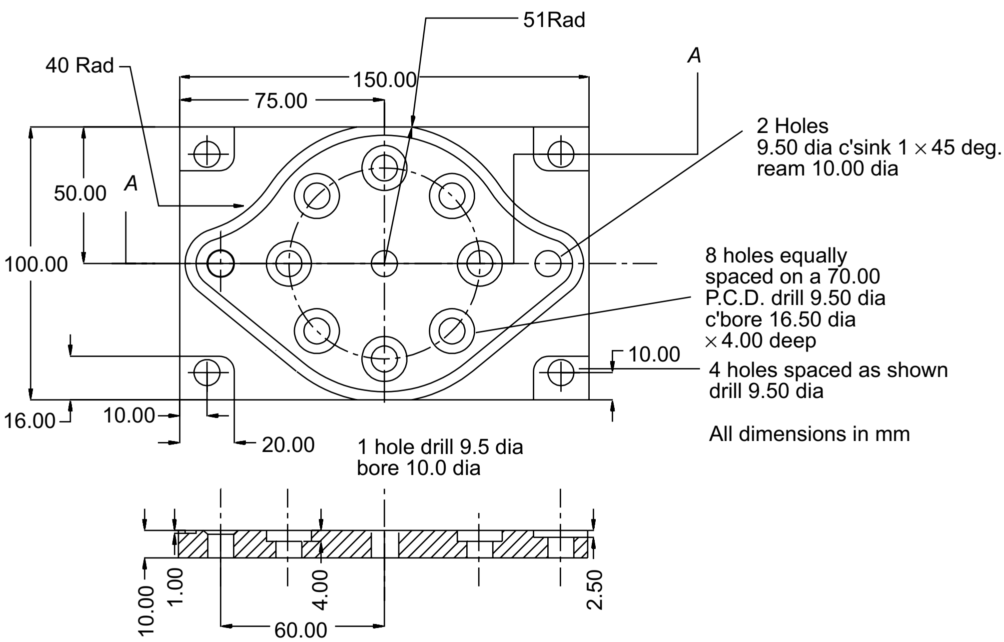


Fig. 17.36

Multiple Choice Questions

17.1 The type of applications where NC machines can be used profitably
- (a) For jobs requiring many set-ups and/or the setups very expensive
- (b) For jobs requiring very high accuracy and repeatability.
- (c) For the parts having complex contours, that cannot be manufactured by conventional machine tools.
- (d) All of the above

17.2 Advantage of numerical control machining is
- (a) Faster manufacturing
- (b) Parts requiring frequent design changes can be done
- (c) One-off production can be produced accurately
- (d) All of the above

17.3 Disadvantage of numerical control machining is
- (a) Skill of the required is high
- (b) Higher investment in equipment costs
- (c) Special training required for the operator and maintenance personnel
- (d) All of the above

17.4 For which application a point-to-point numerical control system can be used
- (a) A lathe machine
- (b) A milling machine
- (c) A punching press
- (d) None of the above

17.5 Numerical control manufacturing can be useful for
- (a) Only for mass production
- (b) Only for small batch production
- (c) Only for single piece production
- (d) None of the above

17.6 While specifying the axes of CNC machine tool (as per the ISO standards), the spindle axis is considered as
- (a) X-axis (b) Y-axis
- (c) Z-axis (d) A-axis

17.7 "The need for Jigs and fixtures is completely eliminated when using numerical control." This statement is
- (a) True
- (b) False
- (c) True when dealing with really complex parts
- (d) True when dealing with hard materials

17.8 Preparatory functions in a CNC program (in word address format) are identified by the letter
- (a) A (b) G
- (c) P (d) F

17.9 Using word address format, the centre of an arc in XY plane using circular interpolation is programmed using the letters
- (a) I and J (b) I and K
- (c) J and K (d) C and R

17.10 Using word address format, the centre of an arc in XZ plane using circular interpolation is programmed using the letters
- (a) I and J (b) I and K
- (c) J and K (d) C and R

17.11 Using word address format, the centre of an arc in YZ plane using circular interpolation is programmed using the letters
- (a) I and J (b) I and K
- (c) J and K (d) C and R

Answers to MCQs

17.1 (d)	17.2 (d)	17.3 (d)	17.4 (c)	17.5 (d)
17.6 (c)	17.7 (b)	17.8 (b)	17.9 (a)	17.10 (b)
17.11 (c)				

GATE Previous Years' Questions

CHAPTER 2

2.1 In an orthogonal cutting process the tool used has rake angle of zero degree. The measured cutting force and thrust force are 500 N and 250 N, respectively. The coefficient of friction between the tool and the chip is ____________.

(GATE-2016-ME-SET-1, 1-Mark)

2.2 The tool life equation for HSS tool is $VT^{0.14} f^{0.7} d^{0.4}$ = Constant. The tool life (T) of 30 min is obtained using the following cutting conditions:

$V = 45$ m/min, $f = 0.35$ mm, $d = 2.0$ mm

If speed (V), feed (f) and depth of cut (d) are increased individually by 25%, the tool life (in min) is

(a) 0.15 (b) 1.06
(c) 22.50 (d) 30.0

(GATE-2016-ME-SET-1, 2-Marks)

2.3 The following data is applicable for a turning operation. The length of job is 900 mm, diameter of job is 200 mm, feed rate is 0.25 mm/rev and optimum cutting speed is 300 m/min. The machining time (in min) is ____________.

(GATE-2016-ME-SET-2, 1-Mark)

2.4 For an orthogonal cutting operation, tool material is HSS, rake angle is 22°, chip thickness is 0.8 mm, speed is 48 m/min and feed is 0.4 mm/rev. The shear plane angle (in degrees) is

(a) 19.24 (b) 29.70
(c) 56.00 (d) 68.75

(GATE-2016-ME-SET-3, 2-Marks)

2.5 In a single point turning operation with cemented carbide tool and steel work piece, it is found that the Taylor's exponent is 0.25. If the cutting speed is reduced by 50%, then the tool life changes by ______________ times.

(GATE-2016-ME-SET-3, 2-Marks)

2.6 Under certain cutting conditions, doubling the cutting speed reduces the tool life to $(1/16)^{th}$ of the original. Taylor's tool life index (n) for this tool-work piece combination will be ______________.

(GATE-2015-ME-SET-1, 1-Mark)

2.7 An orthogonal turning operation is carried out under the following conditions: rake angle = 5°, spindle rotational speed = 400 rpm; axial feed = 0.4 m/min and radial depth of cut = 5 mm. The chip thickness t_c, is found to be 3 mm. The shear angle (in degrees) in this turning process is ______________.

(GATE-2015-ME-SET-1, 2-Marks)

2.8 A single point cutting tool with 0° rake angle is used in an orthogonal machining process. At a cutting speed of 180 m/min, the thrust fore is 490 N. If the coefficient of friction between the tool and the chip is 0.7, then the power consumption (in kW) for the machining operation is ______________.

(GATE-2015-ME-SET-2, 2-Marks)

2.9 In a machining operation, if the generatrix and directix both are straight lines, the surface obtained.

(a) Cylindrical
(b) Helical

(c) Plane
(d) Surface of revolution

(GATE-2015-ME-SET-3, 1-Mark)

2.10 Orthogonal turning of a mild steel tube with a tool of rake angle 10° carried out at a feed of 0.14 mm/rev. If the thickness of the chip produced is 0.28 mm, the values of shear angle and shear strain will be respectively
(a) 28° 20' and 2.19
(b) 22° 20' and 3.53
(c) 24° 30' and 3.53
(d) 37° 20' and 5.19

(GATE-2015-ME-SET-3, 2-Marks)

2.11 During pure orthogonal turning operation of a hollow cylindrical pipe, it is found that the thickness of the chip produced is 0.5 mm. The feed given to the zero degree rake angle tool is 0.2 mm/rev. The shear strain produced during the operation is ________.

(GATE-2014-ME-SET-1, 2-Marks)

2.12 If the Taylor's tool life exponent n is 0.2, and the tool changing time is 1.5 min, then the tool life (in min) for maximum production rate is ________.

(GATE-2014-SET-1, 2-Marks)

2.13 Which pair of following statements is correct for orthogonal cutting using a single-point cutting tool?
P. Reduction in friction angle increases cutting force
Q. Reduction in friction angle decreases cutting force
R. Reduction in friction angle increases chip thickness
S. Reduction in friction angle decreases chip thickness
(a) P and R (b) P and S
(c) Q and R (d) Q and S

(GATE-2014-SET-3, 2-Marks)

2.14 The normal force acting at the chip-tool interface in N is
(a) 1000 (b) 1500
(c) 2000 (d) 2500

(GATE-2013-ME-2-Marks)

2.15 A single-point cutting tool with 12° rake angle is used to machine a steel work-piece. The depth of cut, i.e. uncut thickness is 0.81 mm. The chip thickness under orthogonal machining condition is 1.8 mm. The shear angle is approximately
(a) 22° (b) 26°
(c) 56° (d) 76°

(GATE-2011-ME-2-Marks)

2.16 For tool A, Taylor's tool life exponent (n) is 0.45 and constant (K) is 90. Similarly for tool B, $n = 0.3$ and $K = 60$. The cutting speed (in m/min) above which tool A will have a higher tool life than tool B is
(a) 26.7 (b) 42.5
(c) 80.7 (d) 142.9

(GATE-2010-ME-2-Marks)

2.17 Friction at the tool-chip interface can be reduced by
(a) Decreasing the rake angle
(b) Increasing the depth of cut
(c) Decreasing the cutting speed
(d) Increasing the cutting speed

(GATE-2009-ME-1-Mark)

2.18 The values of shear angle and shear strain, respectively, are
(a) 30.3° and 1.98 (b) 30.3° and 4.23
(c) 40.2° and 2.97 (d) 40.2° and 1.65

(GATE-2006-ME-2-Marks)

2.19 The coefficient of friction at the tool-chip interface is
(a) 0.23 (b) 0.46
(c) 0.85 (d) 0.95

(GATE-2006-ME-2-Marks)

2.20 The percentage of total energy dissipated due to friction at the tool-chip interface is
(a) 30% (b) 42%
(c) 58% (d) 70%

(GATE-2006-ME-2-Marks)

2.21 In an orthogonal cutting test on mild steel, the following data were obtained

Cutting speed	:	40 m/min
Depth of cut	:	0.3 mm
Tool rake angle	:	+ 5°

Chip thickness	:	1.5 mm
Cutting force	:	900 N
Thrust force	:	450 N

Using Merchant's analysis, the Friction angle during the machining will be

(a) 26.6° (b) 31.5°
(c) 45° (d) 63.4°

(GATE-2004-ME-2-Marks)

2.22 In a machining operation, doubling the cutting speed reduces the tool life to 1/8th of the original value. The exponent *n* in Taylor's tool life equation $VT^n = C$, is

(a) $\frac{1}{8}$ (b) $\frac{1}{4}$
(c) $\frac{1}{3}$ (d) $\frac{1}{2}$

(GATE-2004-ME-2-Marks)

2.23 A built-up-edge is formed while machining
(a) Ductile materials at high speed
(b) Ductile materials at low speed
(c) Brittle materials at high speed
(d) Brittle materials at low speed

(GATE-2002-ME-2-Marks)

2.24 During orthogonal cutting of mild steel with a 10° rake angle tool the chip thickness ratio was obtained as 0.4, the shear angle (in degrees) evaluated from this data is

(a) 6.53 (b) 20.22
(c) 22.94 (d) 50.00

(GATE-2001-ME-2-Marks)

2.25 Tool life testing on a lathe under dry cutting conditions gave '*n*' and '*C*' of Taylor tool life equation as 012 and 130 m/min, respectively. When a coolant was used, *C* increased by 10%. Find the percent increase in tool life with the use of coolant at a cutting speed of 90 m/min. **(GATE-2001-ME-5-Marks)**

2.26 In an orthogonal cutting experiment with a tool of rake angle $\gamma = 7°$, the chip thickness was found to be 2.5 mm when the uncut chip thickness was set to 1 mm.
(a) Find the shear angle ϕ
(b) Find the friction angle β assuming that Merchant's formula holds good.

(GATE-1999-ME-5-Marks)

2.27 In an orthogonal machining operation, the chip thickness and the uncut thickness are equal to 0.45 mm. If the tool rake angle is 0°, the shear plane angle is

(a) 45° (b) 30°
(c) 18° (d) 60°

(GATE-1998-ME-2-Marks)

2.28 In a typical metal cutting operation, using a cutting tool of positive rake $\gamma = 10°$, it was observed that the shear angle was 20°. The friction angle is

(a) 45° (b) 30°
(c) 60° (d) 40°

(GATE-1997-ME-2-Marks)

2.29 A cutting tool has a radius of 1.8 mm. The feed rate for a theoretical surface roughness of $R_a = 5$ μm is

(a) 0.36 mm/rev (b) 0.187 mm/rev
(c) 0.036 mm/rev (d) 0.0187 mm/rev

(GATE-1997-ME-2-Marks)

2.30 To get good surface finish on a turned job, one should use a sharp tool with a ______________ feed and ______________ speed of rotation of the job.

(GATE-1994-ME-2-Marks)

2.31 Tool life of 10 hours is obtained when cutting with single point tool at 63 m/min.

If Taylor's constant *C* = 257.35, tool life on doubling the velocity will be

(a) 5 hours (b) 25.7 min
(c) 38.3 min (d) unchanged

(GATE-1996-ME-2-Marks)

Chapter 4

4.1 A shaft of length 90 mm has a tapered portion of length 55 mm. The diameter of the taper is 80 mm at one end and 65 mm at the other. If the taper is made by tailstock set over method, the taper angle and the set over respectively are
(a) 15°32′ and 12.16 mm
(b) 15°32′ and 15.66 mm

(c) 11°22′ and 10.26 mm
(d) 10°32′ and 14.46 mm

(GATE-2015-ME-SET-3, 2-Marks)

4.2 Match the Machine Tools (Group A) with the probable Operations (Group B):

Group A	Group B
(P) Centre lathe	(1) Slotting
(Q) Milling	(2) Counter-boring
(R) Grinding	(3) Knurling
(S) Drilling	(4) Dressing

(a) P-1, Q-2, R-4, S-3
(b) P-2, Q-1, R-4, S-3
(c) P-3, Q-1, R-4, S-2
(d) P-3, Q-4, R-2, S-1

(GATE-2014-ME-SET-2, 1-Mark)

4.3 A steel bar 200 mm in diameter is turned at a feed of 0.25 mm/rev with a depth of cut of 4 mm. The rotational speed of the workpiece is 160 rpm. The material removal rate in mm^3/s is

(a) 160 (b) 167.6
(c) 1600 (d) 1675.5

(GATE-2013-ME-1-Mark)

4.4 Cutting power consumption in turning can be significantly reduced by

(a) Increasing rake angle of the tool
(b) Increasing the cutting angles of the tool
(c) Widening the nose radius of the tool
(d) Increasing the clearance angle

(GATE-1995-ME-1-Mark)

Chapter 6

6.1 A cast iron block of 200 mm length is being shaped in a shaping machine with a depth of cut of 4 mm, feed of 0.25 mm/stroke and the tool principal cutting edge angle of 30°. Number of cutting strokes per minute is 60. Using specific energy for cutting as 1.49 J/mm^3, the average power consumption (in watt) is ________.

(GATE-2014-ME-SET-4, 2-Marks)

6.2 A 600 mm × 300 mm flat surface of a plate is to be finish machined on a shaper. The plate has been fixed with the 600 mm side along the tool travel direction. If the tool over-travel at each end of the plate is 20 mm, average cutting speed is 8 m/min, feed rate is 0.3 mm/stroke and the ratio of return time to cutting time of the tool is 1:2, the time required for machining will be

(a) 8 minutes (b) 12 minutes
(c) 16 minutes (d) 20 minutes

(GATE-2005-ME-2-Marks)

Chapter 7

7.1 A milling cutter having 8 teeth is rotating at 150 rpm. If the feed per tooth is 0.1, the table speed in mm per minute is

(a) 120 (b) 187
(c) 125 (d) 70

(GATE-1993-ME-2-Marks)

7.2 In horizontal milling process ________ (up/down) milling provides better surface finish and ________ (up/down) milling provides longer tool life.

(GATE-1992-ME-2-Marks)

Chapter 8

8.1 In a single pass drilling operation, a through hole of 15 mm diameter is to be drilled in a steel plate of 50 mm thickness. Drill spindle speed is 500 rpm, feed is 0.2 mm/rev and drill point angle is 118º. Assuming 2 mm clearance at approach and exit, the total drill time in seconds is

(a) 35.1 (b) 32.4
(c) 31.2 (d) 30.1

(GATE-2012-ME-2-Marks)

8.2 Trepanning is performed for

(a) Finishing a drilled hole
(b) Producing a large hole without drilling

(c) Truing a hole for alignment
(d) Enlarging a drilled hole

(GATE-2002-ME-1-Mark)

8.3 The time taken to drill a hole though a 25 mm thick plate with the drill rotating at 300 rpm and moving at a feed rate of 0.25 mm/revolution is

(a) 10 sec (b) 20 sec
(c) 60 sec (d) 100 sec

(GATE-2002-ME-2-Marks)

8.4 The rake angle in a drill

(a) Increases from centre to periphery
(b) Decreases from centre to periphery
(c) Remains constant
(d) is irrelevant to the drilling operation

(GATE-1996-ME-1-Mark)

8.5 A hole of 20 mm diameter is to be drilled in a steel block of 40 mm thickness. The drilling is performed at rotational speed of 400 rpm and feed rate of 0.1 mm/rev. The required approach and over run of the drill together is equal to the radius of drill. The drilling time (in minute) is

(a) 1.00 (b) 1.25
(c) 1.50 (d) 1.75

(GATE-2014-ME-SET-2, 2-Marks)

Chapter 9

9.1 Diamond wheels should not be used for grinding steel components. State True or False.

(GATE-1996-ME-2-Marks)

9.2 If each abrasive grain is viewed as a cutting tool, then which of the following represents the cutting parameters in common grinding operations?

(a) Large negative rake angle, low shear angle and high cutting speed
(b) Large positive rake angle, low shear angle and high cutting speed
(c) Large negative rake angle, high shear angle and low cutting speed
(d) Zero rake angle, high shear angle and high cutting speed

(GATE-2006-ME-2-Marks)

9.3 The hardness of a grinding wheel is determined by the

(a) Hardness of abrasive grains
(b) Ability of the bond to retain abrasives
(c) Hardness of the bond
(d) Ability of the grinding wheel to penetrate the work piece

(GATE-2002-ME-1-Mark)

9.4 Abrasive material used in grinding wheel selected for grinding ferrous alloys is:

(a) silicon carbide
(b) diamond
(c) aluminium oxide
(d) boron carbide

(GATE-2000-ME-1-Mark)

9.5 Ideal surface roughness, as measured by the maximum height of unevenness, is best achieved when the material is removed by

(a) an end mill
(b) a grinding wheel
(c) a tool with zero nose radius
(d) a ball mill

(GATE-1998-ME-1-Mark)

9.6 In machining using abrasive material, increasing abrasive grain size

(a) increases the material removal rate
(b) decreases the material removal rate
(c) first decreases and then increases the material removal rate
(d) first increases and then decreases the material removal rate

(GATE-1998-ME-1-Mark)

9.7 Among the conventional machining processes, maximum specific energy is consumed in

(a) Turning (b) Drilling
(c) Planning (d) Grinding

(GATE-1995-ME-1-Mark)

Chapter 10

10.1 Internal gears are manufactured by
(a) Hobbing
(b) Shaping with pinion cutter
(c) Shaping with rack cutter
(d) Milling
(GATE-2016-ME-SET-3, 1-Mark)

10.2 Internal gear cutting operation can be performed by
(a) Milling
(b) Shaping with rack cutter
(c) Shaping with pinion cutter
(d) Hobbing
(GATE-2008-ME-1-Mark)

Chapter 11

11.1 The principle of material removal in electrochemical machining is
(a) Fick's law
(b) Faraday's laws
(c) Kirchhoff's laws
(d) Ohm's law
(GATE-2014-SET-4, 1-Mark)

11.2 During the electrochemical machining (ECM) of iron (atomic weight = 56, valency = 2) at current of 1000 A with 90% current efficiency, the material removal rate was observed to be 0.26 gm/s. If Titanium (atomic weight = 48, valency = 3) is machined by the ECM process at the current of 2000 A with 90% current efficiency, the expected material removal rate in gm/s will be
(a) 0.11 (b) 0.23
(c) 0.30 (d) 0.52
(GATE-2013-ME-2-Marks)

11.3 The non-traditional machining process that essentially requires vacuum is
(a) electron beam machining
(b) electro chemical machining
(c) electro chemical discharge machining
(d) elecro discharge machining
(GATE-2016-ME-SET-1, 1-Mark)

11.4 In an ultrasonic machining (USM) process, the material removal rate (MRR) is plotted as a function of the feed force of the USM tool. With increasing feed force, the MRR exhibits the following behaviour:
(a) Increases linearly
(b) Decreases linearly
(c) Does not change
(d) First increases and then decreases
(GATE-2016-ME-SET-2, 1-Mark)

11.5 In a wire-cut EDM process the necessary conditions that have to be met for making a successful cut are that
(a) Wire and sample are electrically non-conducting
(b) Wire and sample are electrically conducting
(c) Wire is electrically conducting and sample is electrically non-conducting
(d) Sample is electrically conducting and wire is electrically non-conducting
(GATE-2016-ME-SET-3, 1-Mark)

11.6 The primary mechanism of material removal in electrochemical machining (ECM) is
(a) Chemical corrosion
(b) Etching
(c) Ionic dissolution
(d) Spark erosion
(GATE-2015-ME-SET-2, 1-Mark)

11.7 The following four unconventional machining processes are available in a shop floor. The most appropriate one to drill a hole of square cross section of 6 mm × 6 mm and 25 mm deep
(a) Is abrasive Jet Machining
(b) Is Plasma Arc Machining
(c) Is Laser Beam Machining
(d) Is Electro Discharge Machining
(GATE-2014-SET-2, 1-Mark)

11.8 The process utilizing mainly thermal energy for removing material is
(a) Ultrasonic Machining
(b) Electrochemical Machining
(c) Abrasive Jet Machining
(d) Laser Beam Machining
(GATE-2014-SET-3, 1-Mark)

11.9 In abrasive jet machining, as the distance between the nozzle tip and the work surface increases, the material removal rate
(a) Increases continuously
(b) Decreases continuously
(c) Decreases, becomes stable and then increases
(d) Increases, becomes stable and then decreases
(GATE-2012-ME-1-Mark)

11.10 A researcher conducts electrochemical machining (ECM) on a binary alloy (density 6000 kg/m^3) of iron (atomic weight 56, valency 2) and metal P (atomic weight 24, valency 4). Faraday's constant = 96500 coulomb/mole. Volumetric material removal rate of the alloy is 50 mm^3/s at a current of 2000 A. The percentage of the metal P in the alloy is closest to
(a) 40 (b) 25
(c) 15 (d) 79
(GATE-2008-ME-2-Marks)

11.11 In electrodes charge machining (EDM), if the thermal conductivity of tool is high and the specific heat of work piece is low, then the tool wear rate and material removal rate are expected to be respectively
(a) High and high (b) High and low
(c) Low and low (d) Low and high
(GATE-2007-ME-2-Marks)

11.12 Arrange the processes in the increasing order of their maximum material removal rate.
Electrochemical Machining (ECM)
Ultrasonic Machining (USM)
Electron Beam Machining (EBM)
Laser Beam Machining (LBM) and
Electric Discharge Machining (EDM)
(a) USM, LBM, EBM, EDM, ECM
(b) EBM, LBM, USM, ECM, EDM
(c) LBM, EBM, USM, ECM, EDM
(d) LBM, EBM, USM, EDM, ECM
(GATE-2006-ME-2-Marks)

11.13 The mechanism of material removal in EDM process is
(a) Melting and Evaporation
(b) Melting and Corrosion
(c) Erosion and Cavitation
(d) Cavitation and Evaporation
(GATE-2004-ME-1-Mark)

11.14 As tool and work are not in contact in EDM process
(a) No relative motion occurs between them
(b) No water of tool occurs
(c) No power is consumed during metal cutting
(d) No force between tool and work occurs
(GATE-2003-ME-1-Mark)

11.15 In ECM, the material removal is due to
(a) Corrosion (b) Erosion
(c) Fusion (d) Ion displacement
(GATE-2001-ME-1-Mark)

11.16 In Electro-Discharge Machining (EDM), the tool is made of:
(a) Copper
(b) High Speed Steel
(c) Cast Iron
(d) Plain Carbon Steel
(GATE-1999-ME-1-Mark)

11.17 Selection electrolyte for ECM is as follows:
(a) non-passivating electrolyte for stock removal and passivating electrolyte for finish control
(b) passivating electrolyte for stock removal and non-passivating electrolyte for finish control
(c) selection of electrolyte is dependent on current density
(d) electrolyte selection is based on tool-work electrodes
(GATE-1997-ME-1-Mark)

11.18 Inter electrode gap in ECG is controlled by
(a) controlling the pressure of electrolyte flow
(b) controlling the applied static load
(c) controlling the size of diamond particle in the wheel
(d) controlling the texture of the work piece
(GATE-1997-ME-1-Mark)

11.19 Ultrasonic machining is about the best process for making holes in glass which are comparable in size with the thickness of the sheet. **(GATE-1994-ME-2-Marks)**

11.20 In ultrasonic machining process, the material removal rate will be higher for materials with
(a) Higher toughness
(b) Higher ductility
(c) Lower toughness
(d) Higher fracture strain
(GATE-1993-ME-1-Mark)

11.21 In Ultrasonic Machining (USM) the material removal rate would _____ with increasing mean grain diameter of the abrasive material
(a) Increase
(b) Decrease
(c) Increase and then decrease
(d) Decrease and then increase
(GATE-1992-ME-1-Mark)

Chapter 15

15.1 3-2-1 method of location in a jig or fixture would collectively restrict the work piece in *n* degrees of freedom, where the value of *n* is
(a) 6 (b) 8
(c) 9 (d) 12
(GATE-2001-ME-2-Marks)

15.2 When a cylinder is located in a Vee-block, the number of degrees of freedom which area arrested is
(a) 2 (b) 4
(c) 7 (d) 8
(GATE-2003-ME-1-Mark)

Chapter 16

16.1 A hole is dimension $\phi 9^{+0.015}_{+0}$ mm. The corresponding shaft is of dimension $\phi 9^{+0.010}_{+0.001}$ mm. The resulting assembly has
(a) Loose running fit
(b) Close running fit
(c) Transition fit
(d) Interference fit
(GATE-2011-ME-1-Mark)

16.2 What are the upper and lower limits of the shaft represented by $60\, f_8$?
Use the following data:

Diameter 60 lies in the diameter step of 50–80 mm

Fundamental tolerance unit, i, in μm = $0.45D^{1/3} + 0.001$ D, where D is the representative size in mm;

Tolerance value for IT8 = 25i. Fundamental deviation for 'f' shaft = $-5.5D^{0.41}$

(a) Lower limit = 59.924 mm, Upper Limit = 59.970 mm
(b) Lower limit = 59.954 mm, Upper Limit = 60.000 mm
(c) Lower limit = 59.970 mm, Upper Limit = 60.016 mm
(d) Lower limit = 60.000 mm, Upper Limit = 60.046 mm
(GATE-2009-ME-2-Marks)

16.3 A hole is specified as $40^{0.050}_{0.000}$ mm. The mating shaft has a clearance fit with minimum clearance of 0.01 mm. The tolerance on the shaft is 0.04 mm. The maximum clearance in mm between the hole and the shaft is
(a) 0.04 (b) 0.05
(c) 0.10 (d) 0.11
(GATE-2007-ME-2-Marks)

16.4 In order to have interference fit, it is essential that the lower limit of the shaft should be
(a) Greater than the upper limit of the hole
(b) Lesser than the upper limit of the hole
(c) Greater than the lower limit of the hole
(d) Lesser than the lower limit of the hole
(GATE-2005-ME-1-Mark)

16.5 In an interchangeable assembly, shafts of size $25.000^{+0.040}_{-0.0100}$ mm mate with holes of size $25.000^{+0.020}_{-0.000}$ mm. The maximum possible clearance in the assembly will be

(a) 10 microns (b) 20 microns
(c) 30 microns (d) 60 microns

(GATE-2004-ME-1-Mark)

16.6 Allowance in limits and fits refers to
(a) Maximum clearance between shaft and hole
(b) Minimum clearance between shaft and hole
(c) Difference between maximum and minimum size of hole
(d) Difference between maximum and minimum size of shaft

(GATE-2001-ME-1-Mark)

16.7 Holes of diameter $25.0^{+0.040}_{+0.020}$ mm are assembled interchangeably with the pins of diameter $25.0^{+0.005}_{-0.008}$ mm. The minimum clearance in the assembly will be
(a) 0.048 mm (b) 0.015 mm
(c) 0.005 mm (d) 0.008 mm

(GATE-2015-ME-SET-1, 1-Mark)

16.8 Which one of the following statements is TRUE?
(a) The 'GO' guage controls the upper limit of a hole
(b) The 'NO' guage controls the lower limit of a shaft
(c) The 'GO' guage controls the lower limit of a hole
(d) The 'NO GO' guage controls the lower limit of a hole

(GATE-2015-ME-SET-2, 1-Mark)

16.9 For the given assembly: 25 $H7/g8$, match Group A with Group B.

Group A	Group B
(P) H	(I) Shaft Type
(Q) IT8	(II) Hole Type
(R) IT7	(III) Hole Tolerance Grade
(S) g	(IV) Shaft Tolerance Grade

(a) P-I, Q-III, R-IV, S-II
(b) P-I, Q-IV, R-III, S-II
(c) P-II, Q-III, R-IV, S-I
(d) P-II, Q-IV, R-III, S-I

(GATE-2014-ME-SET-1, 2-Marks)

16.10 A GO-No GO plug gauge is to be designed for measuring a hole of nominal diameter 25 mm with a hole tolerance of ±0.015 mm. Considering 10% of work tolerance to be the gauge tolerance and no wear condition, the dimension (in mm) of the GO plug gauge as per the unilateral tolerance system is
(a) $24.985^{+0.003}_{-0.003}$ (b) $25.015^{+0.000}_{-0.006}$
(c) $24.985^{+0.003}_{-0.003}$ (d) $24.985^{+0.003}_{-0.000}$

(GATE-2014-ME-SET-4, 2-Marks)

16.11 A metric thread of pitch 2 mm and thread angle 60º is inspected for its pitch diameter using 3-wire method. The diameter of the best size wire in mm is
(a) 0.866 (b) 1.000
(c) 1.154 (d) 2.000

(GATE-2013-ME-1-Mark)

16.12 Cylindrical pins of $25^{+0.020}_{+0.010}$ mm diameter are electroplated in a shop. Thickness of the plating is $30^{+2.0}$ micron. Neglecting guage tolerances, the size of the GO guage in mm to inspect the plated components is
(a) 25.042 (b) 25.052
(c) 25.074 (d) 25.084

(GATE-2013-ME-2-Marks)

16.13 In an interchangeable assembly, shafts of size $25.000^{+0.040}_{-0.010}$ mm mate with holes of size $25.000^{+0.030}_{+0.020}$ mm. The maximum interference (in microns) in the assembly is
(a) 40 (b) 30
(c) 20 (d) 10

(GATE-2012-ME-1-Mark)

16.14 A ring guage is used to measure
(a) Outside diameter but not roundness
(b) Roundness but not outside diameter
(c) Both outside diameter and roundness
(d) Only external threads

(GATE-2006-ME-1-Mark)

16.15 GO and No-GO plug guage are to be designed for a hole $20.000^{+0.050}_{+0.010}$ mm. Guage tolerances can be taken as 10% of the hole tolerance. Following ISO system of guage design, sizes of GO and NO-GO guage will be respectively

(a) 20.010 mm and 20.050 mm
(b) 20.014 mm and 20.046 mm
(c) 20.006 mm and 20.054 mm
(d) 20.014 mm and 20.054 mm

(GATE-2004-ME-2-Marks)

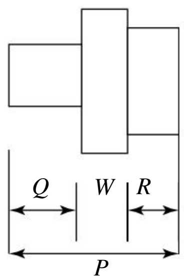
16.16 The dimensional limits on a shaft of 25*h*7 are

(a) 25.000, 25.021 mm
(b) 25.000, 24.979 mm
(c) 25.000, 25.007 mm
(d) 25.000, 24.993 mm

(GATE-2003-ME-1-Mark)

16.17 A part shown in the figure is machined to the sizes given below

P = 35.00 ± 0.08 mm



Q = 12.00 ± 0.02 mm

R = $13.00^{+0.04}_{-0.02}$ mm

With 100% confidence, the resultant dimension W will have the specification

(a) 9.99 ± 0.03 mm
(b) 9.99 ± 0.13 mm
(c) 10.00 ± 0.03 mm
(d) 10.00 ± 0.13 mm

(GATE-2003-ME-2-Marks)

16.18 A shaft (diameter $20^{+0.05}_{-0.15}$) and a hole (diameter $20^{+0.20}_{+0.10}$ mm) when assembled would yield

(a) Transition fit (b) Interference fit
(c) Clearance fit (d) None of these

(GATE-1993-ME-2-Marks)

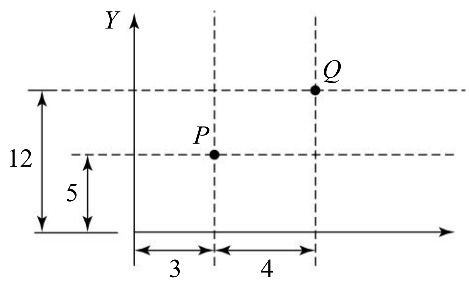
Chapter 17

17.1 The function of interpolator in a CNC machine controller is to

(a) Control spindle speed
(b) Coordinate feed rates of axes
(c) Control tool rapid approach speed
(d) Perform Miscellaneous (*M*) functions (tool change, coolant control etc.)

(GATE-2015-ME-SET-1, 1-Mark)

17.2 A drill is positioned at point *P* and its has to proceed to point *Q*. The coordinates of point *Q* in the incremental system of defining position of a point in CNC part program will be



(a) (3, 12) (b) (5, 7)
(c) (7, 12) (d) (4, 7)

(GATE-2015-ME-SET-3, 1-Mark)

17.3 In a CNC milling operation, the tool has to machine the circular arc from point (20, 20) to (10, 10) at sequence number 5 of the CNC part program. If the center of the arc is at (20, 10) and the machine has incremental mode of defining position coordinates, the correct tool path command is

(a) N 05 G90 G01 X–10 Y–10 R10
(b) N 05 G91 G03 X–10 Y–10 R10
(c) N 05 G90 G03 X–20 Y–20 R10
(d) N 05 G91 G02 X–20 Y–20 R10

(GATE-2015-ME-SET-3, 2-Marks)

17.4 For machining a rectangular island represented by coordinates *P*(0, 0), *Q*(100, 0), *R*(100, 50) and (0, 50) on a casting using CNC milling machine, an end mill with a diameter of 16 mm is used. The trajectory of the cutter centre to machine the island PQRS is

(a) (–8, –8), (108, –8), (108, 58), (–8, 58), (–8, –8)
(b) (8, 8), (94, 8), (94, 44), (8, 44), (8, 8)
(c) (–8, 8), (94, 0), (94, 44), (8, 44), (–8, 8)
(d) (0, 0), (100, 0), (100, 50), (50, 0), (0, 0)

(GATE-2014-ME-SET-1, 1-Mark)

17.5 A CNC vertical milling machine has to cut a straight slot of 10 mm width and 2 mm depth by a cutter of 10mm diameter between points (0, 0) and (100, 100) on the XY plane (dimensions in mm). The feed rate used for milling is 50 mm/min. milling time for the slot (in seconds) is

(a) 120 (b) 170
(c) 180 (d) 240

(GATE-2012-ME-1-Mark)

17.6 In a CNC program block, N002 G02 G91 X40 Z40…, G02 and G91 refer to

(a) Circular interpolation in counterclockwise direction and incremental dimension
(b) Circular interpolation in counterclockwise direction and absolute dimension
(c) Circular interpolation in clockwise direction and incremental dimension
(d) Circular interpolation in clockwise direction and absolute dimension

(GATE-2010-ME-1-Mark)

17.7 Match the following:

NC Code	Definition
P. M05	1. Absolute coordinate system
Q. G01	2. Dwell
R. G04	3. Spindle stop
S. G90	4. Linear interpolation

(a) P-2, Q-3, R-4, S-1.
(b) P-3, Q-4, R-1, S-2.
(c) P-3, Q-4, R-2, S-1.
(d) P-4, Q-3, R-2, S-1.

(GATE-2009-ME-2-Marks)

17.8 NC contouring is an example of

(a) Continuous path positioning
(b) Point-to-point positioning
(c) Absolute positioning
(d) Incremental positioning

(GATE-2006-ME-1-Mark)

17.9 Which among the NC operations given below are continuous path operations?

Arc Welding (AW)	Milling (M)
Drilling (D)	Punching in Sheet Metal (P)
Laser Cutting of Sheet Metal (LC)	Spot Welding (SW)

(a) AW, LC and M
(b) AW, D, LC and M
(c) D, LC, P and SW
(d) D, LC and SW

(GATE-2005-ME-1-Mark)

17.10 The tool of an NC machine has to move along a circular arc from (5, 5) to (10, 10) while performing an operation. The center of the arc is at (10, 5). Which one of the following NC tool path commands performs the above mentioned operation?

(a) N010 G02 X10 Y10 X5 Y5 R5
(b) N010 G03 X10 Y10 X5 Y5 R5
(c) N010 G01 X5 Y5 X10 Y10 R5
(d) N010 G02 X5 Y5 X10 Y10 R5

(GATE-2005-ME-2-Marks)

17.11 During the execution of a CNC part program block N020 G02 X45.0 Y25.0 R5.0 the type of tool motion will be

(a) Circular Interpolation – clockwise
(b) Circular Interpolation – counterclockwise
(c) Linear Interpolation
(d) Rapid feed

(GATE-2004-ME-1-Mark)

17.12 CNC machines are more accurate than conventional machines because they have a high resolution encoder and digital read-outs for positioning.

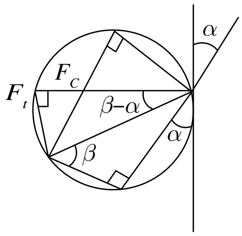
(GATE-1994-ME-2-Marks)

SOLUTIONS

Chapter 2

2.1 (0.5)

$$Tan(\beta-\alpha)=\frac{F_t}{F_c}$$



$$Tan(\beta-0)=\frac{250}{500}$$

$$Tan\beta=1/2=0.5$$

2.2 (b)

$VT^{0.14}f^{0.7}d^{0.4}$ = constant

$$V_1=45\,\text{m/min};\quad f_1=0.35\,\text{mm}\;\; d_1=2\,\text{mm}$$

$$T_1=30\text{ min};\;\; V_2=1.25V_1;$$

$$f_2=1.25f_1,\;\; d_1=1.25d_1;$$

$$T_2=?$$

$$\Rightarrow 45\times T_1^{0.14}f_1^{0.7}d_1^{0.4}=1.25\times 45\times T_2^{0.14}$$

$$\times 1.25^{0.7}f_1^{0.7}\times 1.25^{0.4}d_1^{0.4}$$

$$\Rightarrow T_1^{0.14}=1.25\times 1.25^{0.7}\times 1.25^{0.4}\times T_2^{0.14}$$

$$\Rightarrow T_2=\frac{(30)^{0.14}}{1.25^{2.1}/0.14}=1.055=1.06$$

2.3 (7.539)

L = 900 mm

d = 200 mm

f = 0.25 mm

v = 300 m/min

t = ?

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

$$300\,\text{m/min}=\pi\times 0.2\times \text{N}$$

$$N=\frac{300}{\pi\times 0.2}=478\,\text{RPM}$$

$$t=\frac{L}{fN}=\frac{900}{0.25\times 478}\approx 7.539\text{ min}$$

2.4 (b)

Given $\alpha = 22°$

Thickness of chip = 0.8 mm = (t_2)

Feed = 0.4 mm/rev = f

Speed = 48 m/min

In orthogonal cutting = feed = thickness of uncut chip. (t_1)

$$K=\frac{t_2}{t_1}=\frac{0.8}{0.4}=2$$

$$\tan\theta=\frac{\cos\alpha_\circ}{K-\sin\alpha_\circ}=\frac{\cos 22}{2-\sin 22}=0.57$$

$$\theta=\tan^{-1}(0.57)\approx 29.7$$

2.5 (16)

n = 0.25

$$V_2=\frac{V_1}{2}$$

$$V_1T_1^n=V_2T_2^n$$

$$V_1T_1^n=\frac{V_1}{2}T_2^n$$

$$\left(T_2/T_1\right)^{0.25}=2$$

$$T_2=T_1\times 2^4$$

$$\Rightarrow\quad T_2=16\times T_1$$

Tool life changes by 16 times

2.6 (0.25)

$$VT^n=C$$

$$V_1T_1^n=2V_1\times\left(\frac{T_1}{16}\right)$$

on solving we get 'n' = 0.25

2.7 (18.88°)

$$\alpha=5°, N=400, V_f=0.4\text{ m/min}$$

$$d=5\text{ mm},\, t_c=3\text{ mm}$$

$$V_f=fN\Rightarrow f=\frac{0.4}{400}=1\text{ mm/rev}$$

$$r=\frac{f}{t_c}=\frac{1}{3}$$

$$\tan\phi=\frac{r\cos\alpha}{1-r\sin\alpha}=\frac{\frac{1}{3}\cos 5°}{1-\frac{1}{3}\sin 5°}$$

$$\tan\phi\approx 0.34\Rightarrow\phi=18.88°$$

2.8 (2.1)

$$\mu = \frac{F}{N} = \frac{F_c \sin\alpha + F_T \cos\alpha}{F_c \cos\alpha - F_T \sin\alpha}$$

Given: $\alpha = 0^\circ$

$$\mu = \frac{F_T}{F_c}$$

(OR)

$$0.7 = \frac{490}{F_c}$$

$\therefore F_c = 700\,\text{N}$

Power consumption, $P = F_c \times V_c$

$$= 700 \times \frac{180}{60} \times \frac{1}{1000}(\text{kW})$$

$P = 2.1\,\text{kW}$

2.9 (c)
The surface obtained is plane.

2.10 (a)

$$r = \frac{0.14}{0.28} = 0.5$$

$$\tan\phi = \frac{r\cos\alpha}{1 - r\sin\alpha}$$

$\therefore \phi = 28.3345°$

or 28°20'

Shear strain,
$\gamma = \cot\phi + \tan(\phi - \alpha) = 2.1859 \approx 2.19$

2.11 (2.8 to 3.0)

Chip thickness ratio $r = \frac{0.2}{0.5} = \frac{t_1}{t_2}$

$$\tan\phi = \frac{r\cos\alpha}{1 - r\sin\alpha} = 0.4$$

$\phi = 21.8°$

Shear strain $= \cot\phi + \tan(\phi - \alpha)$

$= \cot 21.8 + \tan(21.8 - \alpha) = 2.9.$

2.12 (5.9 to 6.1)

$$T_{\text{opt}} = \left[\frac{1-n}{n} \times T_C\right] = \frac{1-0.2}{0.2} \times 1.5 = 6\text{ min}.$$

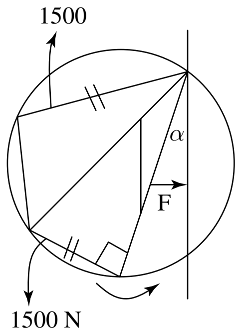
2.13 (d)

2.14 (b)
Normal force

$N = F_H \cos\alpha - F_V \sin\alpha$

$= 1500 \times \cos 0 - F_v \sin 0$

$= 1500$



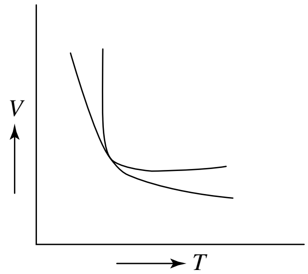
2.15 (b)
Relation between shear angle (ϕ), chip thickness ratio (r) and rake angle (α) is given by

$$\text{Tan}\,\phi = \frac{r\cos\alpha}{1 - r\sin\alpha}$$

Where $r = \frac{0.81}{1.8} = 0.45$

$$\text{Tan}\,\phi = \frac{0.45\cos 12}{1 - 0.45\sin 12} \Rightarrow \phi = 26^\circ$$

2.16 (a)
Tool life Vs Cutting speed diagram is shown for the given conditions for both A and B.



Tool A
Constant K = 90
Exponential constant = 0.45
Tool B
Constant K = 60

Exponential constant = 0.3

At the point of intersection $V_a = V_b$ and $T_a = T_b$

Therefore equating the above conditions we get $T = 14.89$ min

Substituting the t in any tool life equation will gives the $V_a = 26.7$ m/min.

Tool life A will be always higher than tool life B when the cutting velocity is higher than 26.7 m/min. Hence the given condition is satisfied.

2.17 (d)

Friction decreases at the tool-chip interface. With increase in cutting speed, the thickness of chip reduces and with the decrease in rake angle, friction decreases.

2.18 (d)

r = cutting ratio

$$= \frac{t_1}{t_2} = \frac{0.5}{0.7} = 0.714, \tan\phi = \frac{r\cos\alpha}{1 - r\sin\alpha},$$

where $\phi =$ shear angle

α = rake angle $\therefore \phi = 40.2^\circ$

Shear strain = $\cot\phi + \tan(\phi - \alpha) = 1.65$

2.19 (b)

From Merchant's theory,

$$2\phi + \lambda - \alpha = \frac{\pi}{2}, \lambda = \text{friction angle}, \lambda = 24.6,$$

$\mu = \tan\lambda = 0.457$

2.20 (a)

$Q_1 =$ Total heat generated

$= F_c \times V = 24000$ J/min.

$Q_2 =$ Heat dissipated due to friction

$= F \times r \times v$

F = Frictional force,

$$F = F_c \sin\alpha + F_T \cos\alpha = 503.76\,\text{N}$$

$$Q_2 = 7193.69 \text{ J/min.} \frac{Q_2}{Q} \frac{Q_2}{Q_1} = 0.299 \simeq 30\%$$

2.21 (b)

$$\mu = \frac{F}{N} = \frac{F_c \sin\alpha + F_T \cos\alpha}{F_c \cos\alpha - F_T \sin\alpha} = 0.614$$

$$\lambda = \tan^{-1}\mu = 31.5^\circ$$

2.22 (c)

$$V_1 T_1^n = V_2 T_2^n,\ V_1 T_1^n = 2V_1\left(\frac{T_1}{8}\right)^n,\ n = \frac{1}{3}$$

2.23 (b)

2.24 (c)

$$\tan\phi = \frac{r\cos\alpha}{1 - r\sin\alpha} \Rightarrow \phi = 22.94^\circ$$

2.25 (119.68)

Tool life equati

$VT^n = C, n = 0.12,\ c = 130,$

when coolant is used,

$C_1 = 143$

before coolant use $\Rightarrow 90 \times T_1^n = C \Rightarrow T_1 = 21.54\,\text{min}$

after coolant application $\Rightarrow 90 \times T_2^n = C_1 \Rightarrow T_2 = 47.32\,\text{min}$

$$\%\text{ increase} = \frac{T_2 - T_1}{T_1} \times 100 = 119.68\%$$

2.26 $\alpha = 7^\circ,\ r = \text{cutting ratio} = \frac{1}{2.5} = 0.4$

(a) $\tan\phi = \frac{r\cos\alpha}{1 - r\sin\alpha} \Rightarrow \phi = 22.65^\circ,$

(b) $2\phi + \lambda - \alpha = 90^\circ \Rightarrow \lambda = 51.3^\circ$

2.27 (a)

Cutting ratio, $r = 1$,

$$\tan\phi = \frac{r\cos\alpha}{1 - r\sin\alpha} = 1 \Rightarrow \phi = 45^\circ$$

2.28 (c)

$2\phi + \lambda - \alpha = 90^\circ \Rightarrow \lambda = \text{friction angle} = 60^\circ$

2.29 (c)

$$R_a = \frac{f^2}{8r} \Rightarrow f = 0.03 \text{ mm/rev}$$

2.30 Minimum and Maximum

For getting good surface finish, below cutting parameters to be used:

- Low feed
- Low depth of cut
- High speed

2.31 (b)

$T_1 = 10 \text{ hours} = 600 \text{ mins},$

$V_2 = 2V_1 = 2 \times 63 = 126 \dfrac{\text{m}}{\text{min}}$

$V_1 = 63 \dfrac{\text{m}}{\text{min}}, T_2 = ?$

$V_1T_1^n = V_2T_2^n = C \ [\text{given}, C = 257.35]$

$(63) \times (600)^n = 257.35 \Rightarrow n = 0.22$

$(126) \times [T_2]^n = 257.35 \Rightarrow [T_2] = [2.042]^{\frac{1}{0.22}}$

$= 25.7 \text{ mins}$

Chapter 4

4.1 (a)

Rate of taper, $T = \dfrac{80 - 65}{55} = 0.27$

Set over = $\dfrac{T \times L}{2} = \dfrac{0.27 \times 90}{2} = 12.15$

Taper angle $= \text{Tan}^{-1}(0.27) = 15.10$

4.2 (c)

4.3 (d)

$f \times d \times v$

$= (0.25)(4) \times \dfrac{\pi \times 200 \times 160}{60} = 1675.5 \text{ mm}^3/\text{s}$

4.4 (a)

Cutting power can be reduced by reducing the cutting force.

The cutting force value is primarily affected by:

- Cutting conditions (cutting speed V, feed f, depth of cut d)
- Cutting tool geometry (tool orthogonal rake angle)
- Properties of work material

Chapter 6

6.1 (295 to 305)

Specific cutting energy

$= \dfrac{F_C}{b \times t_1} = 1.49 \text{ J/mm}^3$

$= \dfrac{F_C}{4 \times 0.25}$

$F_C = 1.49 \text{ J/mm. stroke}$

$F_C = 1.49 \text{ J/mm} \times \dfrac{60}{\text{min}}$

$F_C = 89.4 \text{ J/mm.min}$

Power $= F_C.l = 89.4 \times 200 \text{ mm J/mm.min}$

$= 17880 \text{ J/min}$

Power $= 298$ J/S or (W).

6.2 (b)

Length travelled in forwarded stroke: Lf = 600 + 20 + 20 = 640 min

Cutting time = 8 min, Return time

$= \dfrac{1}{2} \times \text{cutting time} = \dfrac{1}{2} \times 8 = 4 \text{ min}$

Total time = 8 + 4 = 12 min

Chapter 7

7.1 (a)

Take feed,

$S_t = Z \times S_Z \times N$

$Z = \text{No. of teeth}$

$S_Z = \text{feed per teeth}$

$N = \text{rpm}$

$S_t = 8 \times 0.1 \times 150 = 120 \dfrac{\text{mm}}{\text{min}}$

7.2 Down Milling

In up milling, tool wear is more because the tool runs against the feed. Hence tool life is low.

In down milling tool wear is less compared to up milling, due to cutter rotation with the feed. Hence, more tool life.

In up milling, the cutting chips fall down in front of the cutting tool which again cut the chips causing less surface finish, where as in down milling, the cutting chip falls down behind the tool, gives better surface finish.

Chapter 8

8.1 (a)

$$T_c = \frac{(L_h + A + O + C)}{Nf}$$

$$L_h = 50 \text{ mm}; A = O = 2 \text{ mm}; C = \frac{D}{2}\cot\left(\frac{\beta}{2}\right)$$

$$= 7.5 \times \cot(59) = 4.5\,\text{mm}$$

$$N = 500 \text{ rpm}; \ f = 0.2\,\text{mm/rev}$$

$$T_c = 0.585\,\text{min or } 35.1 \text{ Seconds}$$

8.2 (b)

8.3 (b)

Time taken to drill, $T = \frac{t}{fN} = 20$ sec

8.4 (a)

8.5 (b)

$$T = \frac{L}{f_N}$$

$$L = t + Ap_1$$

$$Ap_1 = 0.5\,D\,(\text{holes diameter}) = 10 \text{ mm}$$

$$t = 40 \text{ mm}$$

$$T = \frac{50}{0.1 \times 400} = 1.25 \text{ min}.$$

Chapter 9

9.1 True
9.2 (a)
9.3 (b)
9.4 (c)
9.5 (c)
9.6 (a)
9.7 (d)

Chapter 10

10.1 (b)
10.2 (c)

Chapter 11

11.1 (b)

11.2 (c)

$$Q = \frac{AI}{F_2} = \frac{0.9 \times 48 \times 2000}{3 \times 96500 \times 3}$$

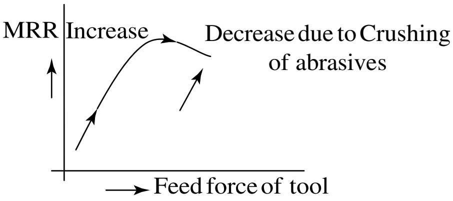
$$Q = 0.3\,\text{gm/s}$$

11.3 (a)

Electron beam machining requires vacuum, to avoid deflection of electrons.

11.4 (d)

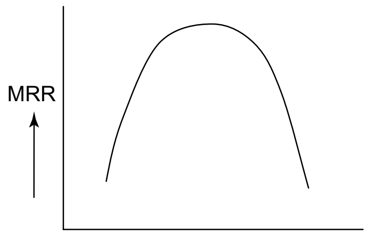
In USM,



11.5 (b)

In this process, a thin metallic wire is fed on to the conducting workpiece, which is submerged in a tank of dielectric fluid such as de-ionized water. Wire is fed in the programmed path and material is cut from the workpiece accordingly. Material removal takes place by a series of discrete discharges between the wire electrode and workpiece in the presence of a dielectric fluid. The dielectric fluid gets ionized in between the tool electrode gap thereby creating a path for each discharge. The area wherein discharge takes place gets heated to very high temperature such that the surface get melted and removed. The cut particles (debris) get flushed away by continuous flowing dielectric fluid. Generally, wire-cut EDM is used for cutting Aluminium, brass, etc. and wire material used for quicker cutting action is zinc coated brass wires.

11.6 (c)
11.7 (d)
11.8 (d)
11.9 (d)



As the distance between the nozzle tip and the Work surface increase, area of contact at which jet strikes increases so, material removal rate increases but further increment in distance causes reduction in velocity striking to the surface.

11.10 (c)
11.11 (d)
11.12 (c)
11.13 (a)
11.14 (d)
11.15 (d)
11.16 (a)
11.17 (a)
11.18 (c)
11.19 True
11.20 (c)

The basic principle of USM is brittle fracture. Hence, the work materials which are brittle i.e., having lower toughness are suitable for this process.

11.21 (a)

Chapter 15

15.1 (c)
15.2 (c)

Chapter 16

16.1 (c)
16.2 (a)

Diameter 60 lies in the diameter step of 50–80 mm.

$\therefore\ D = \sqrt{50 \times 80} = 63.25$ mm

Fundamental tolerance unit, i will be

$i = 0.45\,D^{1/3} + 0.001\,D = 1.83\,\text{microns}$

Tolerance value for IT8 = 25 i = 45.79 microns

For 'f' shaft, fundamental deviation is =

$-5.5(D)^{0.41} = -30.11\,\text{microns}$

Shaft limits are

Upper limit = $60 - 0.03011 = 59.969\,\text{mm}$

Lower limit = $60 - (0.03011 + 0.04579)$

= 59.924 mm

16.3 (c)

Minimum clearance = Minimum hole size – Maximum shaft size

$\therefore$ Maximum shaft size = 40.00 – 0.01 = 39.99 mm

Tolerance on shaft = 0.04 mm

Minimum shaft size = 39.99 – 0.04 = 39.95 mm

Maximum clearance = Maximum hole size – minimum shaft size

= 40.05 – 39.95 = 0.10

16.4 (a)
16.5 (d)
16.6 (a)
16.7 (b)

Minimum clearance

$\Rightarrow$ Minimum hole – Maximum shaft

$= 25 + 0.020 - (25 + 0.005)$

$= 0.015\,\text{mm}$

16.8 (c)

Go size = maximum material limit of component = Lower limit of hole

16.9 (d)

H7 is for hole where 7 indicates its tolerance grade

g8 is for shaft where 8 indicates its tolerance grade

16.10 (d)

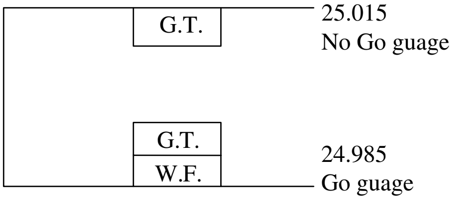
25 ± 0.015

GO-Gauge

U.L = 24.988

L.L = 24.985

$24.985^{+0.003}_{-0.000}$



16.11 (c)

For 60° thread angle, best wire size = 0.57135 × P = 1.154

16.12 (d)

Tolerance on one side = 0.01 + 0.032
= 0.042 mm

⇒ Tolerance on both sides = 0.042 (2)
= 0.084 mm

Size of Go gauge = 25 + 0.084
= 25.084 mm

16.13 (c)

Maximum interference = maximum size of shaft – minimum size of hole

$= 25.040 - 25.020 = 20 \mu m$

16.14 (a)

16.15 (b)

Hole = $20^{+0.05}_{+0.01}$

Hole tolerance $= 20.05 - 20.01 = 0.04$ mm

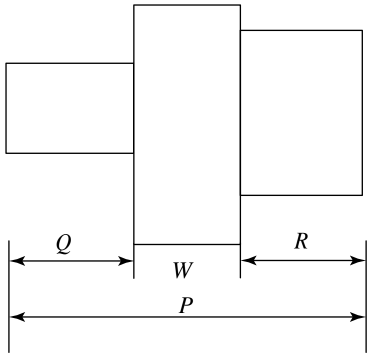
Gauge tolerance = $0.04 \times 0.1 = 0.004$ mm

Size of Go-gauge $= 20.01 + 0.004 = 20.014$

No-Go gauge = $20.05 - 0.004 = 20.46$

16.16 (b)

16.17 (b)



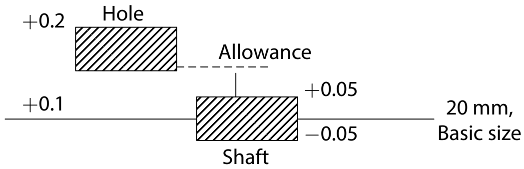
$R = 13.00^{+0.04}_{-0.02} = 13.01^{\pm 0.03}$

$P - (Q + R) = 35 - (12 + 13.01) = 9.99$

$\text{Tol} = 0.08 + 0.02 + 0.03 = 0.13$

$w = 9.99 \pm 0.13$

16.18 (c)



$\text{Allowance} = A_{min} - B_{max} = 20.10 - 20.05$
$= 0.05$ mm

∗Allowance is positive, Hence a clearance fit.

Chapter 17

17.1 (b)

17.2 (d)

In incremental system. Co-ordinates of point Q are (4, 7).

17.3 (b)

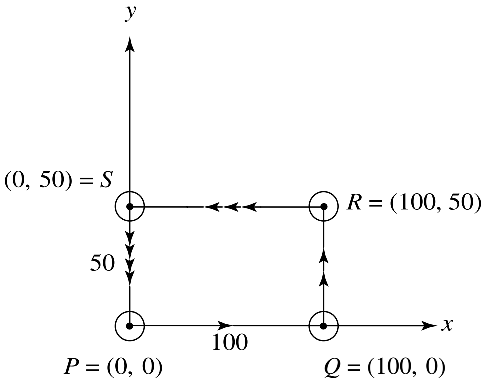
For incremental coordinates (G91) and coordinates of final point are (–10, –10). The tool moves CCW (counter clockwise), So G03.

17.4 (a)

End mill centre = (0,0)

Since Radius of end mill is 8 mm

∴ call point 'p' = –8, –8



Call point 'Q' = (100 + 8, –8 + 0) = (108, –8)

→x direction

Call point 'R' = (108 + 0, 50 + 8) = (108, 58)

→y direction

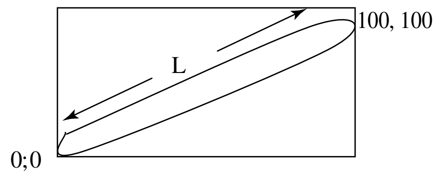
Call point 'S' = (108 – 100 –2 × 8, 58 – 0) = (–8, 58)

→ –x direction

Call point 'P' = (– 8 – 0, 50 – 50 – 8) = (–8, –8)

→ –y direction

17.5 (b)



$$\text{time } = \frac{L}{\text{feed}} = \frac{100\sqrt{2}}{50}$$

$$\Rightarrow 2 \times \sqrt{2} \times 60 = 170 \text{ seconds}$$

17.6 (c)

N002 → Circular interpolation of clockwise direction.

G91 → Incremental dimensions.

17.7 (c)

17.8 (a)

17.9 (a)

17.10 (d)

17.11 (a)

17.12 True

Bibliography

1. Liu, K., Lauwers, B., & Reynaerts, D. (2010). Process capabilities of Micro-EDM and its applications. *The International Journal of Advanced Manufacturing Technology*, *47*(1), 11-19.)
2. Jain, V. K., Sidpara, A., Ravisankar, M., & Das, M. (2012). Micromanufacturing: an introduction. Jain, V. K. ed. *Micromanufacturing Processes*, 1-37, CRC press.
3. Lu, Z., & Yoneyama, T. (1999). Micro cutting in the micro lathe turning system. *International Journal of machine tools and manufacture*, *39*(7), 1171-1183.
4. Aurich, J. C., Engmann, J., Schueler, G. M., & Haberland, R. (2009). Micro grinding tool for manufacture of complex structures in brittle materials. *CIRP Annals-Manufacturing Technology*, *58*(1), 311-314.
5. Ohmori, H., Ebizuka, N., Morita, S., Yamagata, Y., & Kudo, H. (2001). Ultraprecision micro-grinding of germanium immersion grating element for mid-infrared super dispersion spectrograph. *CIRP Annals-Manufacturing Technology*, *50*(1), 221-224.
6. Ikeno, J. I., Tani, Y., & Sato, H. (1990). Nanometer grinding using ultrafine abrasive pellets—manufacture of pellets applying electrophoretic deposition. *CIRP Annals-Manufacturing Technology*, *39*(1), 341-344.
7. Masuzawa, T., & Tönshoff, H. K. (1997). Three-dimensional micromachining by machine tools. *CIRP Annals-Manufacturing Technology*, *46*(2), 621-628.
8. Raju, L., & Hiremath, S. S. (2016). A State-of-the-art review on micro electro-discharge machining. *Procedia Technology*, *25*, 1281-1288.
9. Schuster, R., Kirchner, V., Allongue, P., & Ertl, G. (2000). Electrochemical micromachining. *Science*, *289*(5476), 98-101.
10. Bhattacharyya, B. (2010). Electrochemical Micromachining. Jain, V. K. ed. *Introduction to Micromachining*, 13.1-13.32, Alpha Science Int. Ltd., Oxford.

Index

Symbols

A

B

C

F

G

H

I

N

Q

R

S

T

U

V