MACHINIST

NSQF LEVEL - 4

2nd Year

TRADE PRACTICAL

SECTOR : CAPITAL GOODS & MANUFACTURING

(As per revised syllabus July 2022 - 1200Hrs)

Directorate General of Training

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

NATIONAL INSTRUCTIONAL
MEDIA INSTITUTE, CHENNAI

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

Sector : **Capital Goods and Manufacturing**

Duration : **2 Year**

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

Developed & Printed by

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

First Edition : November 2023 Copies : 1000

Rs.390/-

FOREWORD

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

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

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

Jai Hind

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

November 2023
New Delhi - 110 001

PREFACE

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

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

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

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

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

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

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

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

Chennai - 600 032 **EXECUTIVE DIRECTOR**

ACKNOWLEDGEMENT

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

MEDIA DEVELOPMENT COMMITTEE MEMBERS

Shri. A. Vijayaraghavan	-	Assistant Director of Training (Retd.), ATI, Chennai - 32.
Shri. Suriya Kumar	-	MTAB Technology Center (P) Ltd., Tamilnadu.
Shri. J.D. Dinesh Kumar	-	MTAB Technology Center (P) Ltd., Tamilnadu.
Shri. H.A. Manukumar	-	Training Officer, NSTI, Chennai.
Shri. M. Sampath	-	Training Officer (Retd.), CTI, Chennai - 32.

NIMI CO-ORDINATORS

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

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

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

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

INTRODUCTION

TRADE PRACTICAL

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

The manual is divided into Six modules

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

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

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

TRADE THEORY

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

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

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

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

CONTENTS

LEARNING / ASSESSABLE OUTCOME

On completion of this book you shall be able to

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

SYLLABUS

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

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

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

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

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

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

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

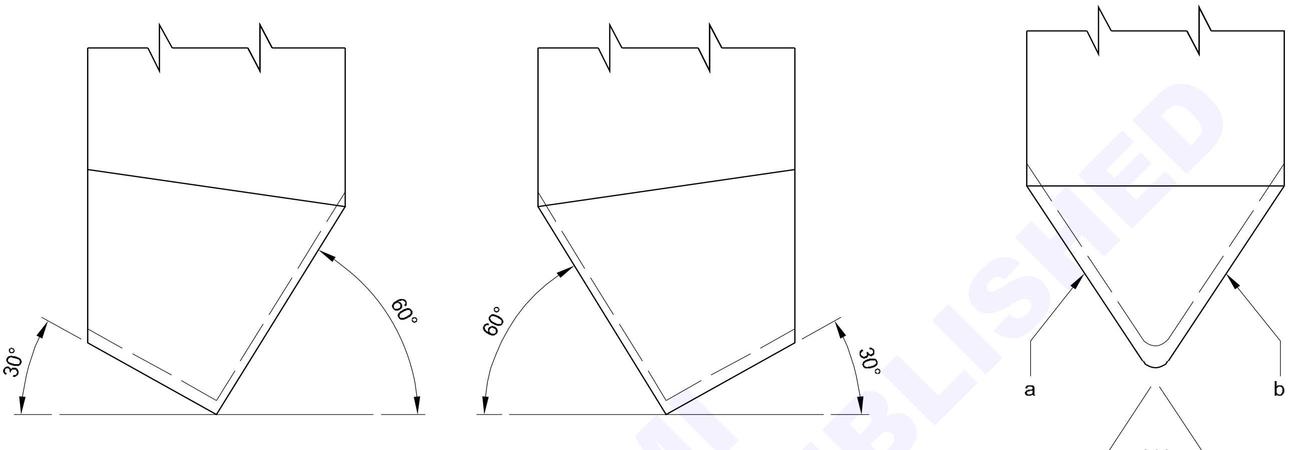
Duration	Reference Learning Outcome	Professional Skills (Trade Practical) with Indicative hours	Professional Knowledge (Trade Theory)
		186.Program checking in dry run, single block modes, on CNC simulator. (04 hrs.) 187.Checking finish size by over or under sizing through tool offsets, on CNC simulator.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

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

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

Grinding different shape of single point tools

Objectives : At the end of this exercise you shall be able to
- **operate a tool and cutter grinder**
- **mount and use cup wheel for tool grinding**
- **grind-single point cutting tools with the specified clearance, rake angles and cutting angle accurately (within ± 30 min) using tool and cutter grinder.**



1. R.H.TOOL 2. L.H.TOOL 3. V TOOL

Job Sequence

TASK 1: **Sharpen a Straight Turning Tool (RH) on T&C Grinder**

- Grind four sides square to 19 mm on a surface grinder.
- Rough grind the tool profile on a pedestal grinder.
- Use a simple protractor to check the clearance, rake and cutting angles.
- Fix taper cup wheel on the spindle of the tool and cutter grinder and dress its face.
- Fix universal vice on the work table and hold the tool for grinding the side cutting edge.
- Set the vice to 30º in the horizontal plane and 8º in the vertical plane and grind the side cutting edge. (Fig 1)
- Set the vice to 60º in the horizontal plane and 8º in the vertical plane and grind the end cutting edge. (Fig 2).

Fig 1

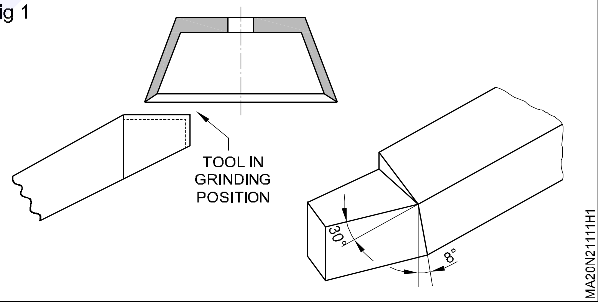
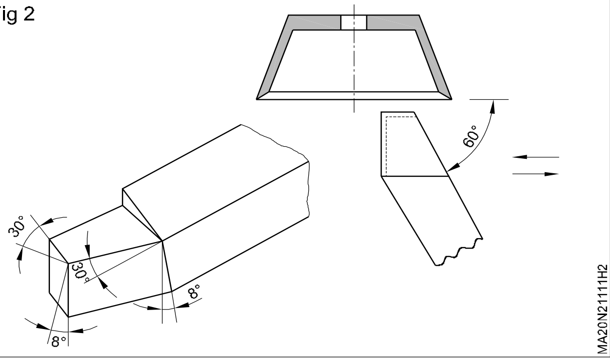


Fig 2

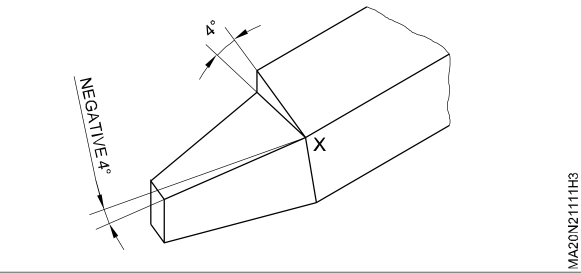


3	□ 20 x 150	Fe310	-	-	-	2.1.111
NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.

SCALE NTS	GRINDING OF DIFFERENT SINGLE POINT TOOLS	DEVIATIONS	TIME :
		CODE NO. MA20N21111E1	

- Mount the disc wheel 150 mm on the machine spindle and dress the face with an abrasive stick.
- Set the vice to 30º in the horizontal plane, 14º in the vertical plane. Hold the tool for 4º negative top rake using a combination bevel protractor and spirit level.
- Grind the top rake (4º negative) and side rake (14º positive) by traversing the table until the grinding wheel face just touches the point 'X'. (Fig 3).
- Remove the tool and hone the nose radius with a fine grit abrasive stick without spoiling the cutting edges.

Fig 3



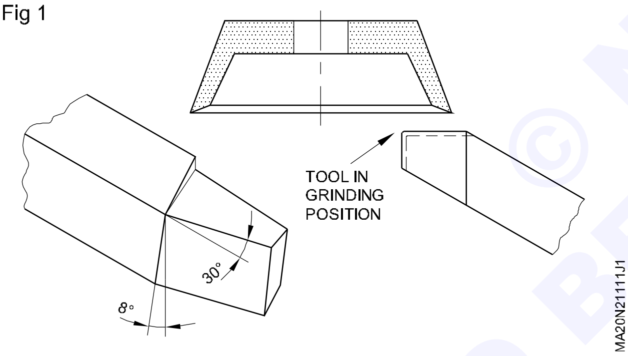
- Deburr.

— — — — — — — —

TASK 2: **Sharpen a straight turning tool (LH) on T & C grinder**

- Grind four sides square to 19mm on a surface grinder.
- Rough grind the tool profile on a pedestal grinder.
- Use simple protractor to check the clearance, rake and cutting angles.
- Fix taper cup wheel on the spindle of the tool and cut ter grinder and dress its face.
- Fix universal vice on the work table and hold the tool for grinding the side cutting edge.
- Set the vice to 30° in horizontal plane and 8° in the vertical plane and grind the side cutting edge (Fig 1)

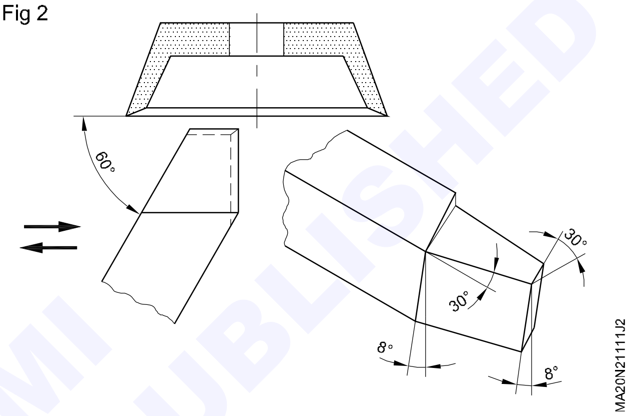
Fig 1



- Set the vice to 60° in the horizontal plane and 8° in the vertical plane and grind the end cutting edge. (Fig 2)

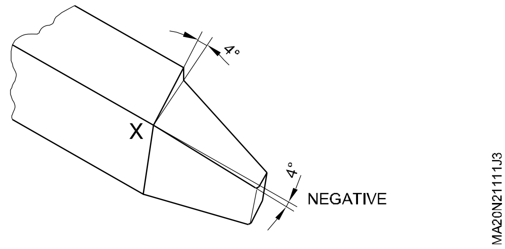
  Mount the disc wheel 150 mm on the machine spindle and dress the face with an abrasive stick.
- Set the vice to 30° in the horizontal plane, 14° in the vertical plane. Hold the tool for 4° negative top rake using a combination bevel protractor and spirit level.

Fig 2



- Grind the top rake (4° negative) and side rake (14° positive) by traversing the table until the grinding wheel face just touches the point 'X' (Fig 3)

Fig 3



- Remove the tool and hone the nose radius with a fine grit abrasive stick without spoiling the cutting edges.
- Deburr.
- Check the angles using bevel protractor.

— — — — — — — —

TASK 3: **Sharpen a 'V' tool on T & C grinder**

- Sharpen a 'V' - Tool (Shaper) on tool and cutter grinder.
- Rough grinding the tool profile on a pedstal grinder or off hand grinding before finish grinding on tool and cut- ter grinder.
- Inspect and mount the flaring cup wheel on the ma- chine spindle.
- Dress the grinding wheel on the machine.
- Mount the universal vice on the table make sure that all the scales are set to zero degrees. (Fig 1)
- Swivel the scale 'A' to 30 degree in anticlock wise di- rection. (Fig 2)
- Swivel the scale 'B' to 3 degree in anticlock wise di- rection. (Fig 2)

Fig 1

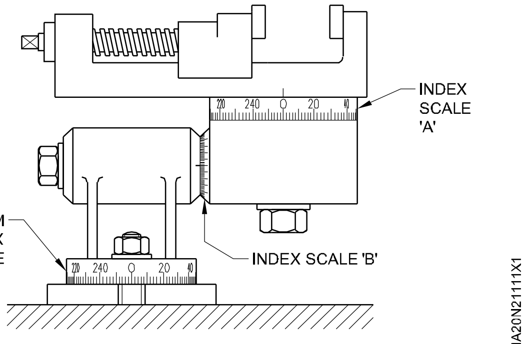
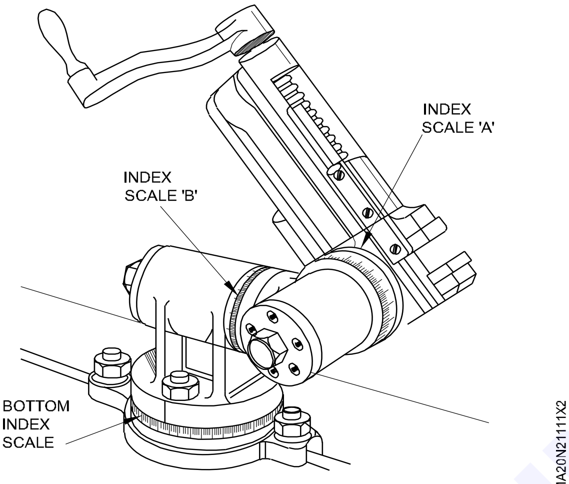
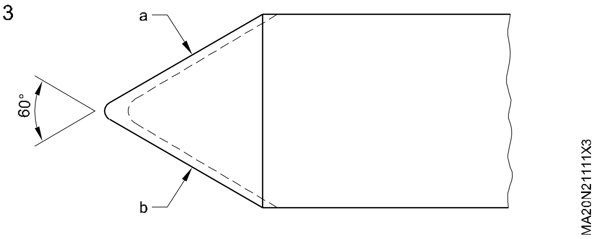


Fig 2



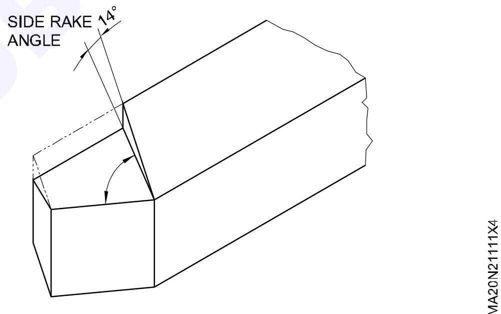
- Fix the tool on the vice such that the face of the tool is on top side and set the stop dogs.
- Start the machine rough and finish grind the side 'a' (Fig 3)

Fig 3



- Remove the tool and refix the tool keeping top side butting on the vice base (up side down).
- Swivel teh scale 'B' to 3 degree in clock wise direction from zero degree.
- Do not change the angle on scale 'A'
- Rough and finish grind side 'b'
- Check the angles with protractor.
- Remove the tool from the vice set all scales to zero degree.
- Fix the tool in vertical position keeping the face of the tool towards the wheel.
- Swivel the scale 'B' to 14 degree in clock wise direction to grind top rake. (Fig 4)

Fig 4



— — — — — — — —

Skill Sequence

Preparing tool and cutter grinder for re - sharpening of single point cutting tool

Objectives : This shall help you to

- **prepare the tool and cutter grinder ready for re - sharpening work**
- **study the machine and understand the purpose and effect of every handle and hand wheel.**

Study the machine and understand the purpose every handle and hand wheel.

Refer to the Instructional manual supplied by the machine manufactures for function and identification of each operating control.

Clean thoroughly the working surface of the work table and other working area with a clean banian waste.

Ascertain the position of every lubricating point and see that all get the required quantity of correct grade of oil regularly.

Make sure that the oil in the wheel head, the work head and the travere gearbox is maintained at the correct level with the oil recommended by the machine manufacturers.

Check the lubrication of the table ways each morning as they wear rapidly if allowed to run dry.

Avoid spots of oil on the grinding wheel while refilling.

Always warm up the machine before commencing to grind or to dress the wheel.

Be familiar with the following electrical controls.

- Wheel spindle - 'ON - OFF' switch for changing the direction of rotation.
- Switch for power elevating 'UP' and 'DOWN'
- Dust extractor - 'ON-OFF' switch.

Be familar with the following mechanical arrangements

- Swivel the wheel head to the required angle and locking it firmly.
- Adjust the height of the wheel head and direction of rotation of the hand wheel manually for 'UP' & 'DOWN' movement and the method of locking it.
- Swivel the work table to the required angle and lock it.

Remove Taper dowels if provided for '0' setting before swivelling the table.

Cross - traverse movement of the work table and remember the value of the graduations on the graduated dial.

- Lock the work table in position.

— — — — — — — —

Mount a grinding wheel on tool and cutter grinder

Objective: This shall help you to
- **mount a grinding wheel on the tool and cutter grinder.**

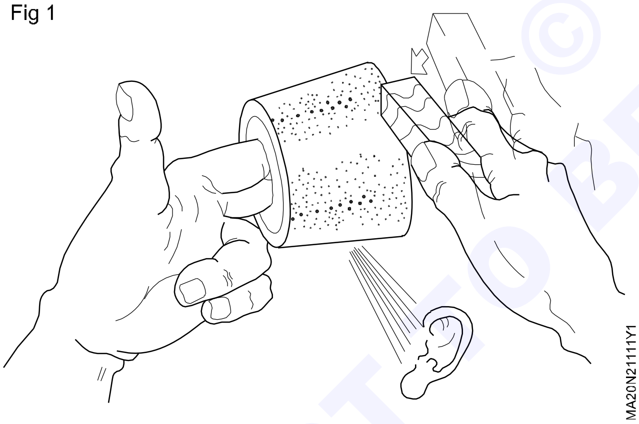
Select and inspect grinding wheel: Select the wheel to be mounted and visually check for cracks.

Check the bore of wheel is the same size as the spigot diameter of the adopter.

" Ring test", for the wheel by holding loosely with fingers in the bore and tap lightly with a piece of wood and listen for a distinct ringing sound to denote wheel is free from cracks. (Fig 1)

If the sound is a dull, the wheel is cracked and should not be used.

Fig 1



Mount grinding wheel

Clean the wheel adopter.

Clean the side of the wheel.

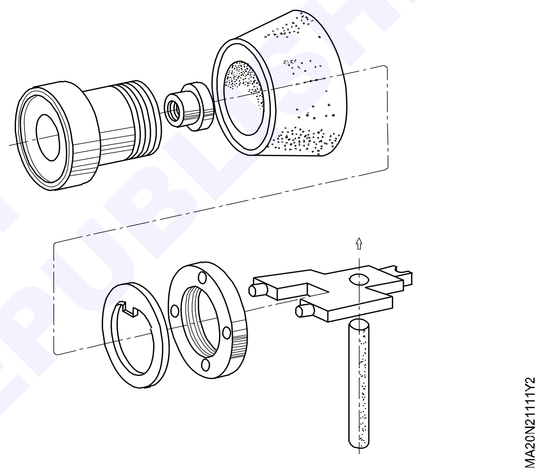
Ensure that paper washer is on each side of the wheel before mounting.

Fit wheel to spigot.

Place locking washer over spigot and up to the front face of the wheel.

Screw on lock nut by hand and tighten with wheel key. (Fig 2)

Fig 2



Dressing a cup grinding wheel

Switch on wheel head spindle.

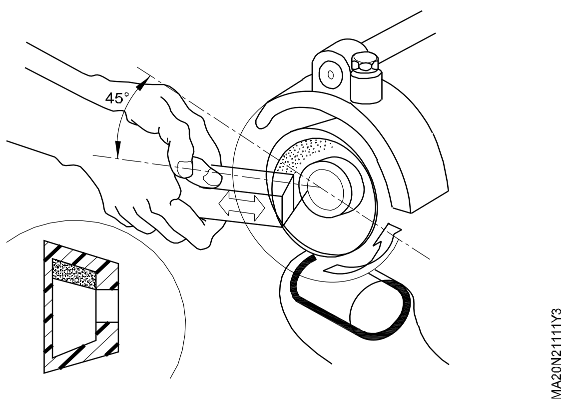
Hold dressing stick firmly with both hands at one end, at a suitable angle.

Move forward until gently touches front inside face of the wheel (Fig 3)

Slowly move dressing stick back ward and forward until wheel space to a thin pointed edge.

When wheel looks black and shining, it needs dressing.

Fig 3



Demonstrate and practice of grinding of slab milling cutter

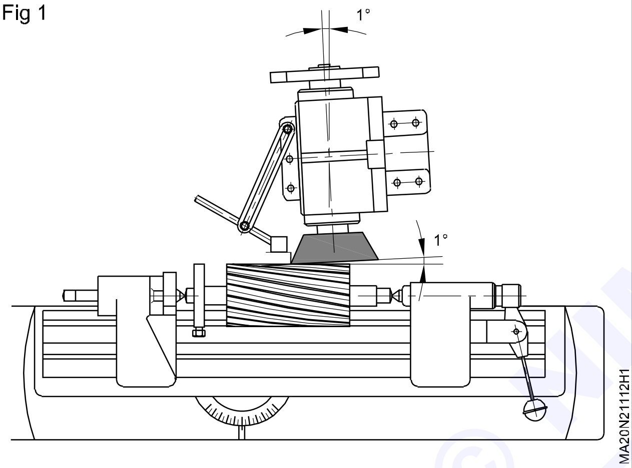
Objectives: At the end of this exercise you shall be able to
- **set the cutter and the wheel head for sharpening slab milling cutter**
- **sharpen the slab milling cutter.**

Job Sequence

Using linear setting method

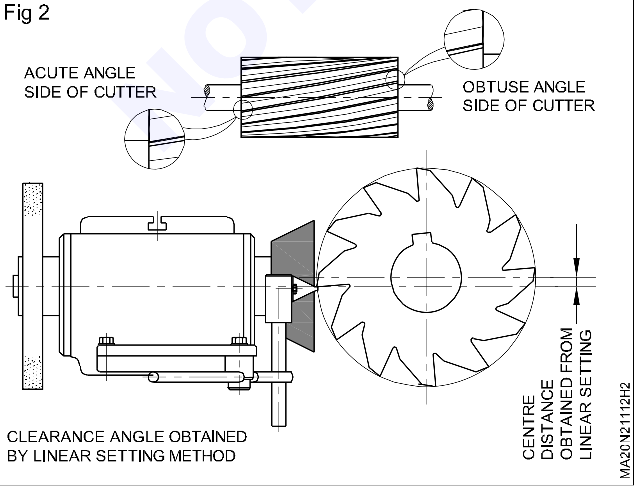
- Mount the cutter on the mandrel and between centres.
- Turn the wheel head to 1º to the table and lock in position. This prevents grinding on both sides of the wheel when the cutter passes in front of it. (Fig 1)

Fig 1



- Set the table graduations at '0' i.e. table set parallel to table movement.
- Adjust the grinding wheel centre height with a height setting gauge.
- Bolt the tooth rest holder to the wheel head and set the tooth rest to the spiral angle of the cutter.
- Set the point of tooth rest on the acute angle side of the cutter and at the same centre height of the table centres with a centre setting gauge. (Fig 2)

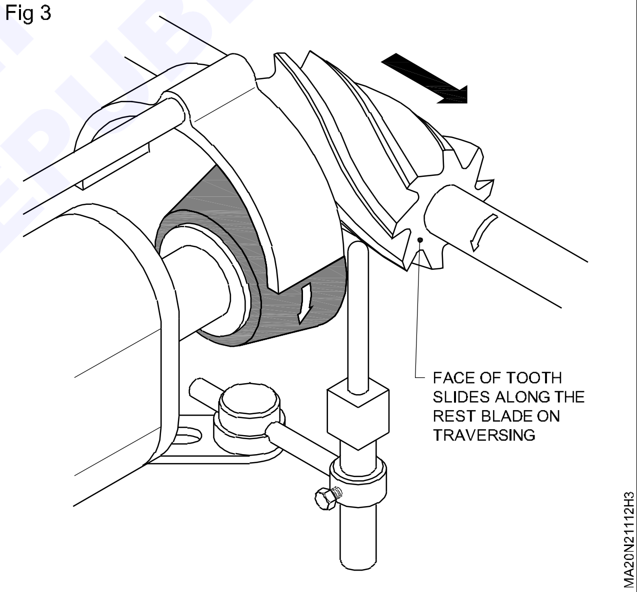
Fig 2



- Lower the wheel head (for off-set) an amount calculated to the required clearance angle formula
- Amount of off-set = Cutter diameter x clearance angle required x 0.0087
- Start wheel head motor, contact the first tooth with the grinding wheel.

Traverse the work table carefully across the wheel face using hand traverse. Maintain a turning force on the cutter to hold the tooth firmly against the tooth rest. (Fig 3)

Fig 3



Do not over traverse the cutter tooth being ground past the tooth rest blade.

During traverse, the tooth face moves over the tooth rest to present the edge of the particular tooth continuously at the correct angle to the wheel. Great care is necessary in setting the tooth rest for achieving the accuracy.

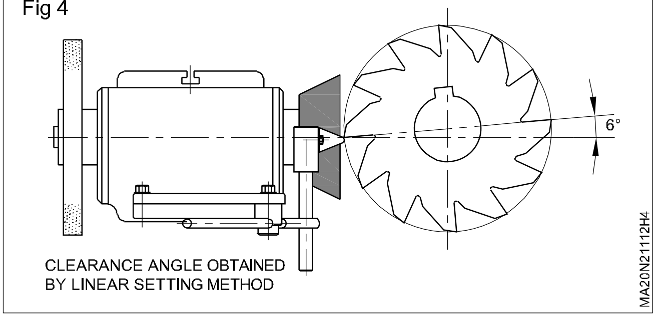
- Grind the tooth opposite to the 1st tooth already ground.
- With an O.S.micrometer check for taper across each end of the ground tooth. If necessary, correct it by aligning the work table.
- Rotate the cutter to bring the next tooth on to the tooth rest and continue until all teeth have been ground.

Using angular setting gauge method

Repeat steps as in linear setting method

- Set the point of tooth rest at acute angle side of the cutter and at the same centre height of the wheel head such that the outer rim of the wheel and the high point of the tooth rest coinside in a point. (Fig 4)

Fig 4



- Bolt the centre setting gauge to the wheel head, rotate the cutter until a point on the face of the cutter, (acute angle side) is in contact with the tip of the gauge. This point of cutter is now on its own centre line.
- Clamp an angular setting gauge to the mandrel with the gauge set at '0'. Rotate the cutter to the required clearance angle and lock in position.
- Lower the wheel until the tooth rest is on the same level as the cutter tooth, with the cutting edge of the wheel opposite to the acute point of cutter tooth. (Fig 4) Unlock the setting gauge, start the wheel head motor and commence grinding.
- Repeat steps as in the linear setting method and complete the grinding.

— — — — — — — —

Skill Sequence

Sharpening the cutting face of a slab milling cutter

Objective: This shall help you to

- **sharpen the cutting face of a slab milling cutter.**

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

This operation is done in two stages.

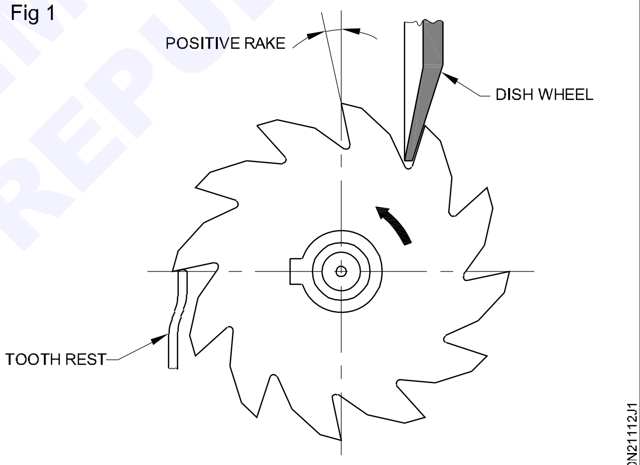
i Gashing, and

ii grinding the cutting face.

Gashing or roughing: the depth of flute is deepened by this operation. Proceed as follows.

- Mount the dish wheel on the extension spindle and true it.
- Mount the cutter on the mandrel and set in between centres.
- Swivel and align the wheel head with the hand of helix and helix angle of the flute, so that the wheel touches the bottom of the flute but not the cutting face of the flute.
- Fix the tooth rest to the wheel head and set its blade touching the cutting face of the flute. (Fig 1) Set table stops.
- Start the wheel head motor and commence grinding by traversing the table and at the same time keeping in constant touch with the tooth rest blade by manually applying a turning force to the mandrel.
- Lower the wheel head for successive depth of cut, until you reach the required depth of flute.

Fig 1



OFF-SET FOR POSITIVE RAKE ANGLE

Avoid heavy cut to avoid the rapid wheel wear.

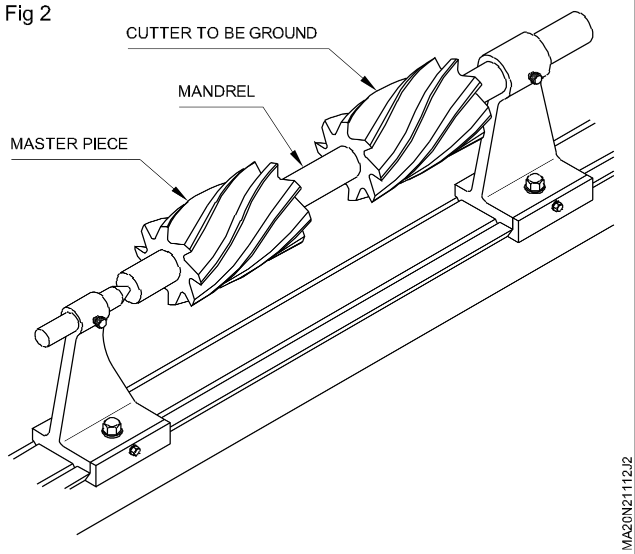
Grinding the cutting face

Mount the dish wheel on the extension spindle in inverted position.

Obtain a new helical fluted cutter from stores/tool grib equal to the number of teeth, helix angle, hand of helix and diameter (approximately) of the cutter to be sharpened. This is to be used as a masterpiece during grinding of the cutting face.

Mount the masterpiece and the cutter to be face-ground on a long mandrel (Fig 2) in perfect alignment and in between centres.

Fig 2



Swivel and align the wheel head with the hand of helix and helix angle of flute, so that the tapered face of the wheel, perfectly matches with the cutting face of the tooth of the cutter to be ground.

Fix the tooth rest with a micrometer adjustment to the wheel head, set its blade touching the cutting face of the tooth of the masterpiece cutter, as done in the gashing operation.

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

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

Lower the tip of the blade of the tooth rest by micrometer collar adjustment, for successive depth of cut {radial cut (Fig 1) see the arrow head} until the entire cutting face is ground forming a keen cutting edge on the tooth.

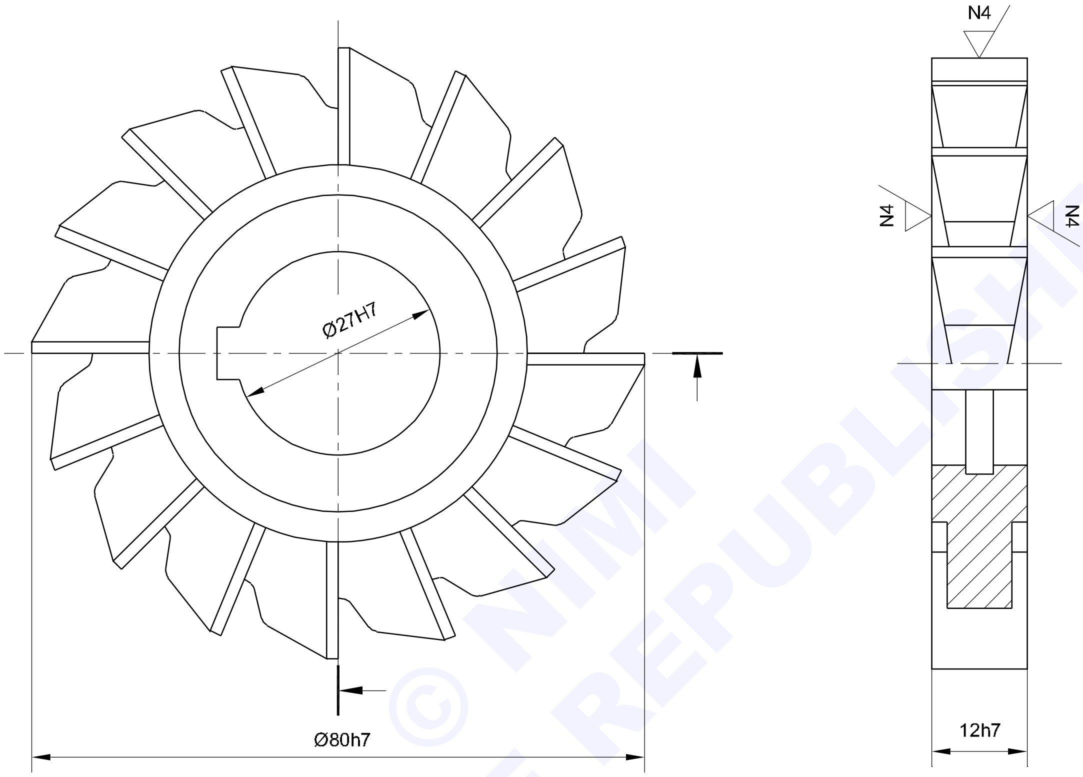
Apply only light cuts. Heavy cuts may cause wheel loading or burning of the cutting edge.

While aligning the wheel, if the cutter is of a quick helix type, utmost care must be taken to see that the wheel does not touch the next tooth.

Grind and complete the cutting face of all the teeth of the cutter.

Re-sharpening side and face milling cutter

Objectives : At the end of this exercise you shall be able to
- **sharpen peripheral teeth of straight fluted side and face cutter**
- **sharpen side teeth of side and face cutter to the given thickness.**



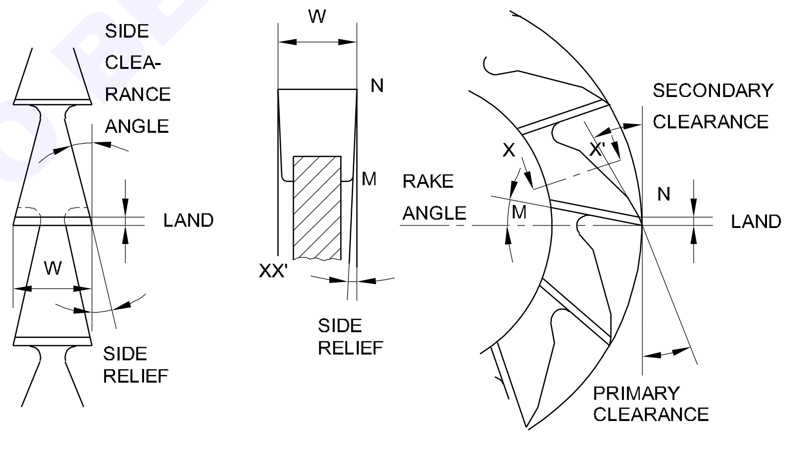
NOTE:

GRIND 7° PRIMARY CLEARANCE
25° SECONDARY CLEARANCE

ON PERIPHERAL TEETH

7° PRIMARY CLEARANCE
15° SECONDARY CLEARANCE

ON SIDE TEETH



Ø20H7 - $20^{0.021}_{0.000}$
Ø20h7 - $20^{+0.000}_{-0.021}$
Ø80h7 - $80^{+0.000}_{-0.035}$

N4 SMOOTH MACHINING Ra - 0.2

1	-		-	-	-	2.1.113
NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.

SCALE 1:2	**RE-SHARPENING SIDE AND FACE MILLING CUTTER**	DEVIATIONS	TIME :
		CODE NO. MA20N21113E1	

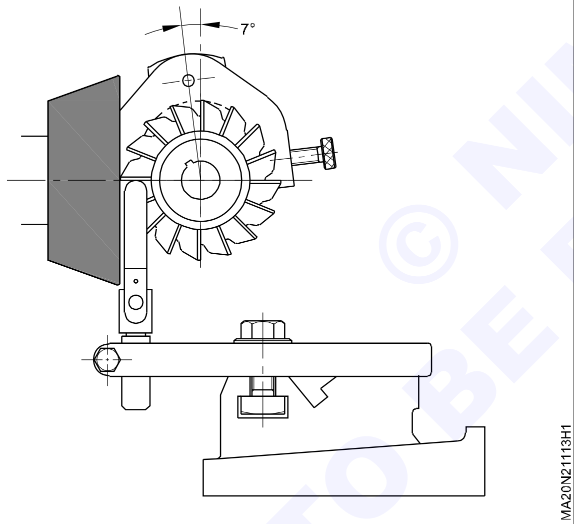
Job Sequence

Grinding primary clearance angle of peripheral teeth of a side and face milling cutter

- Mount taper cup wheel and true it.
- Adjust the grinding wheel centre height with a height setting gauge.
- Set the table graduations at 0°.
- Turn the wheel head to 1° or 2°. This prevents grinding on both sides of the grinding wheel.
- Mount the cutter on the mandrel between centres.
- Set the cutter to primary clearance angle (7°) (Fig 1), by using an angular setting gauge or by any other suitable method.
- Set the tooth rest under the first tooth to be ground and unlock angular setting gauge.

> **Where possible, place the tooth rest under the side teeth, as this gives greater clearance between the wheel and tooth rest, and offers greater convenience for adjustment. (Fig 1)**

Fig 1

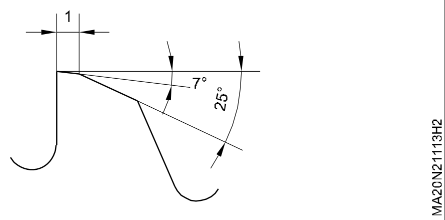


- Start the wheel head motor and commence grinding the 1st tooth.
- Sharpen the opposite tooth and check for dimension and taper.
- Correct, if necessary.
- Complete the grinding of all the teeth.

Grinding secondary clearance

- Rotate the cutter to 25°, for secondary clearance.
- Position the wheel such that the next teeth will not be ground by the grinding wheel.
- Start grinding the secondary clearance on the first tooth. Check the land width. It should be 1 mm approximately. (Fig 2)

Fig 2



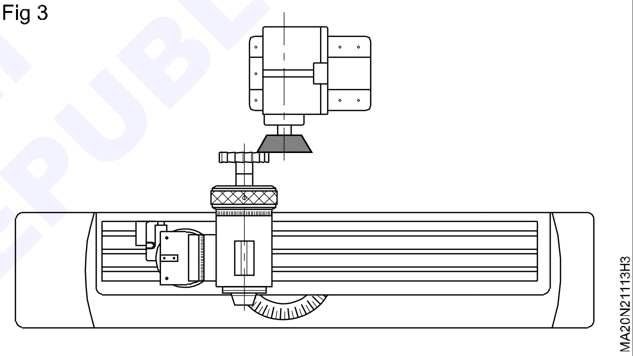
- Repeat and complete all the other teeth leaving uniform width of land on all peripheral teeth.

For grinding side teeth on a side and face milling cutter (primary clearance)

Primary clearance

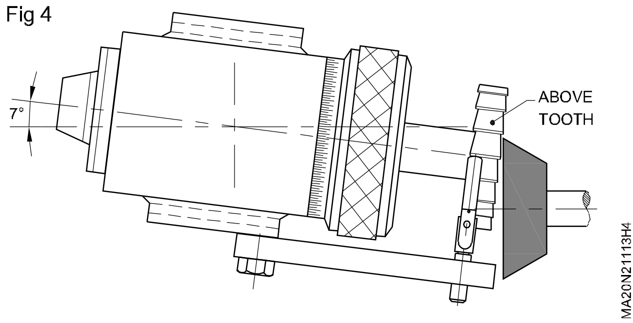
- Mount the cutter on the stub arbor with sufficient spacers behind the cutter and fix the arbor in the cutter head. Spacers are provided to allow the grind wheel to grind the other side and avoid resetting.
- Set the table graduations at '0'.
- Swivel the cutter head to 90° (Fig 3)

Fig 3



- Tilt the cutter head to 7° for primary clearance angle. (Fig 4)
- Fix the tooth rest on the cutter head and adjust the tooth rest until it positions the first tooth to be ground parallel to the table top. This is a visual setting.

Fig 4



- Adjust the wheel head height to cover the first tooth to be ground without fouling the tooth above. (Fig 1)
- Set the stop dogs.
- Start the wheel head motor and grind the first tooth. Repeat and complete all the teeth.

Secondary clearance

- Tilt workhead to 15°, and adjust the height of the wheel head, to clear the above tooth.
- Grind secondary clearance on all teeth leaving 1 mm land.

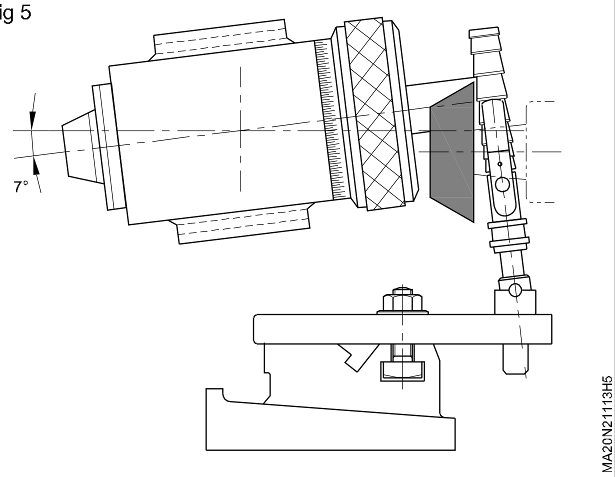
Grinding inside edge (other side) of the cutter (Primary clearance)

- Mount the wheel in an inverted position on the extension spindle and true. (Fig 5)
- Tilt cutter head spindle to 7° in the opposite direction of the previous setting.
- Fix the tooth rest holder to the table, and adjust it to position the first tooth parallel to the top of the table. (Visual setting)

> **Fixing the tooth rest holder on the wheel head in this set up, may cause the wheel to foul the tooth rest. Take care of it.**

- Start grinding primary clearance, on all the teeth.

Fig 5



Secondary clearance

- Tilt the cutter head to 15° and adjust the wheel head position and tooth rest position to grind the secondary clearance.
- Grind secondary clearance on all the teeth leaving 1 mm land.

Skill Sequence

Sharpening the cutting face of straight flute cutter

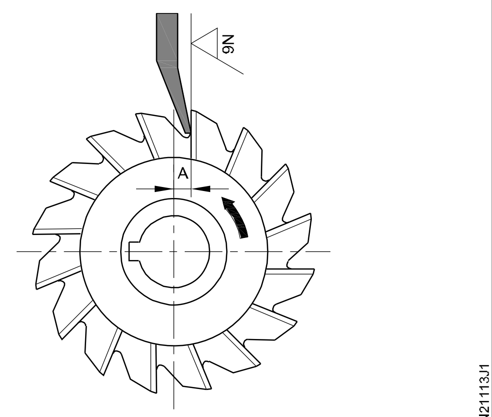
Objective: This shall help you to
- **sharpen the cutting face of a straight flute cutter.**

On repeated re-sharpening the cutting face of the milling cutter becomes short , and thus reduces the chip space. Under such condition the flute has to be deepended by maintaining the original positive rake angle on the cutting face.

To grind the cutting face of straight flute, follow the steps given below.

- Mount the dish wheel on the extension spindle and true.
- Mount the cutter, on the stub arbor with sufficient spacers at the back of the cutter and fix it with the cutter head.
- Set the index-drum to suit the number of teeth of the cutter.
- Lower the wheel head, adjust the cross-feed movement, and align the cutting face parallel and touching the rotating face of the wheel, by adjusting the index drum adjusting screws. (Fig 1)
- Start the wheel head motor and commence grinding.

Fig 1

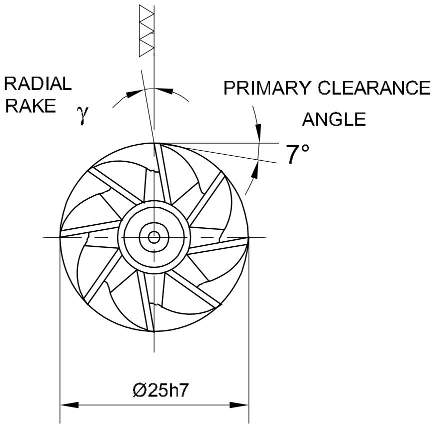


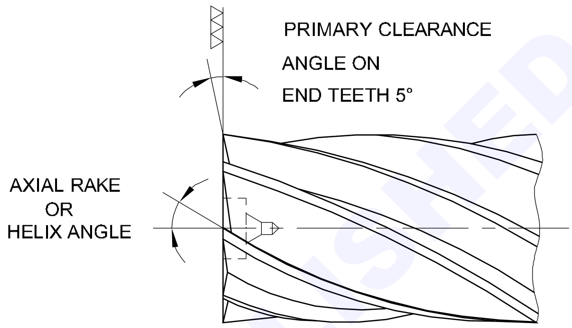
A = OFF-SET FOR POSITIVE RAKE

> **Apply only light cuts, without burning the cutting edge.**
>
> **Apply depth of cut by turning the adjusting screw of the index-drum, not by cross-feed movement (see the arrow in the Fig 1) which will increase the original positive rake angle.**

Grinding of end mill cutter

Objectives : At the end of this exercise you shall be able to
- **set tool and cutter grinder for sharpening end mill cutter**
- **sharpen peripheral teeth of helical fluted teeth**
- **sharpen end teeth of end mill cutter.**





RECOMMENDED TOOL GEOMETRY (END MILL CUTTER)

MATERIAL	MILLING OPERATION	TOOL MATERIAL	AXIAL RAKE	RADIAL RAKE	CLEARANCE ANGLE ON END TEETH	PRIMARY CLEARANCE				
						DIA mm				
STEEL OPERATION	END	H.S.S	30 TO 35	10 TO 20	3 TO 7	6	10	12	16	20
						12°	11°	10°	9°	8°
CAST IRON MACHINABLITY	"	"	"	12	"	"	"	"	"	"
ALUMINIUM	"	"	30 TO 45	15 TO 20	8 TO 12	15°,	14°,	13°,	12°,	10°

NOTE:
Ø 25 h7 - $25^{0.000}_{-0.021}$

1	-	-	-	-	-	2.1.114
NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.

SCALE 1:1	GRINDING OF END MILL CUTTER	DEVIATIONS ±0.06	TIME :
		CODE NO. MA20N21114E1	

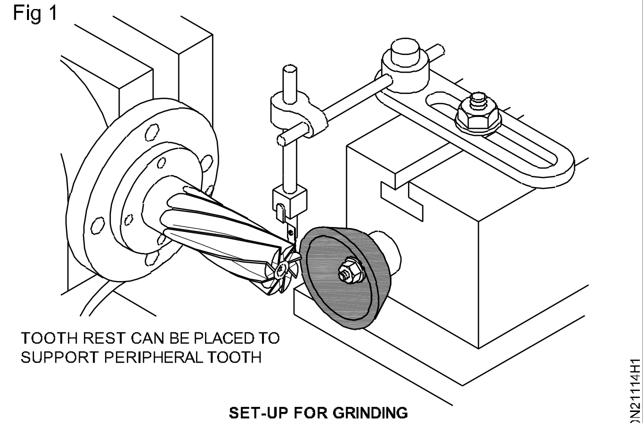
Job Sequence

Sharpen an end mill cutter

> - **An end mill cutter is sharpened in two separate operations.**
> - **Sharpening of peripheral teeth**
> - **Sharpening of end teeth.**

Peripheral teeth (Fig 1)

Fig 1



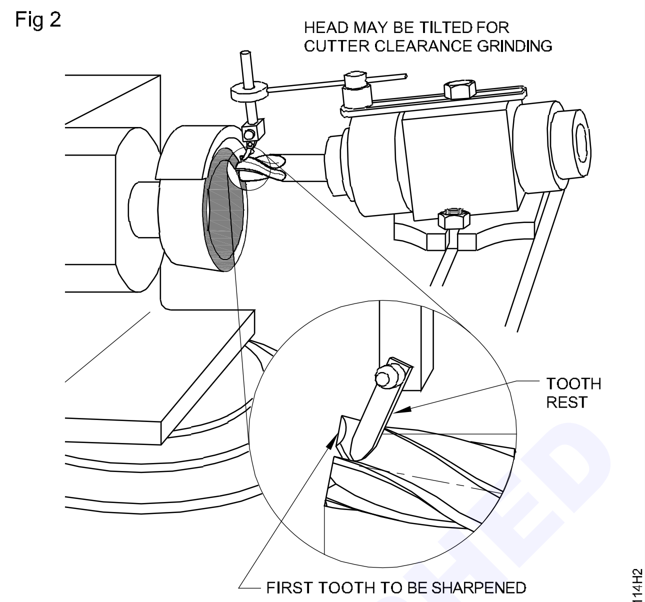
SET-UP FOR GRINDING PERIPHERAL TEETH OF END MILL

- Mount the end mill in the cutter head.
- Mount a taper cup wheel and true its cutting face.
- Align the axis of the wheel spindle and cutter head spindle using a centre gauge.
- Set the work head at zero graduation.
- Fix tooth rest to the wheel head and set it to the desired clearance angle (7°) and helix angle of the flute.
- Offset the wheel head axis by 1° or 2° to clear opposite rim of grinding wheel.
- Sharpen the edge of first tooth by traversing the wheel using the table traverse and at the same time rotate the cutter head spindle by hand following the helical path.
- Rotate the cutter through 180° and sharpen the opposite teeth.
- With a micrometer, check the ground teeth for taper.
- Adjust the cutter head if necessary to eliminate taper.
- Sharpen all other teeth.

End teeth

- Rotate the cutter head to 90°.
- Mount the tooth rest on the cutter head and set it to level with centre gauge.
- Tilt the cutter head to the required primary clearance angle i.e 7°. (Fig 2)

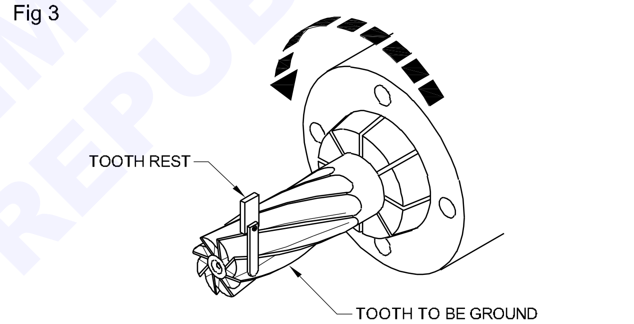
Fig 2



SET-UP FOR GRINDING END TEETH ON END MILL

- Set the first tooth on the tooth rest, parallel to the top surface of the table.(Fig 3)

Fig 3



SET-UP FOR GRINDING CLEARANCE ON END TEETH

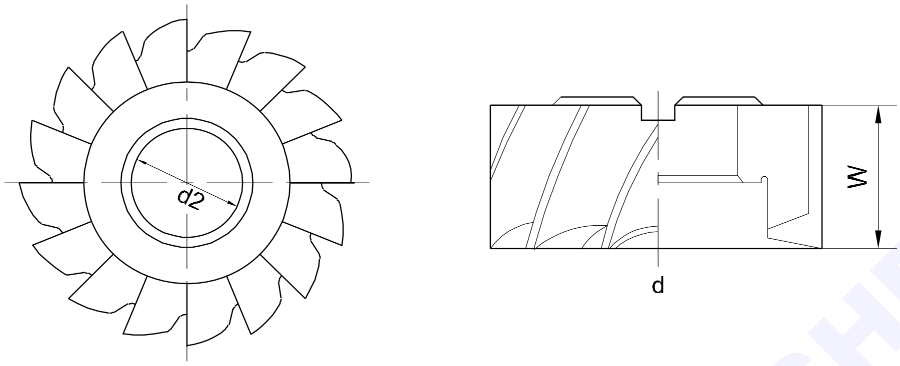
> **Tooth rest should support the peripheral tooth edge as close as possible to the edge of the end tooth.**
>
> **Sharpen the first tooth, taking only light cuts.**
>
> **Sharpen all the remaining teeth in turn.**

Secondary clearance

- Tilt the cutter head to 15° and adjust the tooth rest suitably to grind secondary clearance.
- Grind secondary clearance on all the teeth, leaving 1 mm land.

Re-sharpening of shell end mill cutter

Objective: At the end of this exercise you shall be able to
- **set the cutter to primary clearance angle and grind.**



A SHELL END MILL **B**

Job Sequence

Peripheral teeth

- Mount the shell end mill in the cutter head.
- Mount a taper cup wheel and true its cutting face.
- Align the axis of the wheel spindle and cutter head spindle using a centre gauge.
- Set the work head at zero graduation.
- Fix tooth rest to the wheel head and set it to the desired clearance angle (7°) and helix angle of the flute.
- Offset the wheel head axis by 1° or 2° to clear opposite rim of grinding wheel.
- Sharpen the edge of first tooth by traversing the wheel using the table traverse and at the same time rotate the cutter head spindle by hand following the helical path.
- Rotate the cutter through 180° and sharpen the opposite teeth.
- With a micrometer, check the ground teeth for taper.
- Adjust the cutter head if necessary to eliminate taper.
- Sharpen all other teeth.

End teeth

Primary clearance

- Rotate the cutter head to 90°
- Mount the tooth rest on the cutter head and set it to level with centre gauge.
- Tilt the cutter head to the required clearance angle i.e 7°
- Set the first tooth on the tooth rest, parallel to the top surface of the table.

Tooth rest should support the peripheral tooth edge as close as possible to the edge of the end tooth.

Sharpen the first tooth, taking only light cuts.

Sharpen all the remaining teeth in turn.

Secondary clearance

Tilt the cutter head to 15° and adjust the tooth rest suitably to grind secondary clearance.

Grind secondary clearance on all the teeth, leaving 1 to 3mm land.

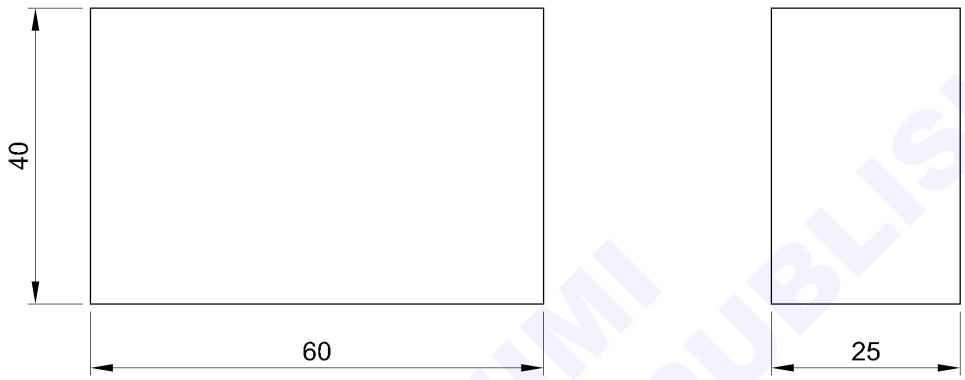
The clearance angle of the cutter will vary according to its diameter.

1	-	-	-	-	-	2.1.115
NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.

SCALE NTS	RESHARPENING OF SHELL END MILL CUTTER	DEVIATIONS	TIME :
		CODE NO. MA20N21115E1	

Practice of facing on milling machine

Objectives: At the end of this exercise you shall be able to
- **setting up of job on milling machine**
- **setting up of side and face milling cutter**
- **milling by side and face milling cutter.**



Job Sequence

- Check the raw material for the correctness of the size (deburr if necessary)
- Align the machine vice with reference to the column using a dial indicator.
- Mount ϕ 27 stub arbor and a shell end mill cutter of size ϕ 63 X 40 X 27 bore on vertical milling machine.
- Set the r.p.m of the cutter near to 100.
- Hold the work piece in the vice and align the milling cutter for machining.
- Move the longitudinal slide manually towards the cutter such that the workpiece comes in contact with the cutter gently, sudden contact may damage the workpiece and the cutter may break.
- Machine the block to size and check.

NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.
1	30x45x65	-	Fe310	-	-	2.2.116

SCALE NTS	FACE MILLING	DEVIATIONS ±0.06	TIME :
		CODE NO. MA20N22116E1	

Skill Sequence

Mount a stub arbor and shell end mill cutter on vertical milling machine

Objective: This shall help you to

- **mount a stub arbor and shell end mill cutter on a vertical milling machine.**

Set the lowest available spindle speed to avoid free rotation of the spindle nose.

Ensure that the machine is switched off to avoid any accident while fixing the stub arbor and cutter.

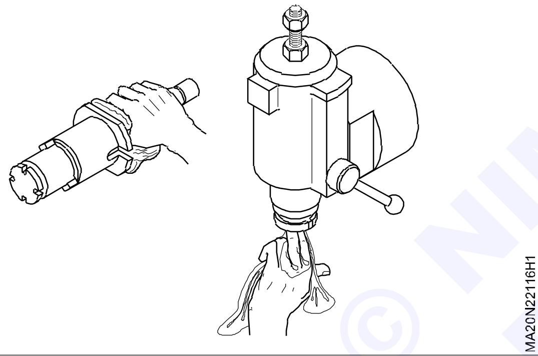
Clean the taper portion of the arbor and spindle nose to make it free from dust, chips etc and to ensure correct fitting and true running of the arbor. (Fig 1)

For cleaning use soft cloth free from dust, chips etc, to avoid scratches on the surfaces.

Insert the stub arbor taper shank into the spindle nose.

Hold the stub arbor in position and screw in the draw - bar from the top side of the spindle nose.

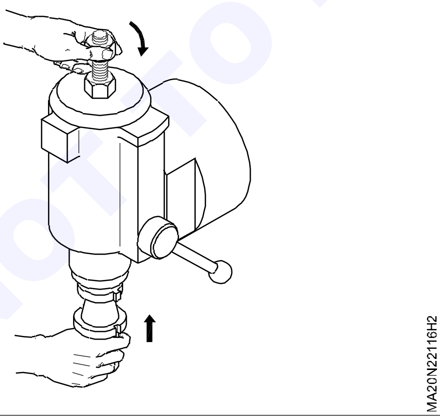
Fig 1



Ensure that the arbor notches fit in the keys of the spindle nose to get the drive.

Tighten the draw - bar and lock - nut. (Fig 2)

Fig 2



Remove the locking screw screw from the arbor end. (Fig 3)

Clean the arbor spigot, bore and contact faces of the cutter and notches.

Slide the cutter on the spigot such that the notches fit over the arbor keys. (Fig 4)

Fig 3

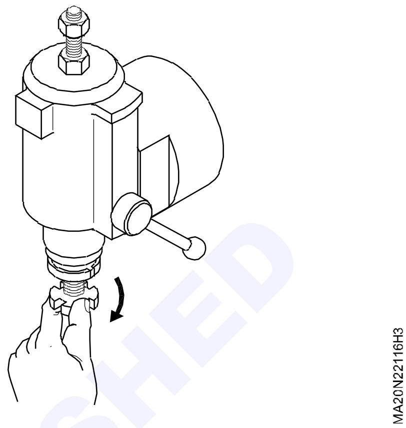
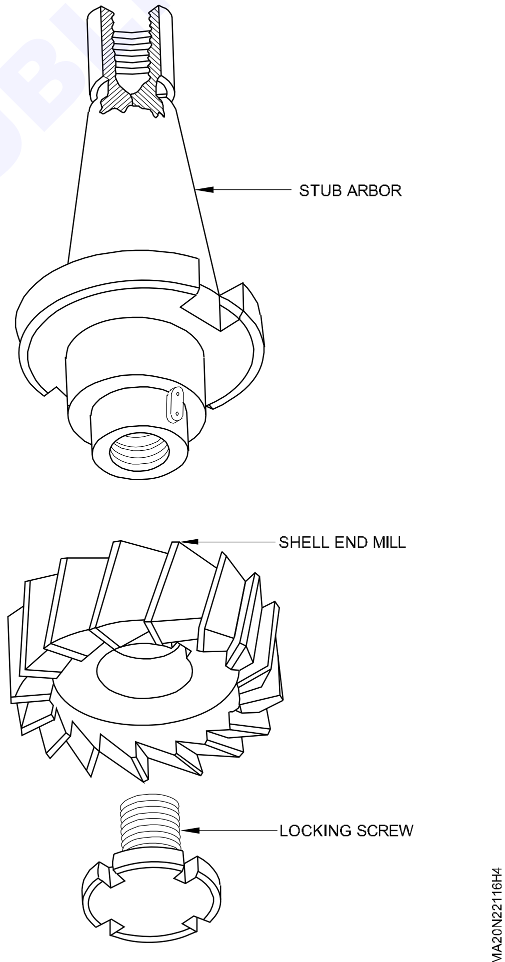


Fig 4



Use a rag while handling the cutter to avoid injury to the hands due to the sharp edges of the cutter.

Hold the cutter in position and tighten the locking screw using a pin spanner.

Avoid excessive tightening to prevent any damage.

Ensure that the end face of the locking screw is not projecting beyond the cutting edges of the cutter to avoid fouling with the work piece. This may happen after the cutter has been re - sharpened a number of times.

Switch on the machine and check visually that the cutter runs true.

Get famlliar thoroughly with the machine before attempting to operate it. When in doubt, obtain additional instructions/ guidance from the instructor.

Mill a flat surface on vertical milling machine

Objective: This shall help you to

- **mill a flat surface on a vertical milling machine.**

Mount and align the machine vice on the table of the vertical milling machine.

Hold the work piece in the machine vice.

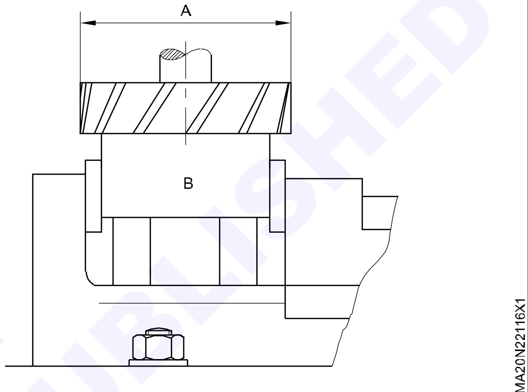
Mount the stub arbor in the spindle.

Mount the face milling cutter / shell end milling cutter into the stub arbor.

Select a H.S.S cutter with right hand helix for proper shearing of the material due to the positive rake angle.

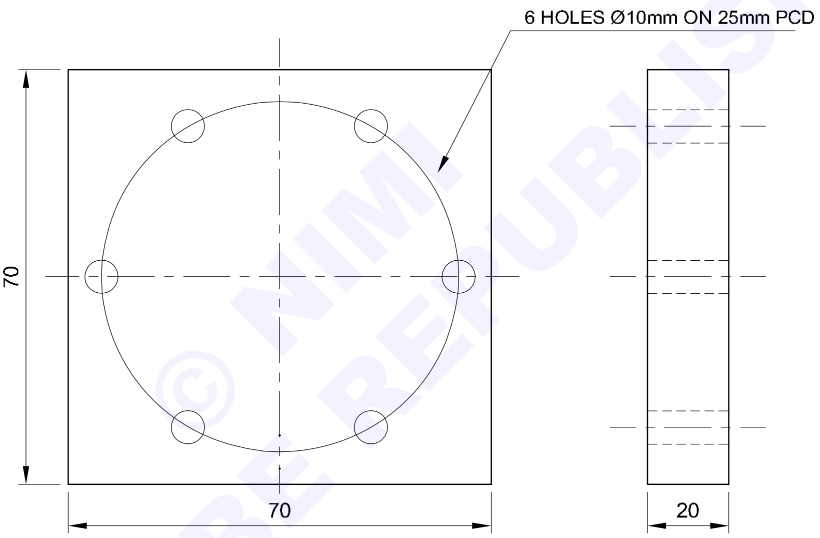
Select the diameter or the cutter (A) more than the width of the job (B), so that the full surface is machined in one pass. (Fig 1).

Fig 1



Drill on P.C.D. on milling machine with accuracy ± 0.02 mm

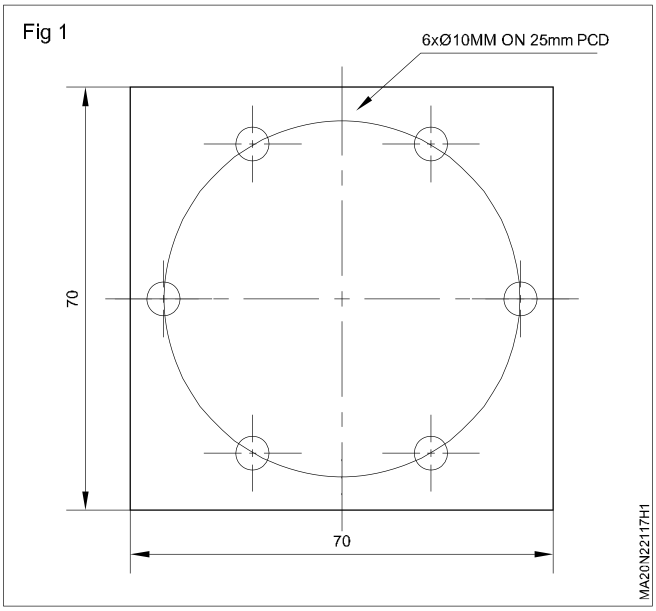
Objectives: At the end of this exercise you shall be able to
- **set rotary table on milling machine**
- **clamp job on rotary table**
- **drill on pitch circle diameter using rotary table.**

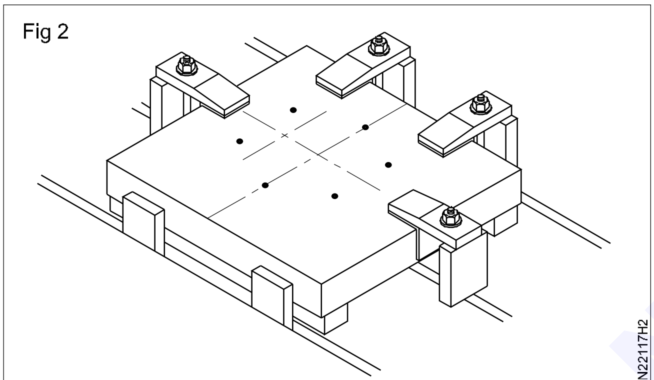


Job Sequence

- Mill the Job to the size of 70 x 70 x 20 mm.
- Mark the Job as per Drawing and punching witness marks. (Fig 1)
- Mount the rotary table on the vertical milling machine.
- Hold the Job on the rotary table using suitable work holding devices. (Fig 2)

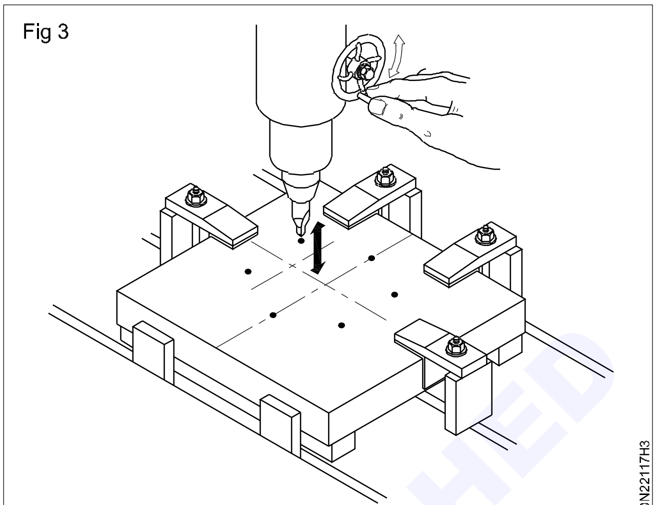
1	80x80x25	-	Fe310	-	-	2.2.117
NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.
SCALE NTS	PCD DRILLING ON MILLING MACHINE				DEVIATIONS ±0.02	TIME :
					CODE NO. MA20N22117E1	





WORKPIECE FITTED ON ROTATE TABLE

- Mount the drill in the vertical milling machine spindle.
- Set the drill to the centre of the Job.
- Off set the drill to the half of the pitch circle diameter. (Fig 3)



CENTRE DRILLING THE WORKPIECE

- Now set the number of holes to be drilled say it 6.

$$\text{Index} = \frac{360}{\text{No of divisions}} = \frac{360^\circ}{6} = 60^\circ$$

- Rotate the table for 60º and then drill the hole.
- Perform above move for five more times.
- So that a complete 360º and six holes are achieved.
- Six holes get drilled.
- Deburr the holes.
- Remove the attachments.

Skill Sequence

Circular table attachment

Objective: This shall help you to
- **drill holes on milling by using rotary table.**

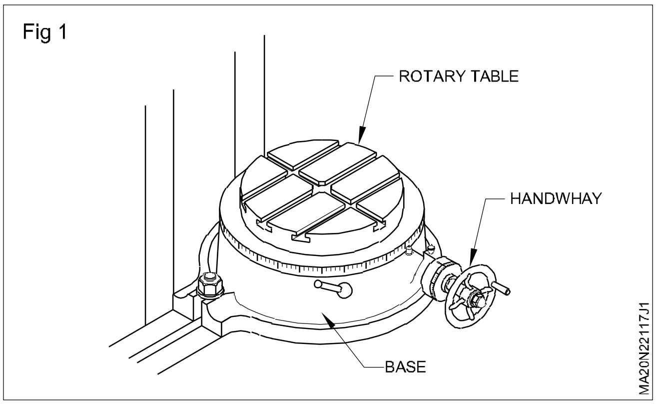
This attachment is used for profile milling, surfacing quantities of small pieces in the one set up, and for many other circular operations in the horizontal plane such as slotting and dove tailing.

It consists of a base, a worm gear drive mechanism and a small circular work table. (Fig 1)

The base is bolted to the table. A crank for manual feed is provided.

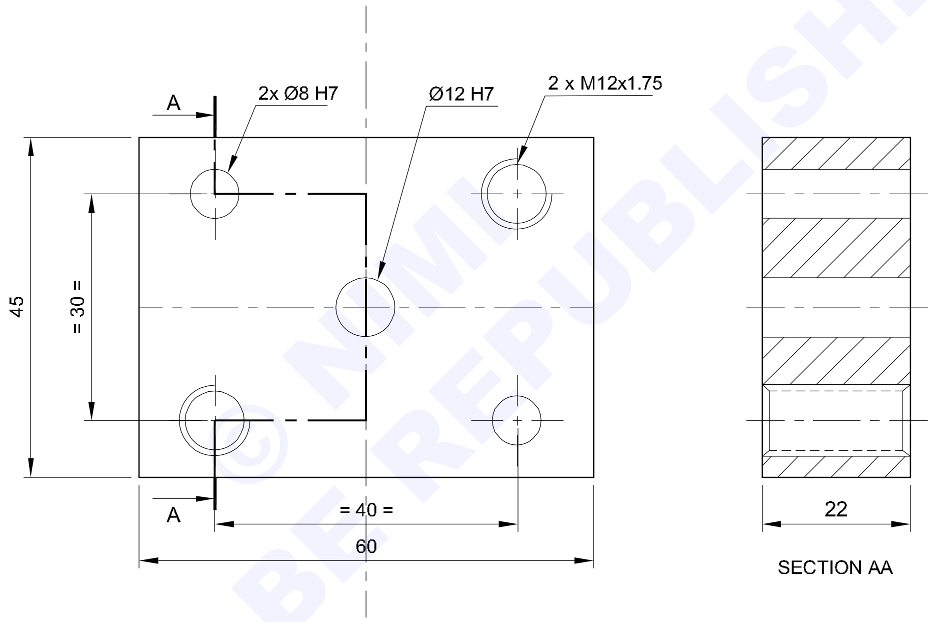
The work piece is secured to the circular table and the table is rotated by the crank.

By combining the rotary motion with one or more of the other movement of the machine, profiles of almost any shape can be milled. The hand crank can be replaced by an indexing device for requiring accurately spaced slots, holes or grooves.



Tapping and reaming operation using milling machine

Objectives: At the end of this exercise you shall be able to
- **set the job on the machine vice**
- **tap and ream using milling machine.**



NOTE:

Ø8H7 - 8.0 $^{+0.015}_{+0.000}$
Ø12h7 - 12.0 $^{+0.000}_{-0.018}$

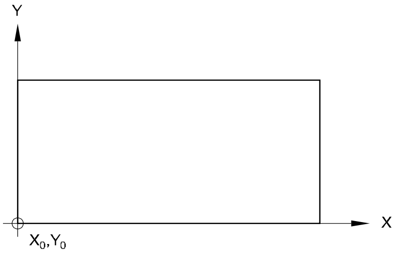
1	50x25-65	-	Fe310	-	-	2.2.118
NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.

SCALE 1:1	**TAPPING AND REAMING ON MILLING MACHINE**	DEVIATIONS ±0.02	TIME :
		CODE NO. MA20N22118E1	

Job Sequence

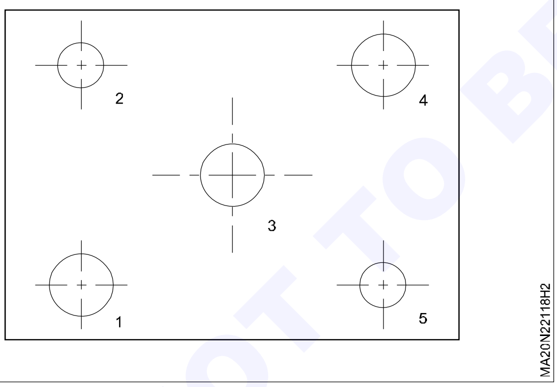
- Check the size of the raw material with steel rule
- Machine the block to size 60X45X22mm on vertical milling machine
- Mark the drill holes using vernier height gauge
- Punch hole centres with center punch
- Fix the machine vice on vertical milling machine table
- Align the vice jaw parallel to the column using dial test indicator.
- Fix the work piece in the vice with suitable parallel block.
- Located the zero point in x and y axes as shown in Fig 1.

Fig 1



- Hold the drill chuck on the machine spindle using collet chuck.
- Fix center drill.
- Calculate the x and y axes movement for 5 hole location from the origin, as shown in Fig 2.

Fig 2



- Move the x and y axes to the calculated values and lock the table movement in x and y axes to position 1.
- Drill center drill, drill, ream and tap. Similarly move the table on other location drill center drill, drill, ream and tap.
- Remove the work piece ,clean deburr and apply oil and preserve it for evaluation.

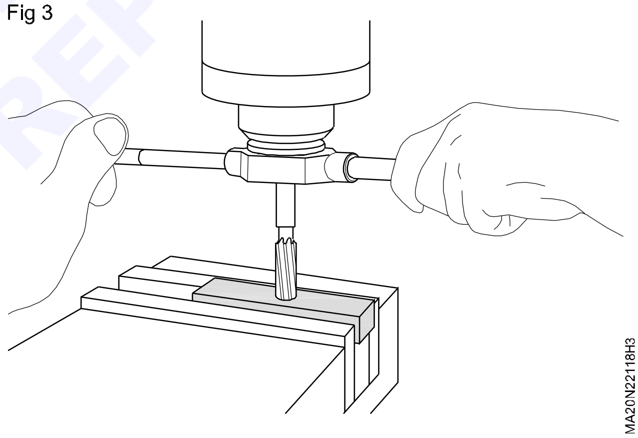
Reaming

- Check the size of the pre machined work piece
- Mark the holes centers as per drawing
- Punch mark the hole center with center punch
- Set the align the machine vice jaw parallel to the machine column
- Fix the work piece on the vice with suitable parallel blocks without obstruction
- Fix the center drill in drill chuck
- Set the RPM
- Align the hole center with spindle center
- Lock the table movement in longitudinal and transverse movement that is x and y axes and set zero on both the axes

Don't disturb hole location

- Put on the spindle
- Drill the center drill to the required depth
- Drill pilot drill ∅a 6mm
- Drill ∅11.75mm
- Fix the machine reamer ∅12 mm on the spindle
- Set 1/3 of the RPM for drilling ∅12mm
- Put ON the spindle
- Raise the table towards the reamer until it just touch the work surface. (Fig 3)

Fig 3



- Engage the upward auto feed or slowly feed the the spindle down wards until; the reamers just clear the bottom of the hole.
- Stop the spindle rotation.
- Move the table downwards or spindle upward.
- Unlock the table movement.

Taping

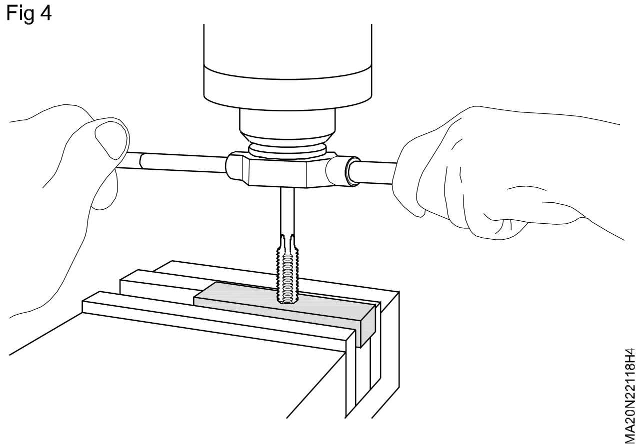
- Position the tap hole centre by moving X and Y axes to the required distance as in the job drawing.

Don't disturb hole location

- Lock the table movement

- Drill center drill, pilot drill and tap drill for M12.
- Fix the dead center or spring center on the spindle.
- Place the tap with tap wrench as shown in Fig 4.
- Apply pressure on tap wrench while rotating the wrench in clockwise direction.
- After 2 or 3 complete rotation , rotation the tap wrench in opposite direction about ¼ turn to break the metal chips.
- Follow the same steps until you complete the tapping to the full length of the hole.
- Remove the tap from the work piece by rotating the tap in counter clockwise direction.
- Similarly position and tap the other tap hole as per drawing.

Fig 4



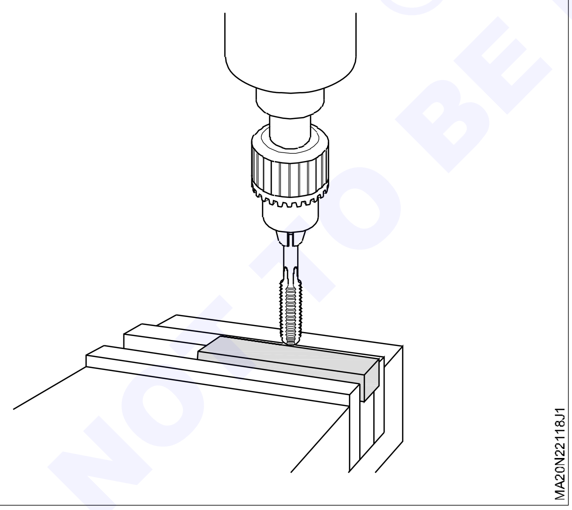
Skill Sequence

Rigid Tapping

Objective: This shall help you to
- **perform rigid tapping on milling machine.**

Rigid tapping is the second most common method of tapping on the mill. With this technique the tap is clamped in the spindle and threaded into the workpiece under spindle power. Smaller taps up to 3/8" can be clamped in a key less Jacobs-style chuck as shown in Fig 1. Larger taps should be clamped using a split sleeve tap driver or a collet chuck.

Fig 1



The process for rigid tapping using a mill is as follows:

1 Clamp the part securely, ensuring adequate room beneath if tapping through holes.

2 Select an appropriate tap. Spiral point (i.e. "gun") taps.

3 Load the tap into a Jacobs-style chuck, or for larger taps into a split sleeve tap driver or collet chuck.

4 Make sure the spring loaded spindle lock (the micrometer stop) is at its lowest position so it cannot engage the quill depth stop.

Place the mill in low range, the slower the speed the better for rigid tap.

6 Apply cutting oil to the tap.

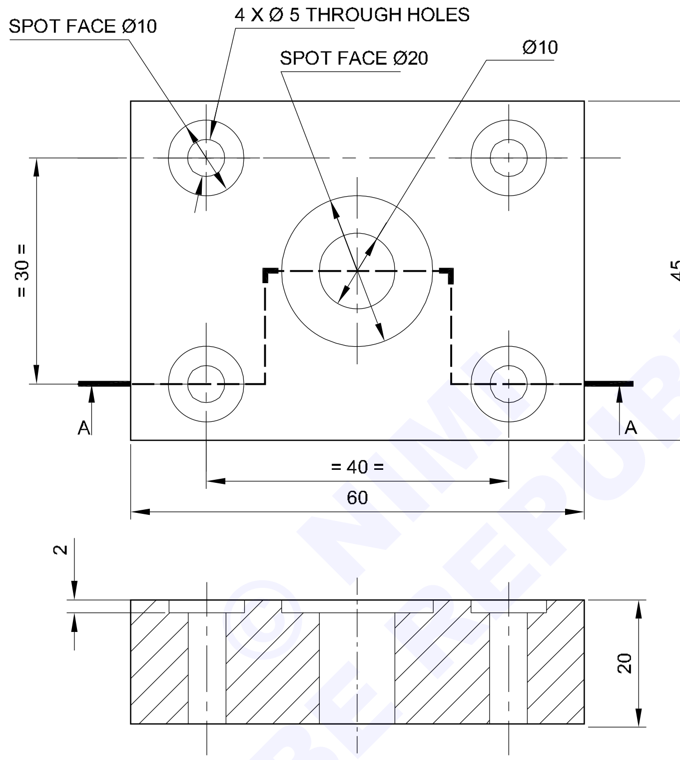
7 Turn on the spindle in the forward direction and firmly bring the tap into engagement with the part. Keep your hand on the power switch.

8 After 4 or 5 threads, turn the spindle off and to the reverse direction to break the chip or withdraw the tap from the hole. You can switch directions very quickly if necessary to prevent over-threading.

9 Be cautious to not run the tap past the location where the threads end (left), unless using a reduced shank tap (right).

Perform spot facing operation using milling machine with accuracy ± 0.02 mm

Objectives: At the end of this exercise you shall be able to
- **mill the work piece to the given size**
- **mark and drill holes using vertical milling machine**
- **spot face hole using slot / end drill.**



Job Sequence

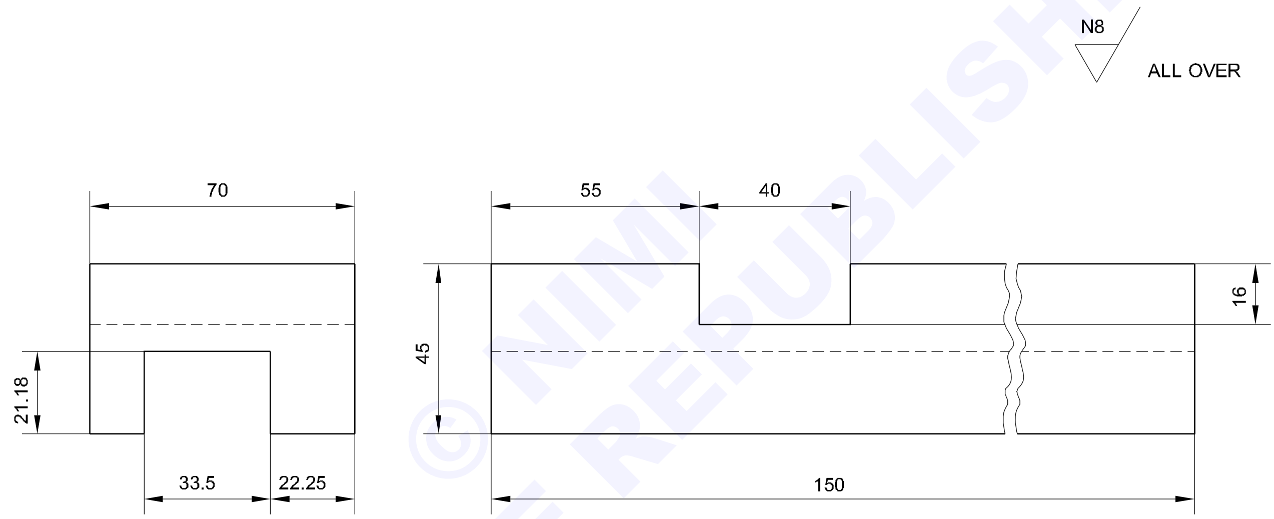
- Mark the work piece as per drawing to drill hole and spot face.
- Clamp the work piece on machine vice jaw parallel to the vertical column.
- Mount the collet in vertical milling machine spindle. Align the hole location to the spindle center drill and as per marking.
- Mount slot drill in collet chuck.
- Set required RPM
- Raise the table to touch the cutting edge just touch the work surface
- Set the vertical movement dial to zero
- Feed the table towards the slot drill to spot face the required depth.

01	50 x 25-65	-	Fe310	-	-	2.2.119
NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.

SCALE 1:1	SPOT FACING OPERATION	DEVIATIONS	TIME :
		CODE NO. MA20N22119E1	

Make a slot on face of the Job using milling machine with an accuracy ± 0.02 mm

Objectives: At the end of this exercise you shall be able to
- **set the Job on milling machine**
- **hold the cutter on machine spindle**
- **mill the slots by selecting proper speed and feed.**



NOTE:

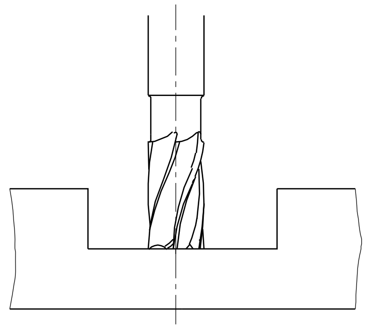
N8 - SMOOTH MACHINING Ra-1.6

1	155x50x75MM		CAST IRON	23	1	2.2.120
NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.
SCALE 2:1	SLOT MILLING				DEVIATIONS ±0.02	TIME
					CODE NO. MA20N22120E1	

Job Sequence

- Mill the job to the size of 150 x 45 x 70 mm.
- Mark the slots to size as per drawing and punch witness marks.
- Mount and align the vice on the vertical milling machine table.
- Mount ϕ22 mm end mill cutter for milling in the vertical milling machine.
- Set the r.p.m. closer to 300.
- Mill the slot 33.5 x 21.18 x 150 mm long to size. (Fig 1)
- Deburr and check the dimensions of the slot with a depth micrometer and vernier caliper.

Fig 1



- Reset and align the job for the next slot of the job.
- Mill the slot 40 x 16 x 70 mm long to size.
- Deburr and check the dimensions of the slots for their size.

Skill Sequence

Mill a slot by end mill cutter on vertical milling machine

Objective: This shall help you to
- **mill a slot on a vertical milling machine.**

Mark the job as per drawing for slot milling and punch the witness marks on the lines.

Mount the plain machine vice on the machine such that the vice jaws are parallel to the column.

Clamp the workpiece in the machine vice such that the face is 5 to 6 mm above the vice jaws.

Select an end mill cutter which is smaller than the width of the slot to be milled. This is only for the manipulation of the cutter in the slot while milling.

Mount the end mill cutter in the collet chuck.

Ensure that the end mill cutter is gripped without any wobbling to avoid breakage of the cutter or vibration during the cutting operation.

Set the r.p.m., table feed and clockwise cutter rotation.

Set the nearest lower speed and feed available if the machine does not have exact values.

Stick tissue paper to the reference sides of the workpiece and set the datum for the cross and vertical slides.

Set the cutter at the middle of the slot to (X + W/2) mm taking reference from the side datum where X is marked in the drawing. W is the width of the slot. (Fig 1)

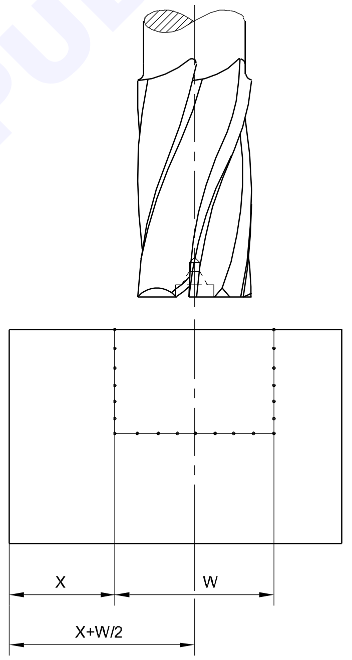
Set the depth of cut taking reference from the top surface of the job. (Fig 2)

Be sure that whenever depth of cut or setting is carried out the cutter is away from the job.

Consider the following hints while setting the depth of cut.

The maximum depth of cut that can be applied is equal to the diameter of the end mill.

Fig 1



If the diameter of the end mill is between 2 and 6 mm apply a small depth of cut (e.g. 0.4 to 0.5 mm).

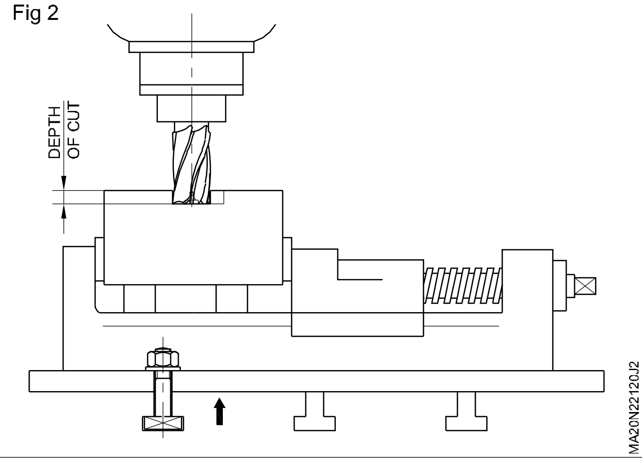
Keep 0.4 to 0.5 mm on both the sides and depth of the slot for final finish.

Lock the vertical and cross-slides.

Adjust the coolant nozzle point on the cutter.

Start the spindle and the coolant pump.

Fig 2

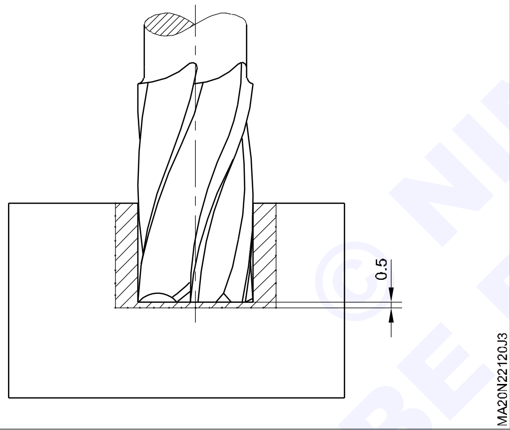


Move the longitudinal slide manually towards the cutter such that the workpiece comes into contact with the cutter gently.

Sudden contact may damage the workpiece and the cutter may break.

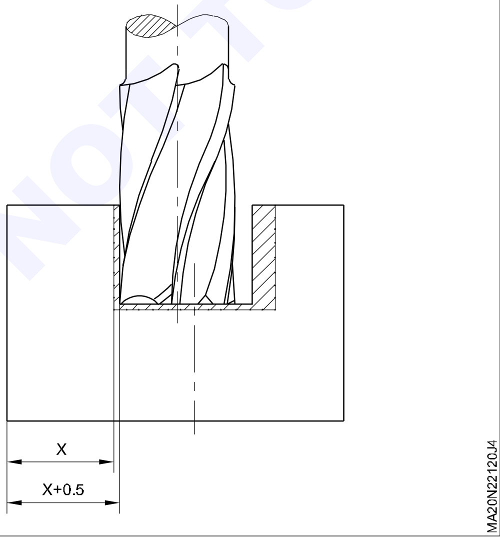
Mill the depth at the middle of the slot leaving 0.4 to 0.5 mm for the final finish. (Fig 3)

Fig 3



Move the cutter to the left to a diameter of X + 0.5 mm and mill the side of the slot. (Fig 4)

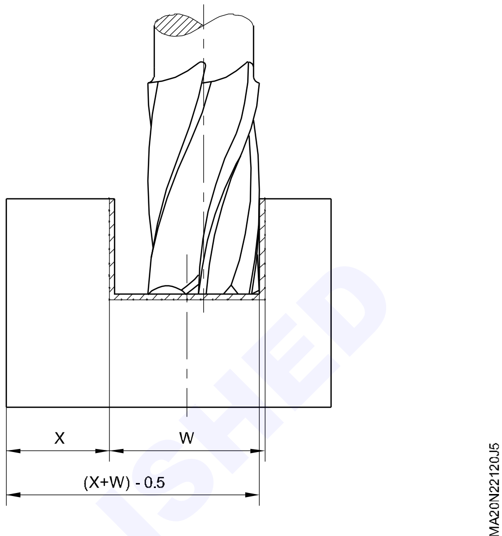
Fig 4



Ensure that the datum is set again whenever the cutter is changed or the cutter overhang is adjusted or the job disturbed.

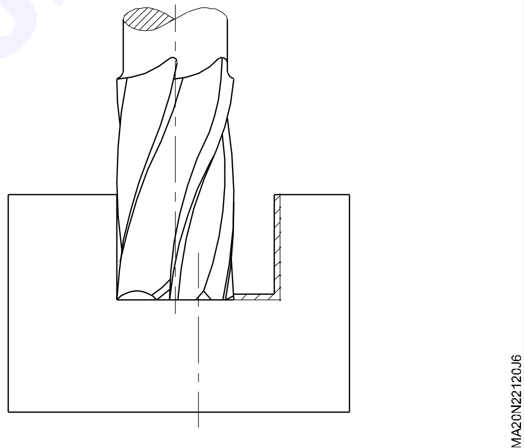
Move the cutter to the right to a distance of (X + W) - 0.5 mm and mill the right slide of the slot. (Fig 5)

Fig 5



Follow the above procedure and mill the slot to size. (Fig 6)

Fig 6



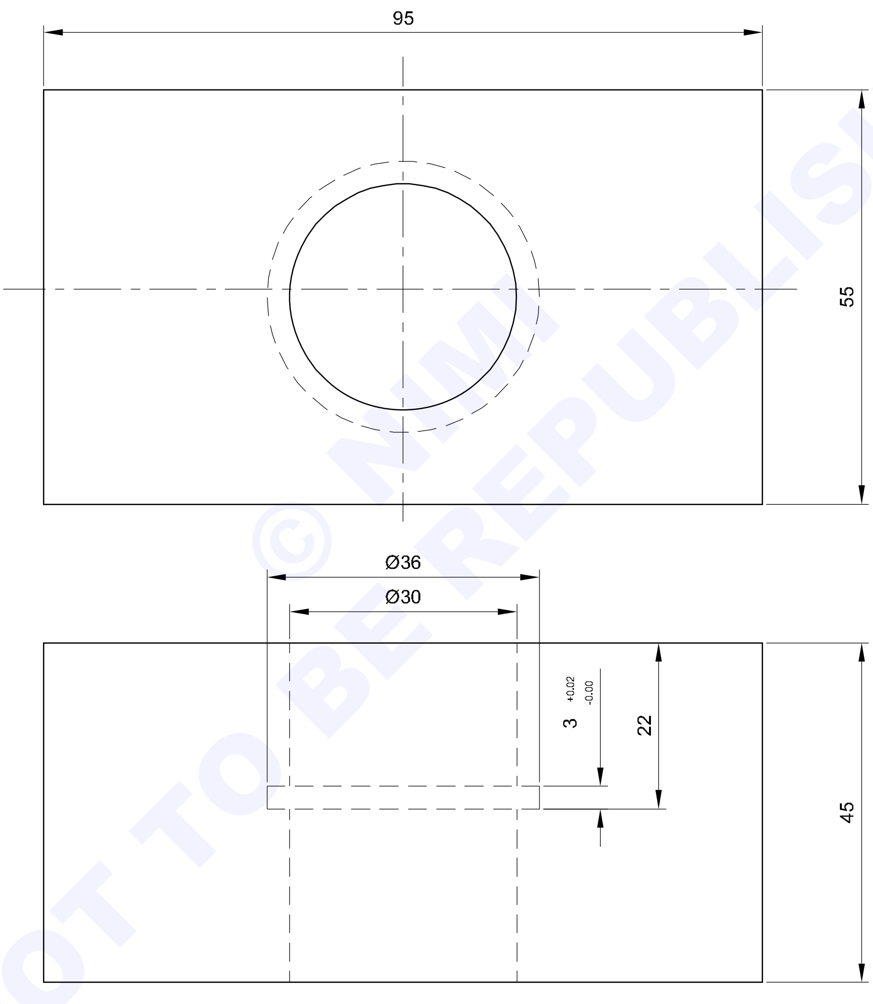
Deburr the job.

Check the slot for dimension.

Do not talk to anyone while operating the machine, not allow anyone to turn on your machine for you.

Internal groove milling with an accuracy of ± 0.02 mm

Objectives: At the end of this exercise you shall be able to
- **mill the work piece to the required size**
- **set the rotary table on vertical milling machine**
- **clamp and set the work piece rotary table**
- **align the hole center and spindle center and rotary table center in one line, bore and cut internal groove using boring head.**



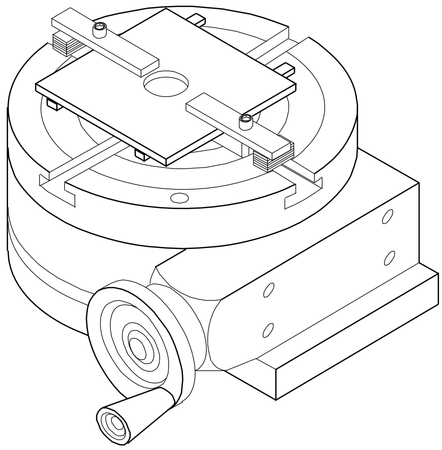
01	C.I BLOCK 60X50X100	-	C.I BLOCK	-	-	2.2.121
NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.

SCALE 1:1	**INTERNAL GROOVING**	DEVIATIONS	TIME :
		CODE NO. MA20N22121E1	

Job Sequence

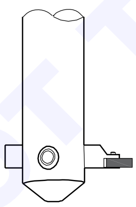
- Check the size of the raw material with steel rule.
- Mill the block to 55X45X95mm maintaining all the surface mutually perpendicular and parallel.
- Mark the center portion of hole center as shown in drawing.
- Clamp the rotary table on vertical milling machine table and set the work piece on rotary table with suitable parallel blocks. (Fig 1)

Fig 1



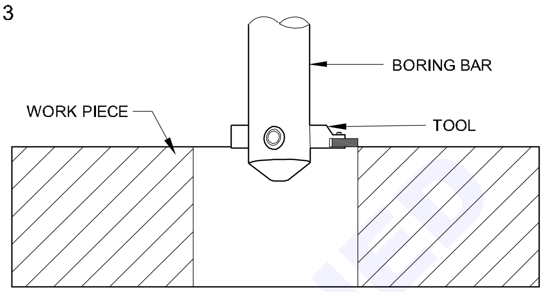
- Align the hole center to the spindle center in one line
- Lock the movement X and Y, set zero on X and Y movement
- Drill hole using slot drill or double fluted drill to 20.00 mm diameter.
- Enlarge the hole to diameter using boring head.
- Grind the grooving tool and maintain the width of the tool to 3.00 ± 0.02 mm.
- Select 10mm straight bring bar
- Fix the tool in the straight boring bar as shown in Fig 2.

Fig 2



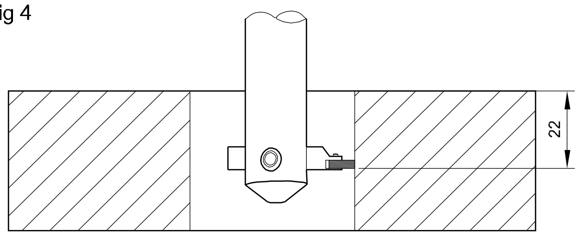
- Fix the boring bar in the head
- Adjust the tool tip projecting 4 mm away from the boring bar
- Select the suitable spindle speed
- Raise and move the table only in X direction, such that tool tip just touch on top of the work piece. (Fig 3)

Fig 3



- Set zero on the vertical screw dial bring back the x axis movement to zero position
- Raise the table to 22mm as shown in Fig 4
- Put on the spindle slowly move the x movement when it touches the inside bore, set zero on x movement
- Move 1 mm in to the bore, and give the feed by rotating the rotary table
- Continue rotating (360°) till its form the full circle
- Similarly give depth in x movement and feed in rotary table until 3mm depth is reached
- With draw the tool to the hole center
- Put off the spindle
- Lower the table (vertical movement)
- Remove and clean the machine and work piece.

Fig 4



Skill Sequence

Center the rotary table with the vertical mill spindle

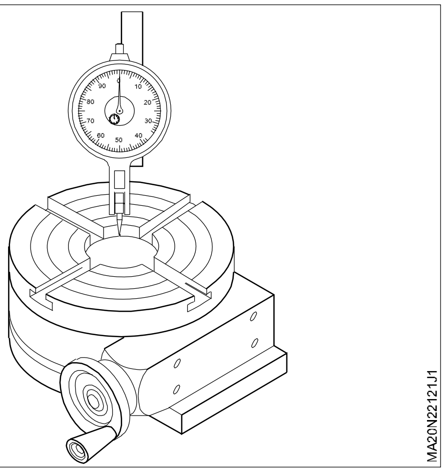
Objectives: This shall help you to
- **mount and centre the rotary table with milling machine spindle**
- **centre the work piece with rotary table.**

Center the rotary with vertical mill spindle

- Set the vertical head perpendicular to the machine table
- Mount the rotary table on the milling machine table.
- Mount a dial indicator in the milling spindle.

- Bring the dial indicator into contact with the bore diameter of rotary table (Fig 1)

Fig 1



In the Fig 1 it looks like tip of the indicator is hanging in space , but it is actually touching the back of the hole in the rotary table. Turn the table through 360 degrees of rotation watch for the maximum deflection on the indicator. Then rotate the spindle 90 degrees to the left and 90 degrees to the right. The true center will be half way between the two readings.

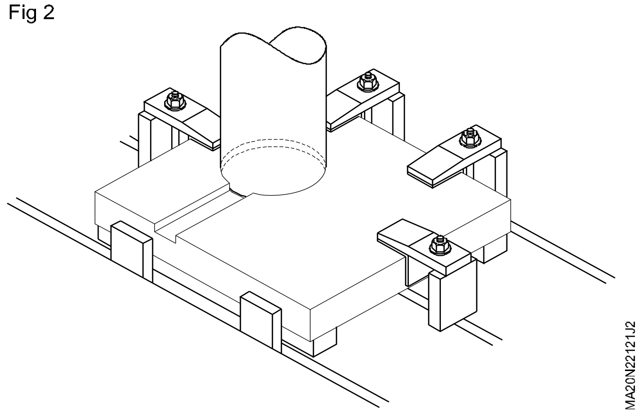
- Adjust the machine table by the longitudinal (X) and cross feed (Y) handles until the dial indicator register no movement.
- Lock the milling machine table and saddle ,and recheck the alignment .
- Readjust if necessary

Center a work piece with rotary table

To quickly align each work piece , a special plug can be made to the fit the center hole of the work piece and the hole in the rotary table. Once the machine spindle has been aligned with the rotary table, each succeeding piece can be aligned quickly and accurately by placing it over the plug (Fig 2)

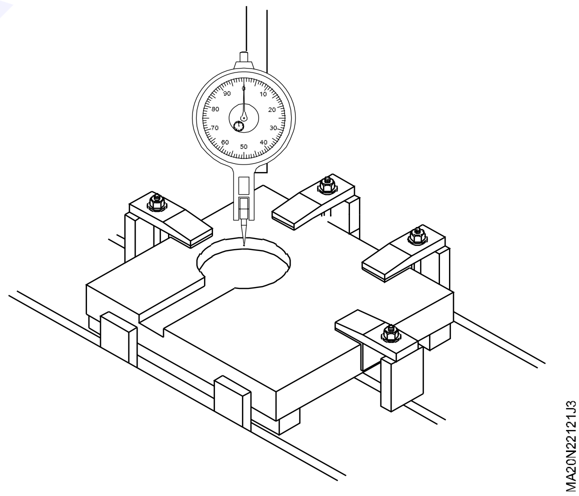
If there are only a few pieces , which would not justify the manufacture of a special plug. Or if the work piece does not have a hole through it center , the following method can be used to center the work piece on the rotary table.

Fig 2



- Align the rotary table with the vertical mill head spindle.
- Lightly clamp the work piece on the rotary table in the center.do not move the longitudinal (X) or cross feed (Y) feed handles.
- Disengage the rotary table worm mechanism .
- Mount a dial indicator in the milling machine spindle or milling machine table, depending upon the work piece. (Fig 3)
- Bring the dial indicator into contact with the surface to be indicated, and revolve the rotary table by hand.
- With a soft metal bar, tap the work piece (away from the indicator movement) until no movement is registered on the indicator in a complete revolution of the rotary table.
- Clamp the work piece tightly, and recheck the accuracy of the setup.

Fig 3



Boring on vertical milling machine

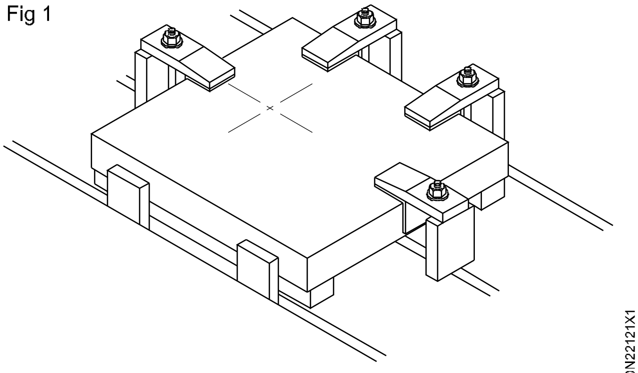
Objectives: This shall help you to
- **drill hole at a particular location**
- **bore hole using boring head.**

Mount and align the machine vice on the table of the vertical milling machine and clamp the workpiece.

If the job is large enough, it can be clamped directly on the machine table using clamps. (Fig 1)

Note that the work-holding device may be clamps, stops, nut, bolts, etc. depending upon the shape and size of the workpiece to be bored.

Fig 1

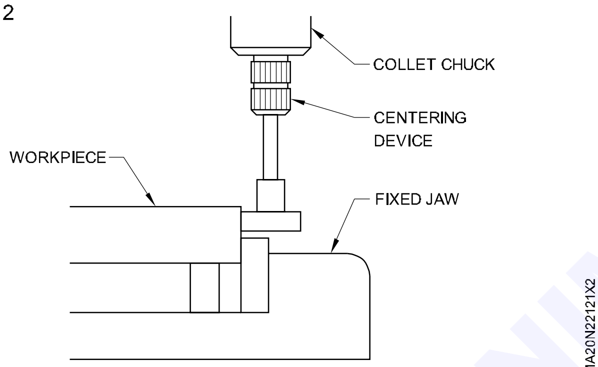


WORKPIECE FITTED TO MACHINE TABLE

Set up a centering device in the chuck in the vertical spindle, touch one edge of the workpiece with the centering device. (Fig 2)

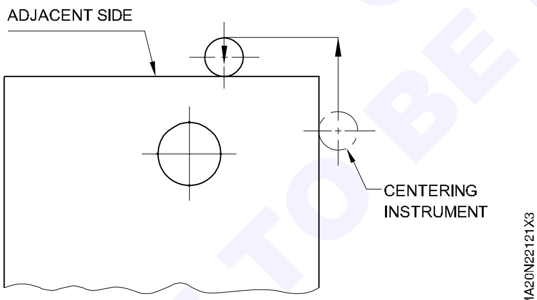
Set the cross-slide dial at zero.

Fig 2



Touch the adjacent side of the workpiece with the centering device.(Fig 3)

Fig 3



Set the longitudinal slide scale at zero.

Wind down the machine table and clear the centering device.

Remove the device and the collet chuck.

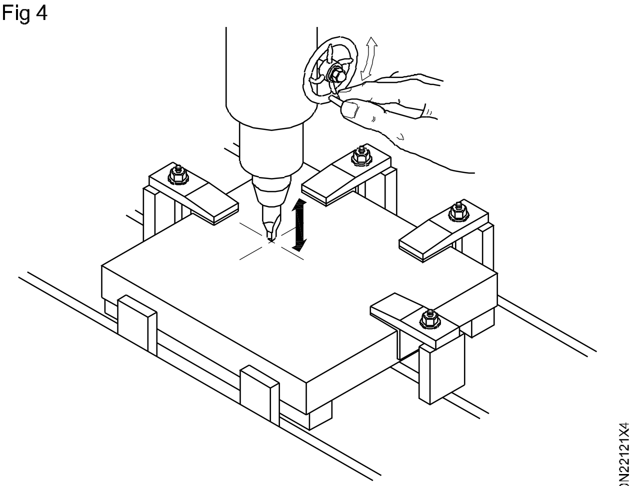
Lock the longitudinal and cross slides.

Set the spindle speed and power feed to suit the material.

Centre drill and drill hole of ∅ 20mm in steps. (Fig 4)

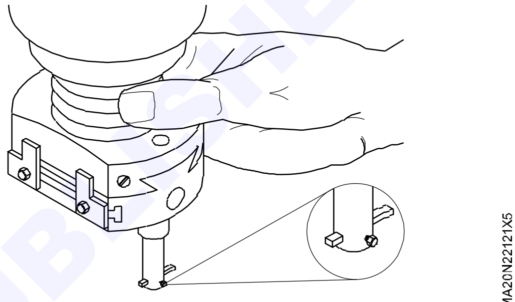
Wipe the taper on the boring head and spindle. Insert the boring head into the spindle and tighten. (Fig 5)

Fig 4



CENTRE DRILLING THE WORKPIECE

Fig 5



Select the boring bar long enough to bore to full depth.

Mount it in the head and tighten firmly.

Set the work piece centre coinside with spindle centre.

Beware of back-lash in the slides movements.

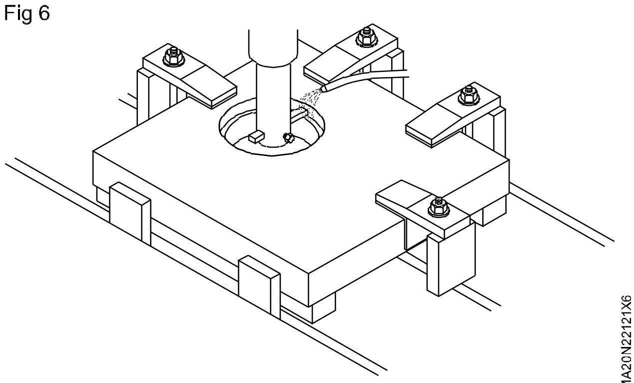
Fix the finishing tool in the boring bar.

Off set the boring bar, such that the cutting tool just touches the inner surface of the bore and lower the table.

Give 0.8mm cut and set speed and feed.

Start the machine. Engage vertical power feeding movement and bore it. Apply sufficient coolant. (Fig 6)

Fig 6



At the end of the cut stop the machine

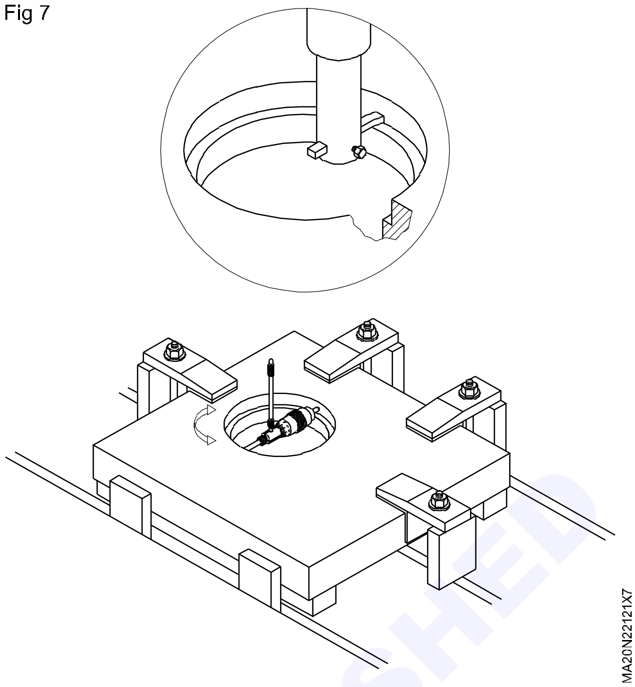
Lower the table, check the bore dia with inside micrometer. (Fig 7)

Adjust the cut for finishing and finish the bore.

Check the final bore diameter using an inside micrometer and the bore location using a standard plug/pin and depth micrometer.

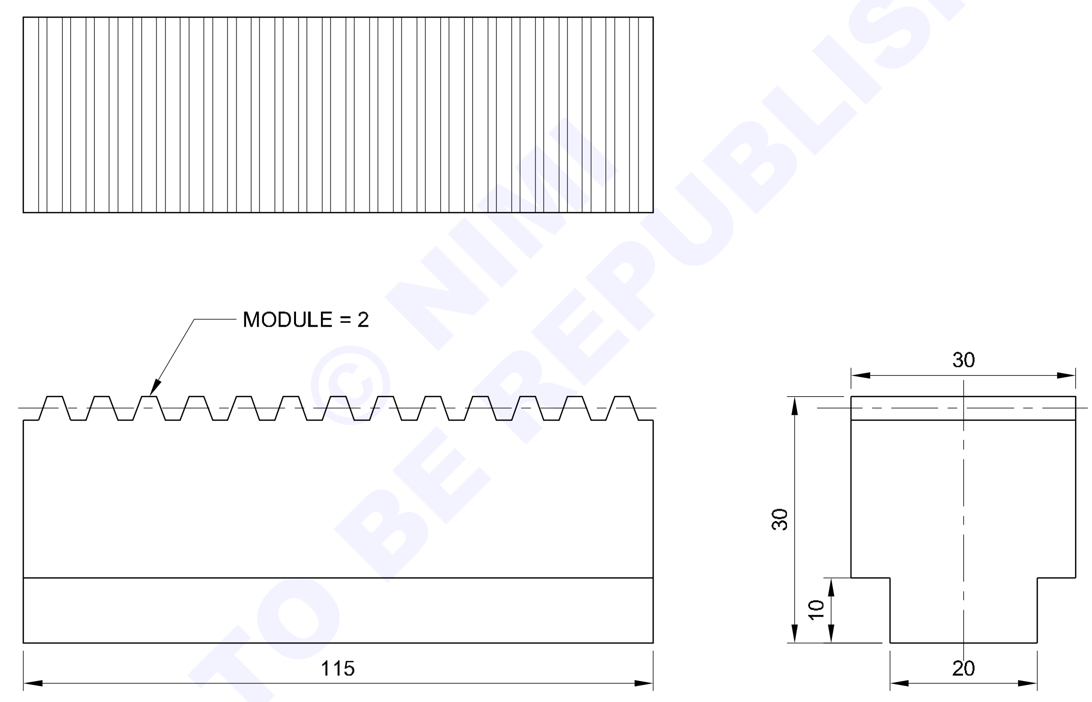
If boring head is not available, use boring bar with collet chuck.

Fig 7



Make straight teeth rack using milling machine with an accuracy 0.05 mm

Objectives: At the end of this exercise you shall be able to
- **mill the rack blank as per drawing**
- **calculate the rack proportions**
- **set the cutter in the arbor**
- **prepare the milling machine for linear indexing**
- **mill the rack using linear indexing.**



1	ISSQ 32-120	-	Fe310	-	-	2.2.122
NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.

SCALE 1:1	STRAIGHT TEETH RACK MILLING	DEVIATIONS ±0.05	TIME
		CODE NO. MA20N22122E1	

Job Sequence

Rack cutting

- Calculate the required rack proportions for a 2 module straight tooth rack.

Whole depth = 2.25 x m
= 2.25 x 2 = 4.5 mm

Linear pitch = π x m
= 3.14 x 2
= 6.28 mm.

Indexing calculation

$$\text{Indexing} = \frac{\text{Linear pitch of rack}}{\text{Pitch of table lead screw} \times \frac{1}{40}} \times \text{Gear ratio}$$

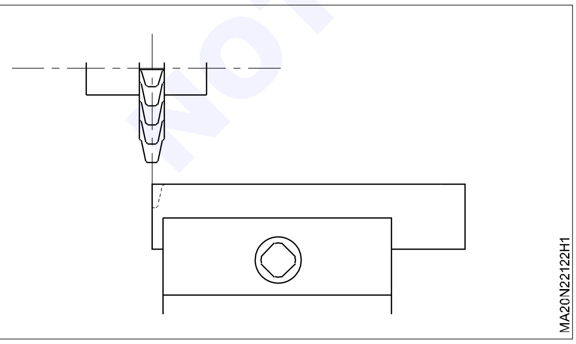
$$= \frac{6.28 \times 40}{5} \times \frac{1}{3}$$

= 16.746 rounded off to 16.75

$$\text{Gear ratio} = \frac{1}{3} = \frac{24}{72}$$

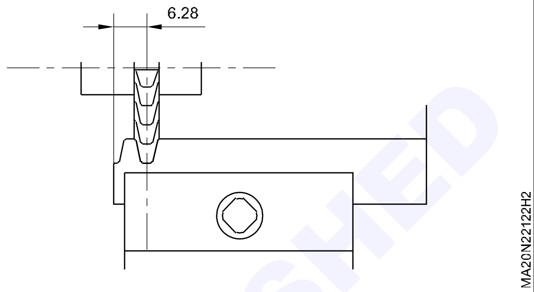
- Mill the rack blank to the given dimension as per the drawing.
- Clean the machine table and mount the machine vice keeping the jaws perpendicular to the machine column.
- Align it with dial test indicator.
- Clamp the job such that it projects 10 mm above the jaw surface.
- Mount the 2 module rack cutter at the middle of the arbor.
- Bring the job under the cutter such that the centre of the cutter form is aligned to the edge of the job. (Fig 1)

Fig 1



- Eliminate the backlash, adjust the graduated collar of the cross-slide and the elevating screw to zero.
- Start the machine and mill the initial portion of the rack to a depth of 4.50 mm in two steps; 3 mm for the roughing cut and 1.5 mm for the finishing cut.
- Clear off the job from the cutter and stop the machine.
- Slide the job using a cross-slide to a pitch distance of 6.28 mm and reset the collar again to zero. (Fig 2)

Fig 2



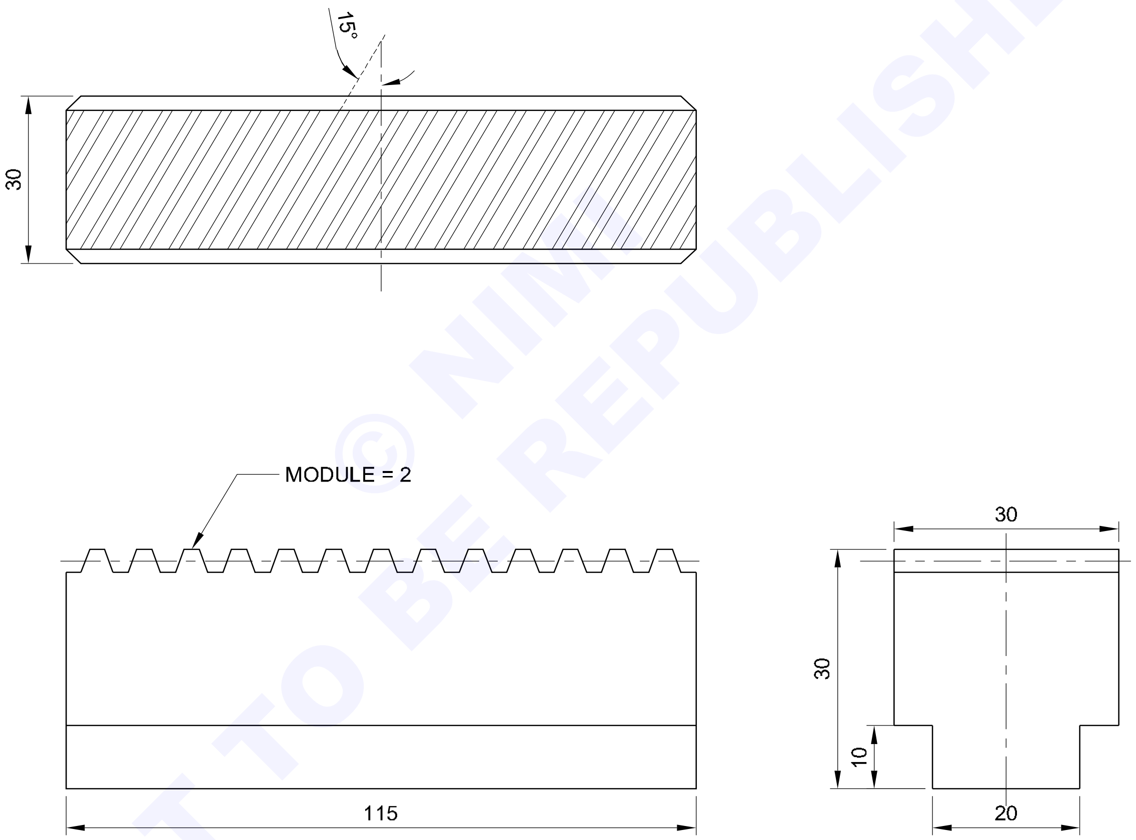
- Start the machine and mill the next tooth.
- Repeat the above sequence and mill the remaining rack teeth over the entire length of the job.

Rack milling using rack milling attachment

- Calculate the required rack proportions of a 2 module of straight tooth rack.
- Mount the machine vice parallel to column face and mount the job.
- Mount the rack cutting attachment.
- Prepare the milling machine for linear indexing with driver 72 teeth (index head spindle), driven 24 teeth (lead screw) and 20 hole circle plate.
- Mount the cutter number 1 on to the spindle of the rack cutting attachment.
- Position the job under the cutter for the first tooth space cutting. (The direction of table movement will depend on the number of idlers for the same rotation of crank.)
- Give the depth of cut to 4.5 mm.
- Mill the first tooth space.
- Index for the next tooth space.
- Index the crank movement = 16 full turns and 15 hole in a 20 hole circle plate.
- Cut the second tooth space.
- Check the thickness of the tooth 3.14 mm, using a gear tooth vernier caliper.
- Repeat the process and finish the rack.

Make helical teeth rack using milling machine with an accuracy 0.05 mm

Objectives: At the end of this exercise you shall be able to
- **mill the rack blank as per drawing**
- **calculate the rack proportions**
- **set the cutter in the arbor**
- **prepare the milling machine for linear indexing**
- **mill the rack using linear indexing.**



1	ISSQ 32-120	-	Fe310	-	-	2.2.123
NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.

SCALE 1:1	**HELICAL TEETH RACK MILLING**	DEVIATIONS ±0.05	TIME
		CODE NO. MA20N22123E1	

Job Sequence

Rack cutting

- Calculate the required rack proportions for a 2 module helical tooth rack.

  Whole depth = nm X 2.25 (nm - normal module)

nm = m x cos α (module = m)

= 2 x cos 15°

= 2 x 0.9659 = 1.93

Whole depth

= 1.93 x 2.25

= 4.34 mm

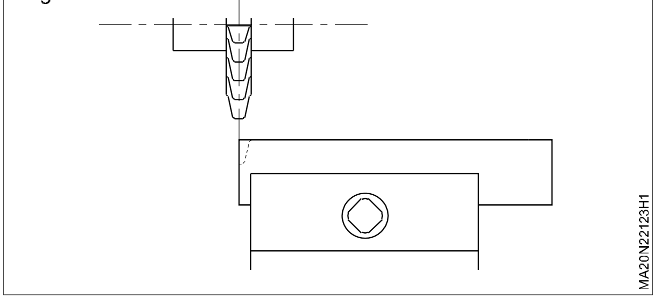
Linear pitch

= π x nm

= 3. 14 x 1.93 = 6mm

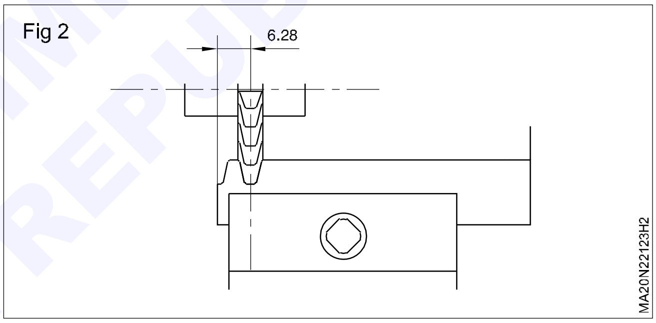
- Clean the machine table and mount the swivel machine vice keeping the jaws perpendicular to the machine column.
- Align it with dial test indicator.
- Swivel the vice 15° with respect to column.
- Clamp the job such that it projects 10 mm above the jaw surface.
- Mount the 2 module rack cutter at the middle of the arbor.
- Bring the job under the cutter such that the centre of the cutter form is aligned to the edge of the job. (Fig 1)

Fig 1



- Eliminate the backlash, adjust the graduated collar of the cross-slide and the elevating screw to zero.
- Start the machine and mill the initial portion of the rack to a depth of 4.34 mm in two steps; 3 mm for the roughing cut and 1.34 mm for the finishing cut.
- Clear off the job from the cutter and stop the machine.
- Slide the job using a cross-slide to a pitch distance of 6.00 mm and reset the collar again to zero. (Fig 2)
- Start the machine and mill the next tooth.
- Repeat the above sequence and mill the remaining rack teeth over the entire length of the job.

Fig 2



Skill Sequence

Cutting rack teeth on universal milling machine using graduated dial

Objective: This shall help you to
- **prepare machine for rack milling and cut rack teeth using graduated collar of cross-feed traverse.**

If length of rack to be cut is short (ie. less than the max. travel of cross feed) it can be done on normal horizontal milling machine.

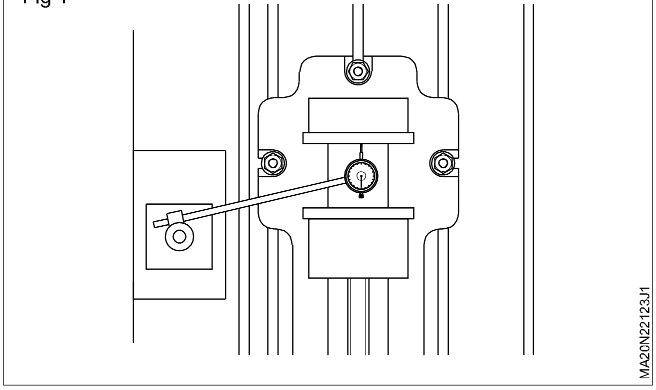
If it is more than the max. traverse of cross feed traverse it should be done using a rack milling attachment.

Mill rack on horizontal milling machine

Clamp the vice on the table in such a way that its jaws are parallel to the spindle axis, and align the fixed jaw with dial test indicator. (Fig 1)

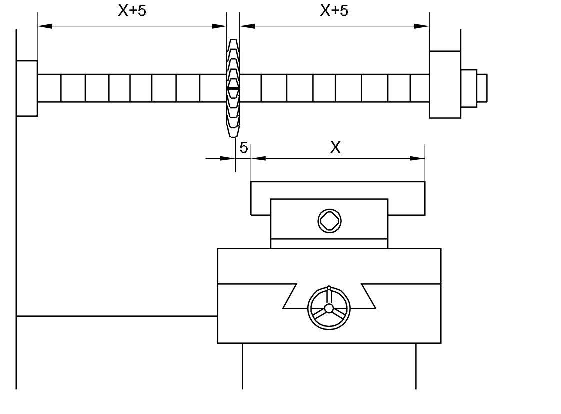
Move the table far away from the column.

Fig 1



Mount the rack cutter (cutter No.1) in the arbor in such a way that there is a 5 to 10 mm. gap between the job and the cutter. (Fig 2)

Fig 2

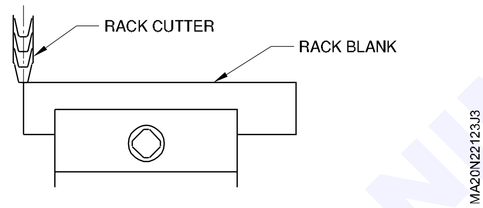


Set the datum (ie '0') on graduated dials of the vertical traverse and cross traverse, when the cutter is touching the job on the top and at the edge. (Fig 3)

Apply depth of cut raising the knee.

Total depth of cut = 2.25 x m.

Fig 3



Cut a groove by feeding the job against the cutter horizontally as per pitch and depth. (Fig 4)

Move the cross-slide using a graduated dial by a distance, P = π x module, towards the column and cut the next groove.

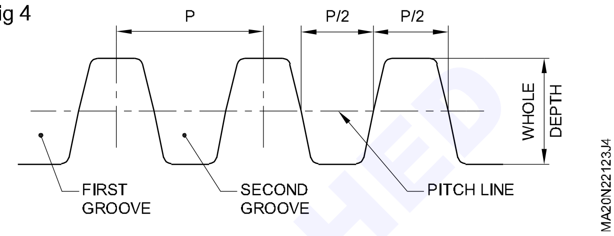
Deburr the tooth and measure the tooth thickness on the pitch line using a gear tooth caliper.

Thickness = 1.5708 x m

Make necessary correction in the depth of cut, if necessary.

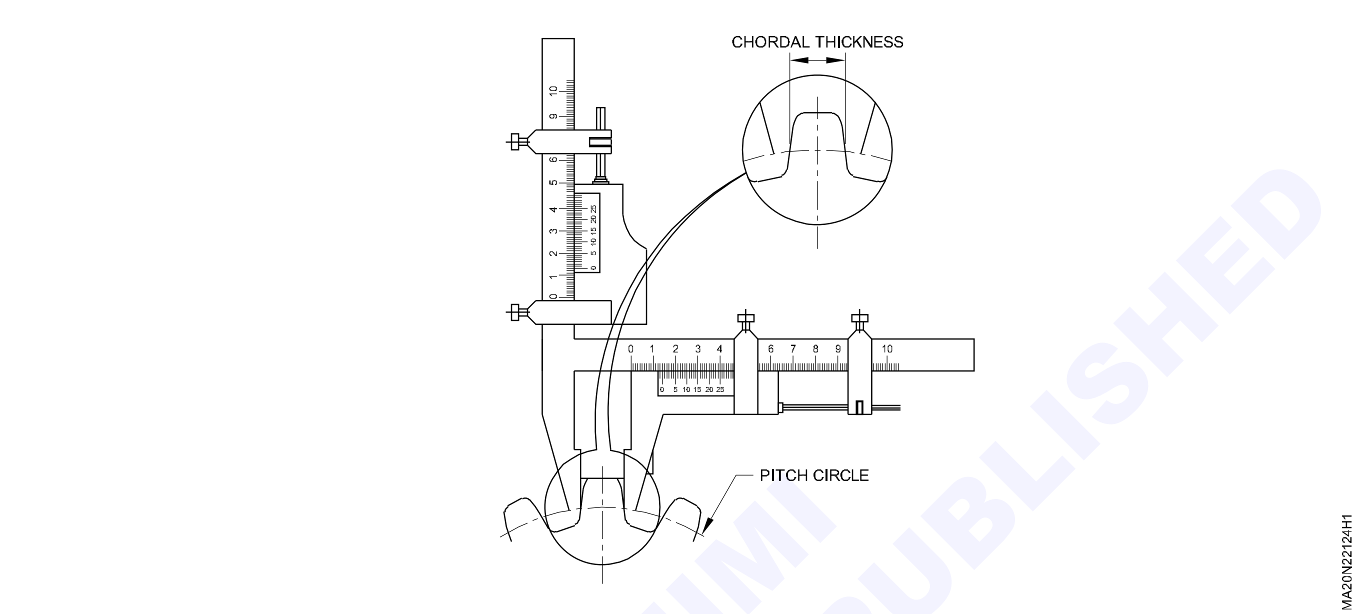
Cut the remaining teeth.

Fig 4



Measurement of teeth by vernier gear tooth caliper

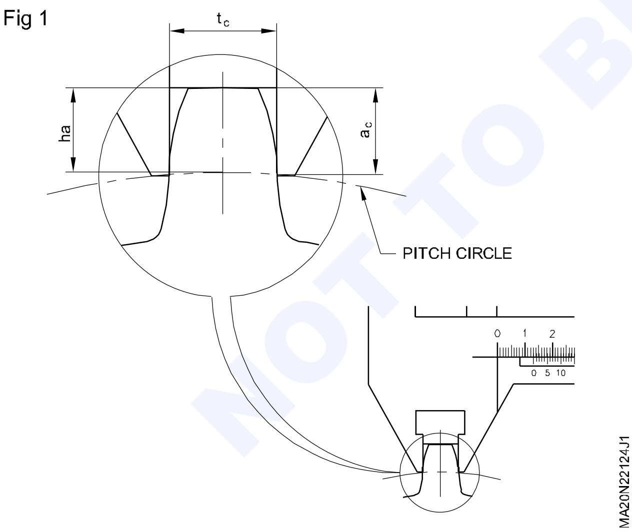
Objectives: At the end of this exercise you shall be able to
- **calculate the values of chordal tooth thick ness and chordal addendum of spur gear**
- **measure the chordal tooth thickness of spur gear.**



Job Sequence

- Calculate the chordal addendum of a spur gear whose module = 2 and number of teeth = 28 Formula to calculate chordal addendum (ac). (Fig 1)

Fig 1



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

Z= number of teeth

$$= 2 + \frac{2 \times 28}{2}\left(1 - \cos\frac{90}{28}\right)$$

= 2+28 (1- cos 3.214)

= 2 + 28 (1 - 0.9984)

= 2 + 28 (0.0016)

= 0.0448 mm

ac = 2.0448 mm

Calculate the chordal tooth thickness of spur gear m = 2 number of teeth = 28 formula to calculate chordal tooth - thickness (tc)

$$tc = m\,z\sin\left(\frac{90}{z}\right)$$

$$= 2 \times 28 \sin\left(\frac{90}{z}\right)$$

= 2x 28 sin (3.214)

= 2x28 x 0.056

tc = 3.136 mm

- Clean the gear tooth Vernier caliper thoroughly
- Set the chordal addendum in vertical scale of gear tooth Vernier caliper
- Place the gear tooth Vernier caliper jaws parallel and perpendicular to the teeth as shown in Fig 2&3

Fig 2

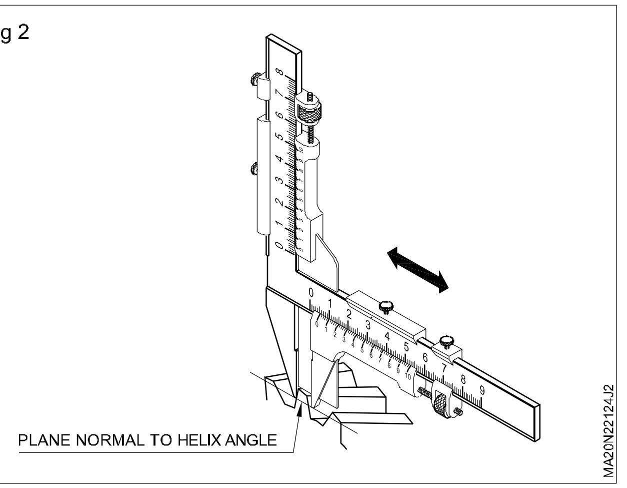
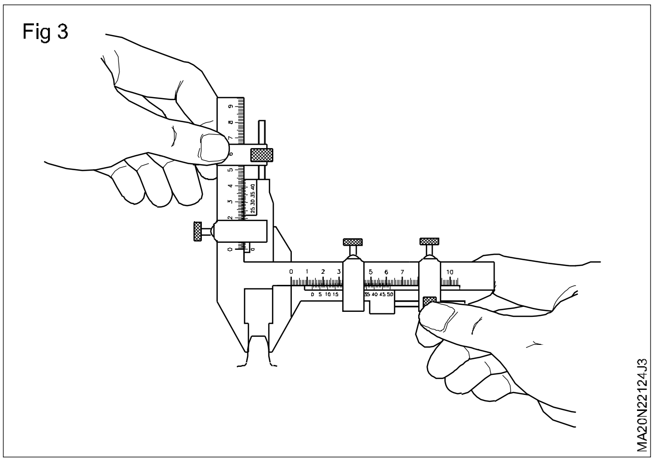


Fig 3



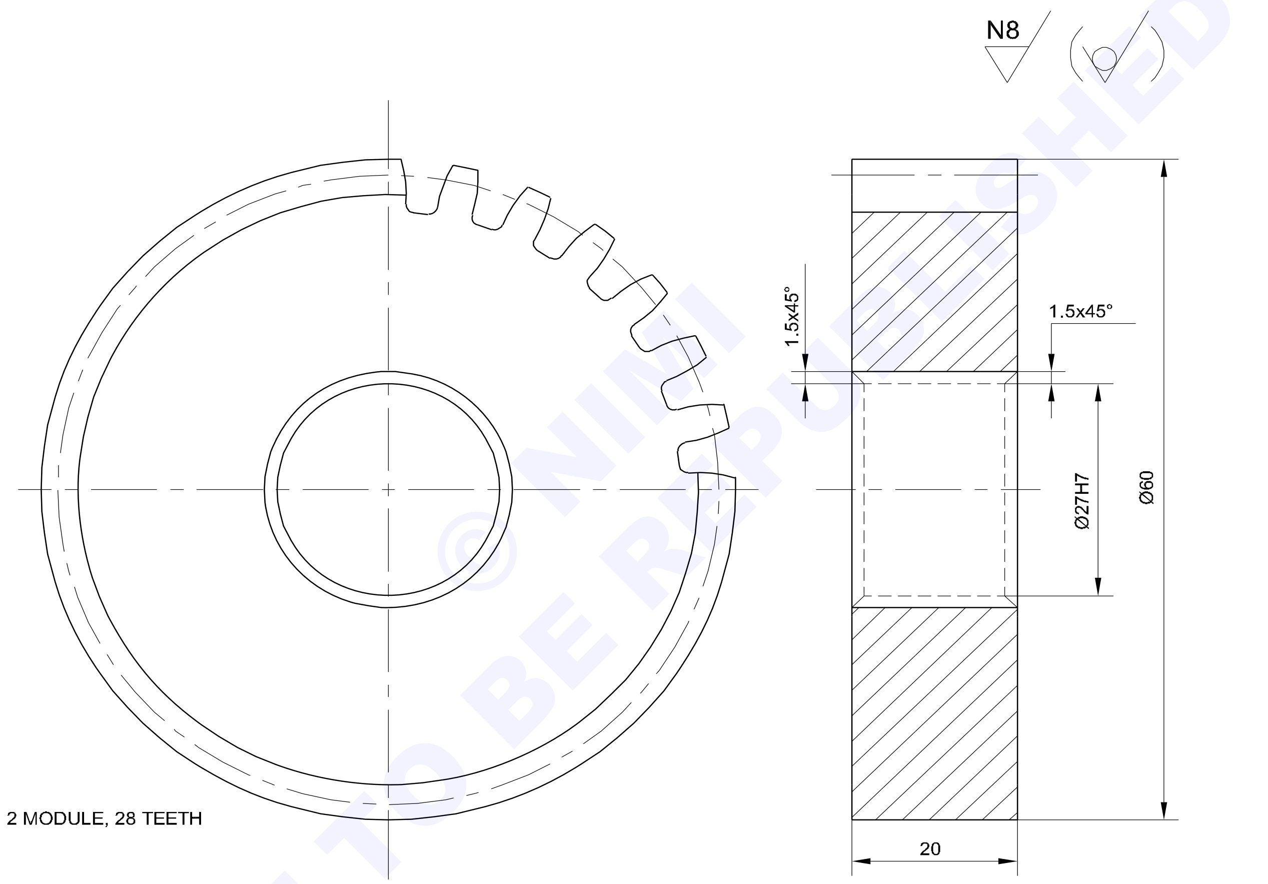
Note : The vertical scale must seat on the top of the gear tooth (on outside diameter)

- Measure the chordal tooth thickness (TC) by adjusting the horizontal scale and note down the reading.
- Repeat the same procedure to measure the other tooth.

Make spur gear using simple indexing with an accuracy 0.05 mm

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

- **mount and align the universal index head on a plain milling machine**
- **set the index head for simple indexing**
- **mount the gear blank in the mandrel and set the mandrel in between centres**
- **mount a module cutter on the arbor**
- **centralise the gear blank with the cutter**
- **cut spur gear on the gear blank**
- **check the chordal addendum with a vernier tooth caliper.**



NOTE:

Ø27H7 - $27^{+0.021}_{+0.000}$

N8 - SMOOTH MACHINING Ra-1.6

PREMACHINED GEAR BLANK AS PER DRAWING

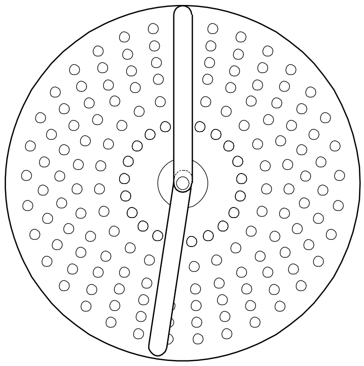
1	Ø65x25mm		C.I			2.2.125
NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.

SCALE 1:1	**SPUR GEAR MILLING BY SIMPLE INDEXING**	DEVIATIONS ±0.05	TIME
		CODE NO. MA20N22125E1	

Job Sequence

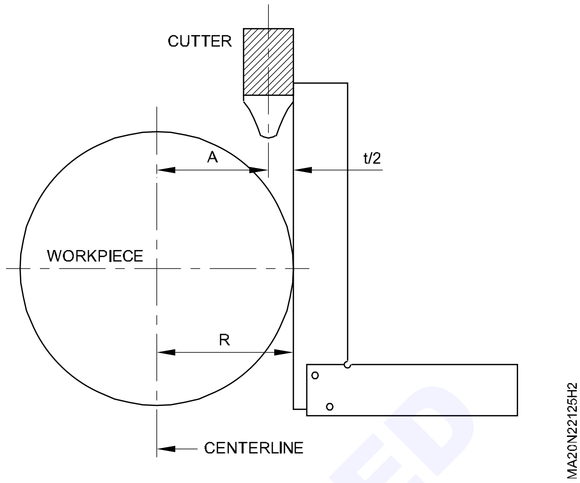
- Check he sizes of the given blank.
- Mount the universal index head and foot stock on the table of the plain milling machine.
- Align the index head and foot stock using a test bar and dial test indicator.
- Set the index head for simple indexing for the number of teeth 28 (one full turns and 9 holes of 21 hole circle)
- Set 9 holes in between the fingers of the sector arms. (Fig 1)

Fig 1



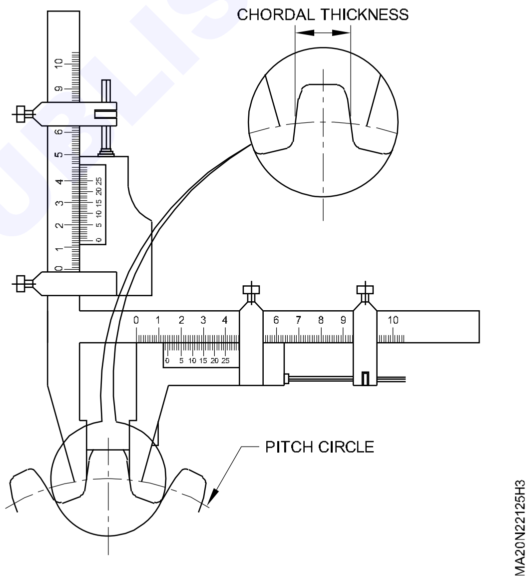
- Mount the blank with the mandrel between the centres of the index head using a drawing plate and drawing dog.
- Mount a 2-module cutter number 4 on the arbor.
- Centralise the gear blank with respect to the cutter centre. (Fig 2) using a try square and cross feed screw as shown in the sketch (A = R - t/2).
- Give a depth of cut equal to 4.5 mm using the vertical feed.
- Set the spindle speed to 90 r.p.m.
- Mill the first tooth space.
- Index for the next tooth space by moving the crank pin through one full turns and 9 holes of 21 hole circle.
- Set the chordal addendum in the gear tooth vernier caliper.

Fig 2



- Check the chordal thickness using a gear tooth vernier caliper. (Fig 3)

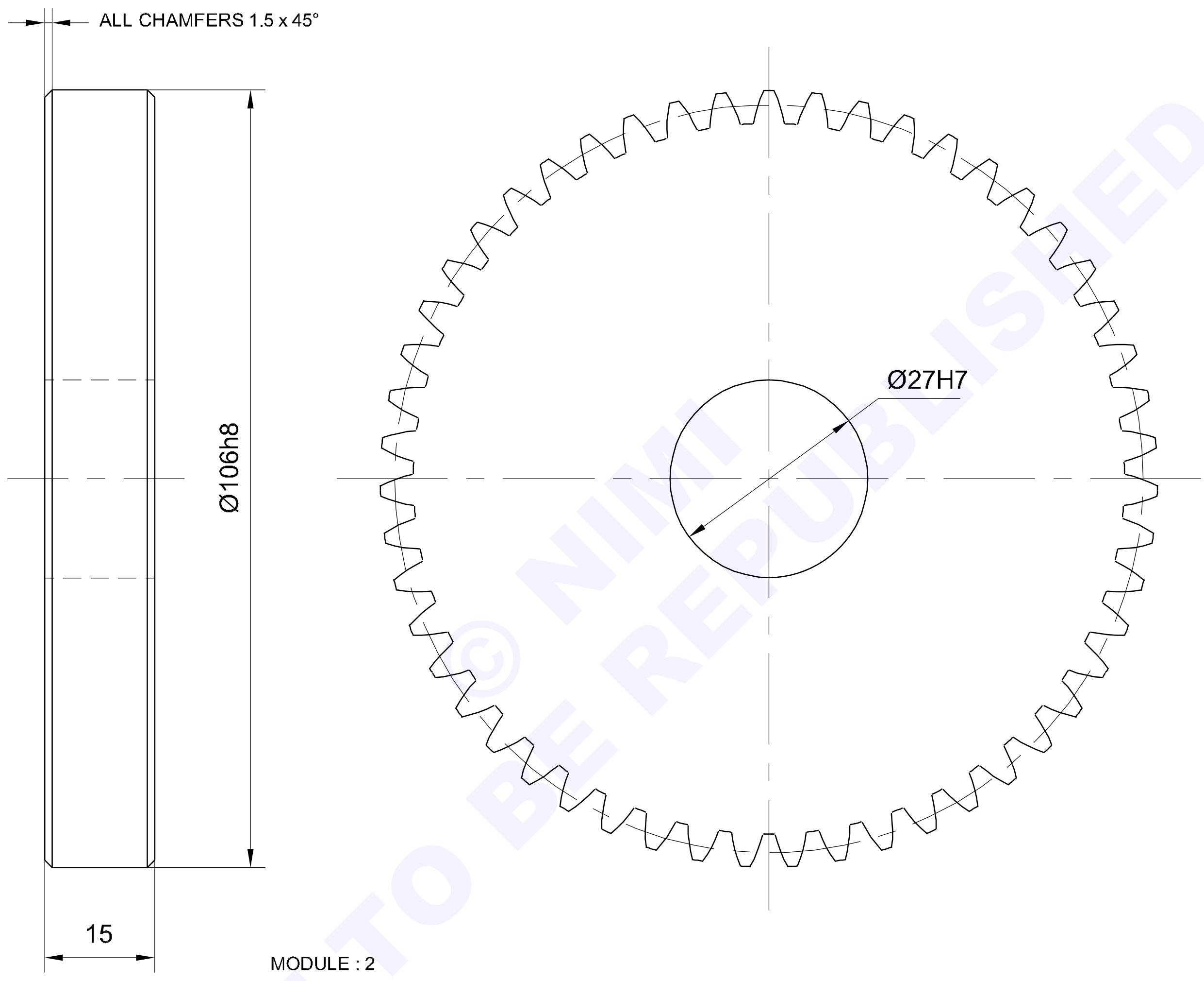
Fig 3



- Repeat the indexing and mill all the other teeth and finish.
- Repeat the indexing and cut all the other teeth and finish.
- Eliminate backlash by moving the index crank only in the forward direction.

Make spur gear using differential Indexing with an accuracy 0.05 mm

Objectives: At the end of this exercise you shall be able to
- **calculate and set the gear train for cutting 51 teeth by differential indexing**
- **check gear tooth with gear tooth vernier caliper**
- **mill the spur gear teeth by differential indexing.**



MODULE : 2

NO. OF TEETH 51

NOTE:

Ø106h8 - 106 $^{+0.000}_{-0.054}$

Ø27H7 - 27 $^{+0.021}_{+0.000}$

PREMACHINED GEAR BLANK AS PER DRAWING

1	Ø110X20mm		C.I	-	-	2.2.126
NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.

SCALE 1:1	**SPUR GEAR MILLING BY DIFFERENTIAL INDEXING**	DEVIATIONS ±0.05	TIME
		CODE NO. MA20N22126E1	

Job Sequence

- Calculate the dia of the gear blank and tooth proportions.
- **Calculation**

OD = m (z + 2)

= 2 (51 + 2)

= 2 (53)

= 106

OD = 106 mm.

Indexing

Gear Ratio $= (A - N)\dfrac{40}{A}$

$= (48 - 51)\dfrac{40}{48}$

$= -3 \times \dfrac{40}{48}$

$= \dfrac{-3\times 24}{1\times 24} \times \dfrac{40}{48}$

$= -\dfrac{72}{24} \times \dfrac{40}{48}$

Driver 72, 40

Driven 24, 48

(A - N) is Negative It Gear train is compound, one idle gear is used.

Index crank movement $= \dfrac{40}{A}$

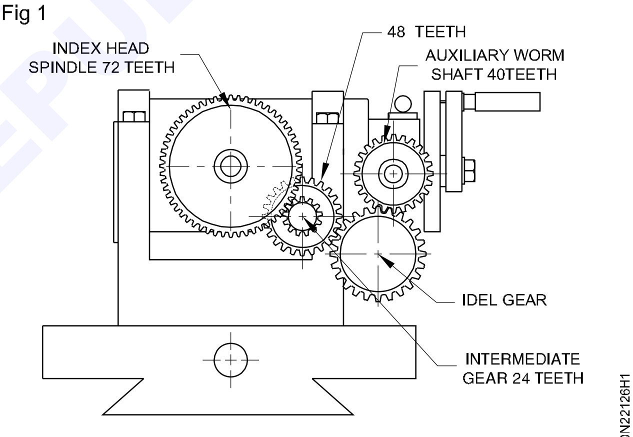
$= \dfrac{40}{48} = \dfrac{10}{12} = \dfrac{5\times 3}{6\times 3}$

$= \dfrac{15}{18}$

Index crank should be move 15 holes in 18 holes circle plate for 51 times.

- Mount the universal index head and foot stock on the table of the plain milling machine.
- Align the index head and foot stock using a test bar and dial test indicator.
- Set the gear train. (Fig 1)
- Mount the cutter No 4 (Module 2) on the arbor.
- Calculate and set the spindle speed.
- Mill the First tooth to the depth of 4.5 mm.
- Index and perform to cut 51 tooth in the same manner.
- Measure the tooth thickness by gear tooth vernier caliper.

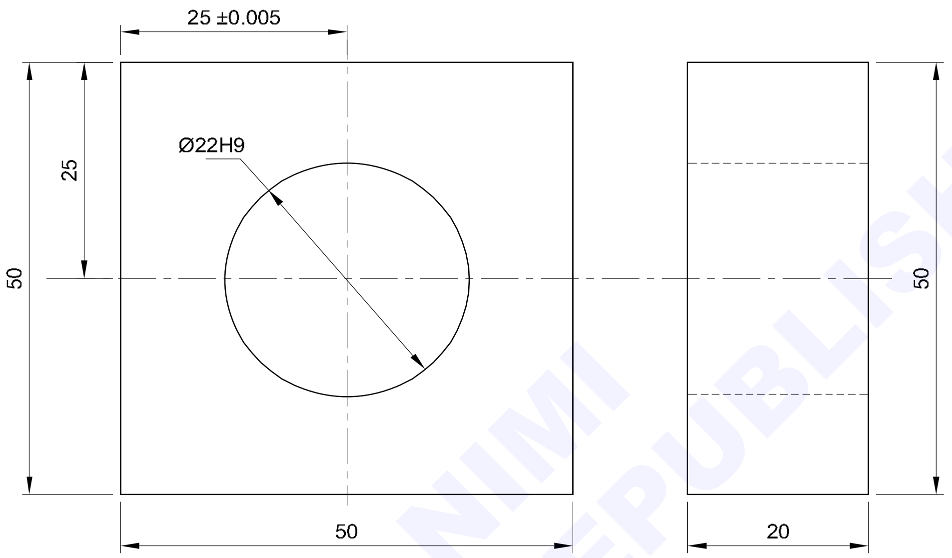
Fig 1



CHANGE GEARS SET UP TO CUT 127 TEETH

Perform boring operation on vertical milling machine with an accuracy 0.05 mm

Objectives: At the end of this exercise you shall be able to
- **prepare vertical milling machine for boring operation**
- **select and mount a boring bar**
- **bore circular holes to an accuracy of H7 using boring bar**
- **check sizes using inside micrometer.**



Job Sequence

- Check the raw material with steel rule.
- Mill to the size of 50 x 50 x 20mm.
- Mark and punch the centre position of ∅ 22 circle.
- Clamp the workpiece on the vice on vertical milling machine.
- Align the centre of the circle with the axis of the spindle using a centering attachment.
- Lock the longitudinal and cross slides.
- Set the spindle speed and power feed to suit the material.
- Centre drill the hole.
- Drill the holes in steps to a maximum of ∅ 20.00mm.
- Mount 15mm boring bar in the boring head.
- Off set the boring bar. Such that the cutting tool just touches the inner surface of the bore and lower the table.
- Give 0.5mm cut and set speed and feed.
- Rough and finish bore to 22mm using boring head.
- Check the size using a inside micrometer.

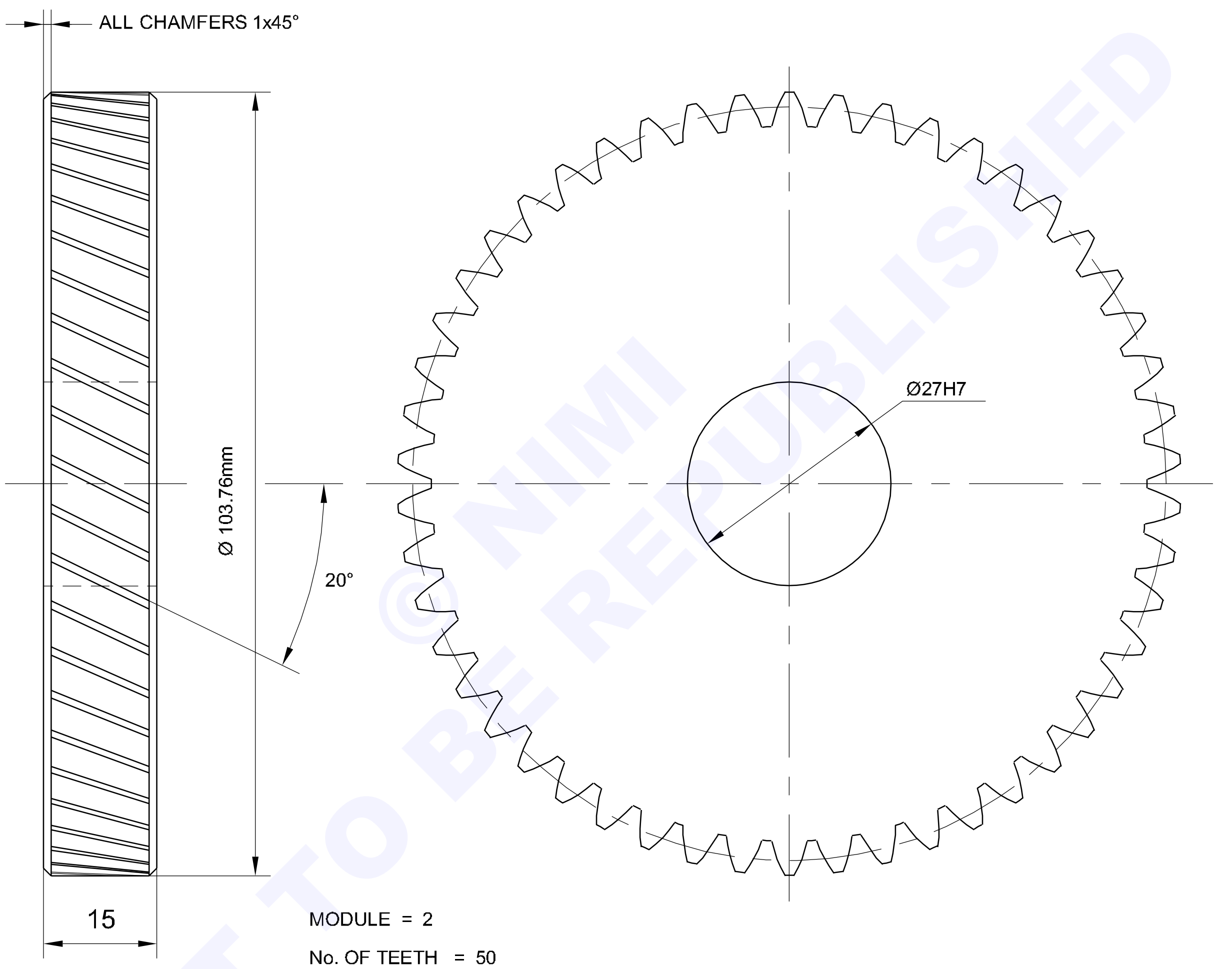
NOTE:
Ø22H9 - $22^{+0.052}_{+0.000}$

1	55x55x25MM	-	C.I	-	-	2.2.127
NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.

SCALE 1:1	BORING OPERATION ON VERTICAL MILLING MACHINE	DEVIATIONS ±0.05	TIME
		CODE NO. MA20N22127E1	

Make helical gear on milling machine with an accuracy 0.05 mm

Objectives: At the end of this exercise you shall be able to
- **set the universal dividing head for helical milling using compound gear trains**
- **mount appropriate module cutter on arbor**
- **swivel table according to the hand of the helix and helix angle**
- **mill the helical teeth to the desired depth.**



MODULE = 2
No. OF TEETH = 50
HELIX ANGLE = 20°R.H

NOTE:

Ø27H7 - $27^{+0.021}_{+0.000}$

PREMACHINED GEAR BLANK AS PER DRAWING

1	Ø 110 X 20 mm	-	-	-	-	2.2.128
NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.
SCALE 1:1	**HELICAL GEAR**				DEVIATIONS ±0.06	TIME
					CODE NO. MA20N22128E1	

Job sequence

Milling helical gear

Calculation of helical gear blank

Given:

Module : 2

No. of teeth : 50

Helix Angle : 20º R.H.

OD = pd + 2 nm

$$pd = \frac{Z \times nm}{Cos\theta},$$

nm = m x cosβ = 2 x 0.9396 = 1.8792

$$pd = \frac{50 \times 1.8792}{0.9396}$$

= 100 mm.

∴ OD = pd + 2 nm

= 100 + (2 x 1.8792)

= 100 + 3.7584

= 103.7584 mm

Tooth depth = 2.25 x nm

= 2.25 x 1.8792

= 4.2282 mm

Selection of Cutter No $2^1 = \frac{Z}{Cos^3\beta}$

$$= \frac{50}{Cos\ 20°}$$

= 60.27 (or) 60

Cutter No. 2

$$\text{Gear Ratio} = \frac{\text{machine lead}}{\text{Job lead}}$$

machine lead = 40 x Pitch of the lead screw.
Assuming Pitch of the lead screw = 5mm
machine lead = 40 x 5 = 200 mm

Job lead : $\tan\beta = \frac{\pi d}{l}$

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

$$l = \frac{3.142 \times 100}{\tan 20^\circ}$$

$$= \frac{3.142 \times 100}{0.3639}$$

= 863.42 mm

Rounded value = 860 mm

$$\text{Gear Ratio} = \frac{\text{machine lead}}{\text{Job lead}}$$

$$\frac{\text{Driver}}{\text{Driven}} = \frac{200}{860} = \frac{20}{86} = \frac{1 \times 20}{2 \times 43} = \frac{1 \times 24}{2 \times 24} \times \frac{20 \times 2}{43 \times 2}$$

$$= \frac{24}{48} \times \frac{40}{86}$$

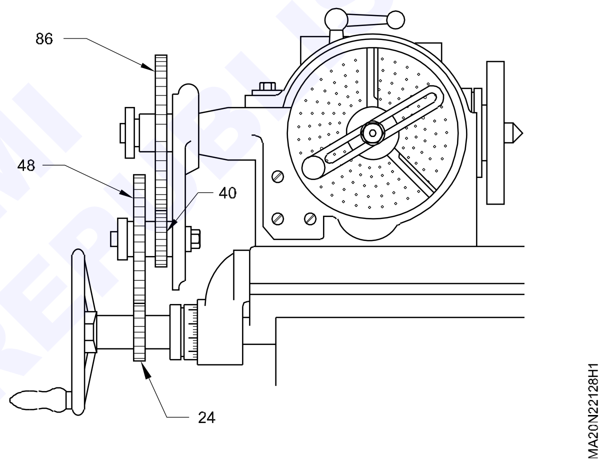
The gears 24, 40 are the driver gears and 48,86 driven gears.

Milling a helical gear

- Set the universal index head for helical milling, using a compound gear train of

$$\frac{24 \times 40}{48 \times 86} = \frac{\text{Drivers}}{\text{Drivens}} \quad \text{(Fig 1)}$$

Fig 1



- Set the index head to index 50 divisions by simple indexing.

$$\frac{40}{N} = \frac{40}{50} = \frac{4 \times 4}{5 \times 4} = \frac{16}{20}$$

- The index crank should be moved 16 holes in 20 holes circle plate for 50 times.
- Mount the job with a mandrel in between centres and align it.
- Mount 2 module cutter no. 2 (55–134 teeth) on the arbor.

Refer to related theory on selection of cutter number for helical gear.

- Centralise the job with respect to the cutter.
- Set the depth of cut to 4.23 mm.
- Lock the vertical feed.
- Swivel the table to 20° for right hand helix.

Refer to Skill Sequence on helix milling for the correct direction of swivelling.

- Feed the blank to mill the first tooth space.

Ensure that the back stopper pin is disengaged from the index plate.

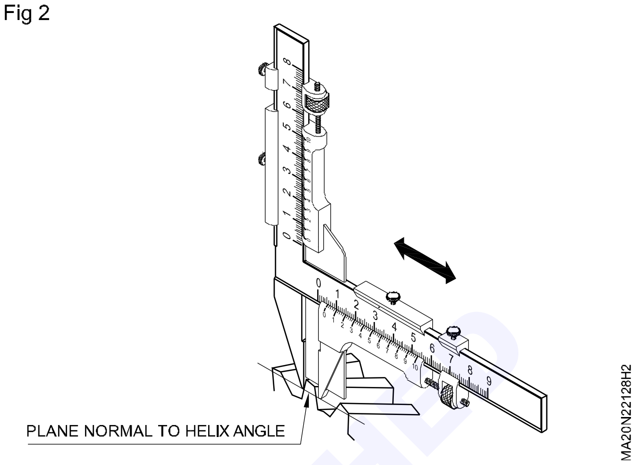
Stop the machine and take the table back to starting point.

- Index the blank (16 spaces in 20 hole circle) for next tooth space.

Eliminate backlash before indexing. Do not move the table while indexing.

- Cut the second tooth space.
- Remove the burrs.
- Check form of involute curve for symmetry and thickness of the tooth using vernier gear tooth caliper, at a point normal to the helix angle. (Fig 2)
- If necessary make the required adjustments.
- Repeat the same steps and mill the remaining teeth.
- Deburr the gear on the centre lathe using a second cut file.

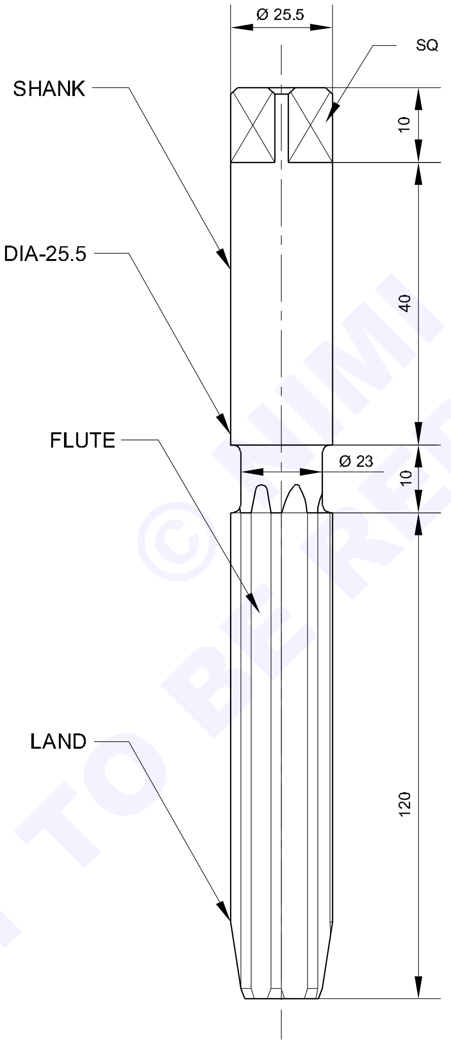
Fig 2



Make straight flute milling on milling machine with an accuracy 0.05 mm

Objectives: At the end of this exercise you shall be able to
- **calculate, set the universal dividing head with proper index plate**
- **select and position the cutter in the arbor**
- **offset the cutter from the centre line of the job to obtain the rake angle**
- **mill flutes and land of the reamer.**

Diameter of blank = 25.5 mm
Number of Teeth = 8



1	Ø30 x 185		Fe310	-	-	2.2.129
NO.OFF	STOCK SIZE	SEMI PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.

SCALE 1:1	**STRAIGTH FLUTE MILLING**	DEVIATIONS ±0.06	TIME
		CODE NO. MA20N22129E1	

Job Sequence

- Check the size of raw material as per blank size by steel rule.
- Machining the blank of straight flute reamer to the required size on the lathe.
- A universal dividing head and tailstock bolted on the milling machine, after setting their axis exactly perpendicular to the machine spindle (horizontal universal milling machine).
- The work (reamer blank) is mounted between the centers or supported centre of tailstock and hold it by three jaw chuck of dividing head.
- Ø3"x1/2"x1"x45° single angle milling cutter is mounted on the middle of the arbor, and is then centered accurately with the dividing head spindle axis by adjusting the positioning of the table.
- The alignment of the cutter with the blank is checked by raising the table when the centre of the cutter must touch the centre point of tailstock.
- Select the proper index plate, bolt it to the dividing head and set the index crank and sector arm.
- The table is raised till the cutter just touches the periphery of the reamer blank. Micro dial of the vertical feed screw is set to the zero position.
- Then the blank is set over from the radius position relative to the cutter centre line, by rotating the index crank one complete turn (9 degree). The blank is again centered with the cutter, by shifting the cross feed screw.
- The table is next raised to give the required depth of cut 3.2 mm. The machine is started and feed is applied to finish the first tooth space of reamer. After the end of cut the table is brought back to the starting position. Index for next tooth space by rotating the 5 complete turn. The operation is repeated till all the teeth are completed.
- Remove the single angle milling cutter from the arbor and then Ø4"x1/2"x1" side and face milling cutter mounted on the arbor.
- By left the space of land 1/8" to 1/64" touches the cutter with the periphery of the blank by lifting the table.
- After that feed applied and finish the first tooth.
- The operation is repeated till all the teeth (8teeth) are completed.
- Then stop the machine.

Necessary calculations

Blank diameter = 25.5mm. Number of teeth = 8

$$\text{Depth of cut} = \frac{\text{Diameter of the blank}}{\text{Number of flute}}$$

$$= \frac{25.5}{8} = 3.18 = 3.2 \text{ (approx)}$$

Then, Depth of cut = 3.2mm

Indexing: $\frac{40}{N}$

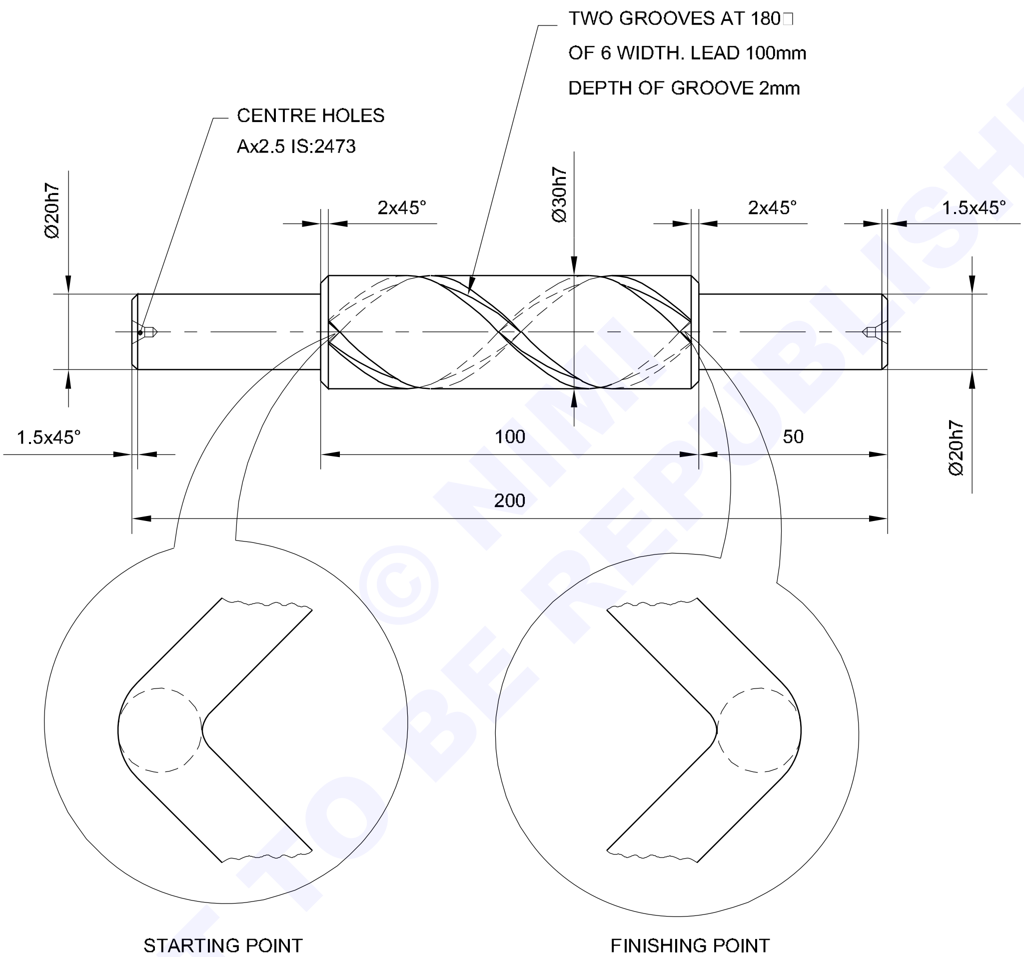
Where, N = Number of flute

Then, $\frac{40}{N} = \frac{40}{8}$ = 5 complete turn

Offset: 9 degree offset to be applied for rake angle by rotating the index crank one complete turn.

Make helical flute on milling machine with an accuracy 0.02 mm

Objectives: At the end of this exercise you shall be able to
- **set the cutter**
- **mount the indexing head (universal) and gear train**
- **set the Helix angle**
- **cut the Helical flute.**



NOTE:

Ø 20h7 - 20 $^{+0.000}_{-0.021}$

Ø 30h7 - 30 $^{+0.000}_{-0.025}$

PREMACHNED BLANK AS PER DRAWING

1	Ø32x205	-	Fe310	-	-	2.2.130
NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.

SCALE 1:1	**HELICAL FLUTE MILLING**	DEVIATIONS ±0.06	TIME
		CODE NO. MA20N22130E1	

Job Sequence

Mill helical flute on vertical milling machine using end mill

- Clean the working surfaces of the table, index head and foot stock. Mount the index head at the left end of the table.
- Arrange a gear train for 100 mm lead of helix and for right hand helix.

$$\text{Gear ratio} = \frac{\text{Machine lead}}{\text{job lead}} = \frac{\text{Driver}}{\text{Driven}}$$

> **There is no need to swivel the table for the helix angle.**

> **Ensure that the gear driving mechanism operates freely and the back stopper pin is disengaged from the index plate.**

- Mount a dia. 6 mm slot drill or end mill in the collet chuck and fix the collet chuck in the spindle of the vertical milling machine.
- Mount the job between centres and fix the dog in the driving bracket.
- Align the end mill to the centre of the job. Adjust the table position at the starting point. (Refer to the Job drawing.)
- Set the spindle speed to suit the 6 mm dia. end mill or slot drill for mild steel.
- Start the machine spindle. Apply depth of cut 2 mm gradually.
- Feed the workpiece slowly by turning the index crank of the index head, and mill the right hand helical flute up to the finishing point. (Refer to the Job drawing.)
- Stop the machine spindle, lower the table and bring back the position of the end mill to the starting point.
- Alter the gear train to mill left hand helix, by adding one idler gear.
- Start the spindle, raise the table and apply depth of cut, until the cutter face just touches the bottom of the flute already milled.
- Start feeding the workpiece by rotating the index crank and mill the left hand helical flute up to the finishing point.

> **Ensure that the back stopper pin is disengaged from the index plate before starting feeding.**
>
> **Take care of backlash, when you reverse the feed direction.**

- Stop the machine spindle, remove the job and deburr helical flute using smooth flat file.

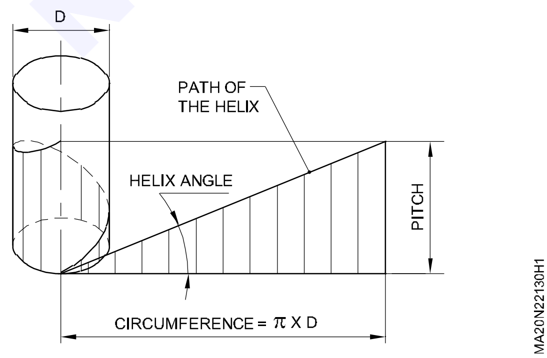
Skill Sequence

Milling helix on universal milling machine

Objectives : This shall help you to
- **cut helical grooves on cylindrical workpieces**
- **cut helical flutes on slab milling cutters, end mill cutters and hand reamers**
- **cut teeth on helical gear.**

Helix is the form of line generated by the progressive rotation of a point around an axis of a cylinder. (Fig 1) This operation of cutting helix is often required in a machine shop, when milling helical teeth on milling cutters, end mills, counterbores, twist drills, reamers, helical gears and cams. Helical milling can be done on the universal milling machines with the help of a universal dividing head.

Fig 1



To cut a helix on universal a milling machine, proceed as follows.

Fix universal dividing head at the left hand end of the work table of the universal milling machine.

Select an index plate suitable for the number of divisions required and fix it. Set the sector arms.

> **Disengage the back pin from the index plate and insert the index plunger in the numbered hole of the index plate.**

Determine the lead of helix for the given workpiece from the working drawing.

$$\text{Rule} = \frac{\pi \times \text{outside dia}}{\text{Tan of helix angle}}$$

> **In case of helical gears instead of outside diameter, P.D ie pitch diameter of the gear should be taken into calculation.**

Calculate, select and fix the change gears suitable for the lead calculated.

$$\text{Rule} = \frac{\text{Lead of machine}}{\text{Lead of helix}}$$

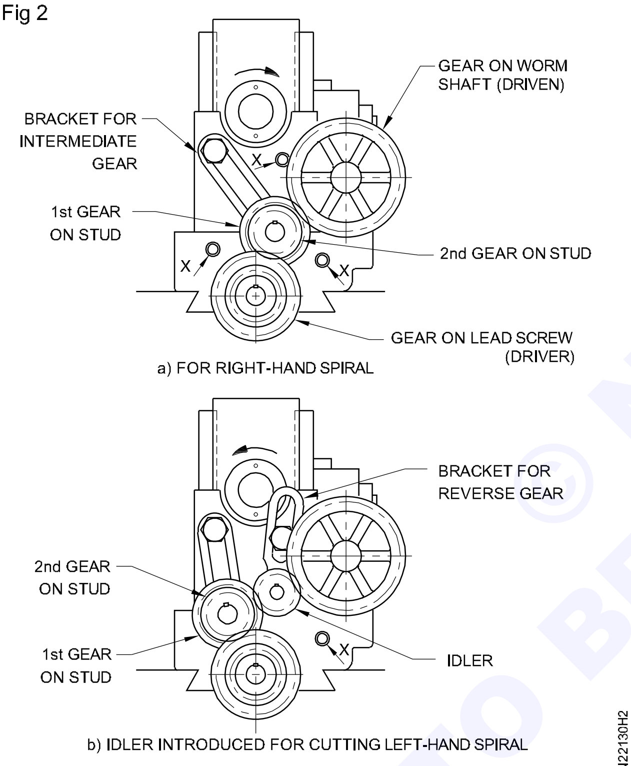
$$= \frac{\text{Driving gears}}{\text{Driven gears}}$$

$$= \frac{\text{gear on lead screw} \times \text{1st gear on stud}}{\text{gear on worm shaft} \times \text{2nd gear on stud}}$$

Refer to Fig 2 for the correct position of gears in the gear train for quick and correct selection of change gears.

Refer to Table I for selection of the change gears supplied for helical milling.

Fig 2



ARRANGEMENT OF CHANGE GEARS

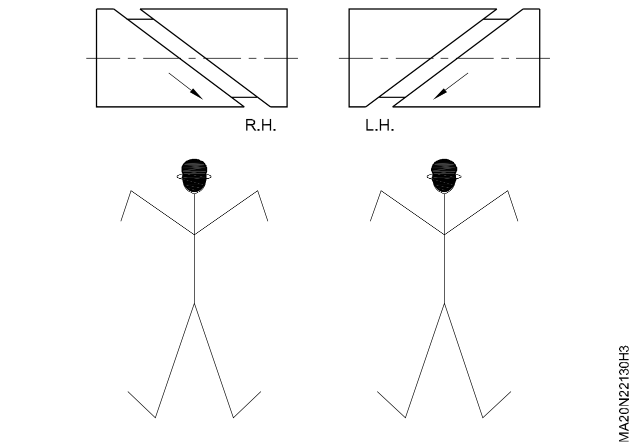
Know the lead of your machine yourself. If you are not thorough in understanding the term "machine lead", refer to Related Theory on helical milling.

Ensure the hand of helix from the working drawing or sample given (Fig 3) and fix accordingly one idler or two idlers, or no idler. (Fig 2)

Check the direction of rotation of the spindle of the dividing head, when the work table advances towards the cutter.

When looking from the back of the dividing head, for R.H. helix, the spindle should rotate in clockwise direction and for L.H. helix in anticlockwise direction. (Fig 2)

Fig 3

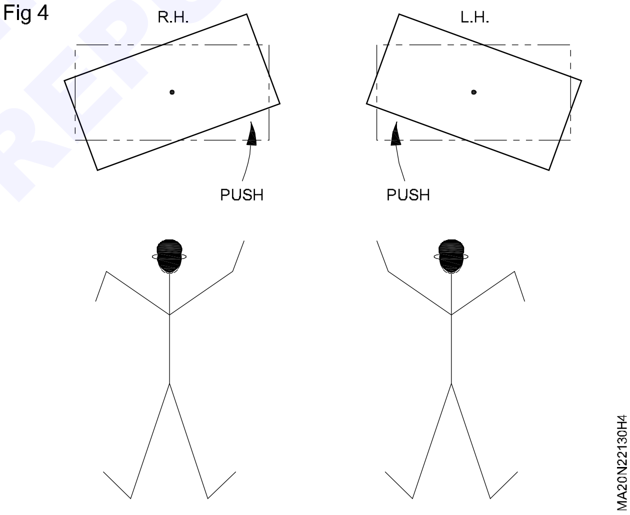


Fix the work between centres, ensure that the driving dog properly 'fits in' (without slack) in the dog driver.

When the job is held between the self-centering chuck and centre, there is a chance the chuck may be loosened particularly when cutting left hand helix, because of the threaded end of the spindle nose of the dividing head. This point should be taken care of.

Swivel the table to the calculated angle of the helix. The direction of table swivelling will be according to the hand of the helix. (Fig 4)

Fig 4

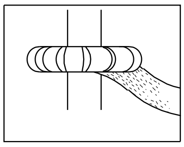
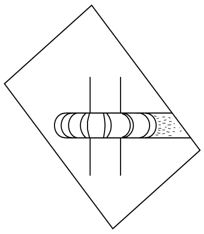
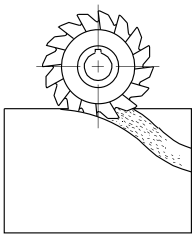
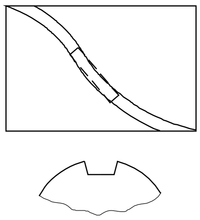
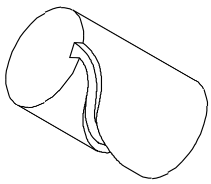
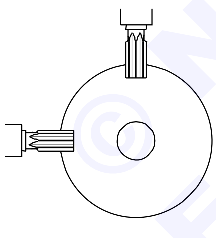


Move the table transversely ('cross-feed') until there is 20 mm or so, clearance between the table and the column.

Fix the arbor (long arbor) in the machine spindle.

The diameter of the arbor should correspond to the diameter of the bore of the cutter to be selected.

Select a cutter suitable for the given helical operation. (Fig 5)

Effect of Table not swivelled during helical milling.	Efects of Table swivelled during helical milling.
 Width of the groove is more than the cutter width.	 Width of the groove is equal to the width of the cutter.
 Shape of the groove will not be the same as the cutter.	 Shape of the groove will be same as the cutter.
 Width and shape is same as the cutter.	 Width and shape is same as the cutter.

Only those cutters, whose side cutting edges incline more or less towards each other, ie. a double equal angle cutter, or double unequal angle cutter or convex cutter or gear tooth cutter may be used.

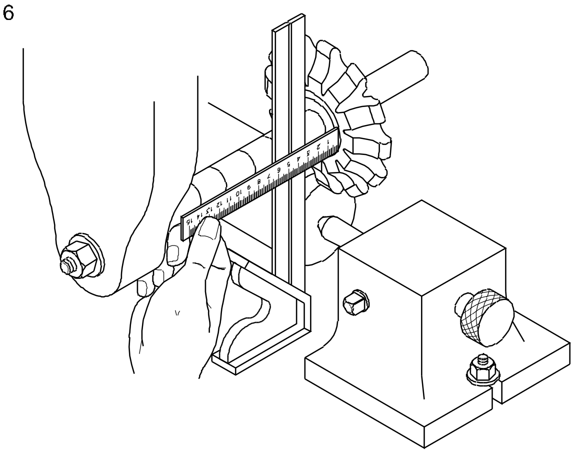
Tighten the cutter on the arbor in its proper position with reference to the workpiece.

Bring the table back to horizontal (zero) and set the end of the work under the axis of the cutter. This is in order to have the cutter in such a position, that the full depth and exact shape of the groove will be cut. It is easier to judge the proper setting when the table is horizontal.

Adjust the position of blank to the centre of milling cutter. (Fig 6)

Swivel the table again to the required angle and clamp tightly.

Fig 6



Set the speed of the spindle for the machine to suit the material of the workpiece. (Refer to Related Theory on cutting to speed and feed of milling cutters.)

Start the machine spindle. Raise the table and get the 'feel' of touching the cutter on the workpiece. Set the graduated dial of the vertical movement at '0'.

Apply depth of cut (full or part thereof) and feed by hand the longitudinal movement. Ensure that the width of groove is exactly equal to that of the cutter.

Direct the flow of coolant at the point of cutting and mill the first groove.

Ensure that the cutter does not foul the dog or the jaws of the chuck, at the end of the cut.

Stop the machine spindle. Reverse the table to the starting position. Leave atleast 10 mm clearance between the cutter and the edge of the workpiece.

Lock the index plate in position by inserting the 'back pin'' and index for the next flute/teeth.

Do not forget to release the back pin as soon as indexing for the next division is over.

Start the machine spindle, cut the second flute/tooth.

Ensure that the index head spindle is not locked by its lock lever while the cutting operation is progressing.

Stop the machine spindle and reverse the table again to the starting position.

Check the thickness of the tooth, (in case of helical gear) by the gear tooth vernier caliper at 90° to the helix angle. In the case of grooves, check the width and depth, by suitable measuring instruments.

If necessary adjust the depth of cut to correct the width of the groove, or the thickness of the tooth.

Repeat the above steps and complete the helical milling operation.

CHANGE GEARS FOR SPIRAL MILLING

Fig A

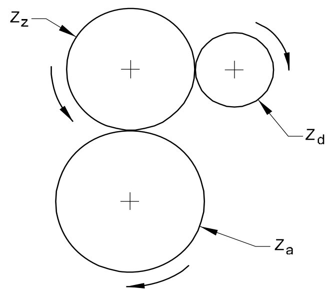


Fig F

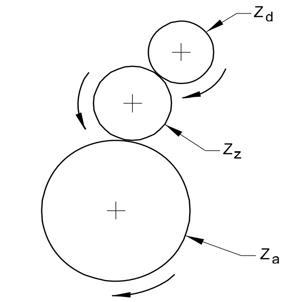


Fig B

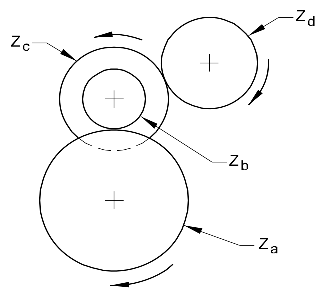


Fig G

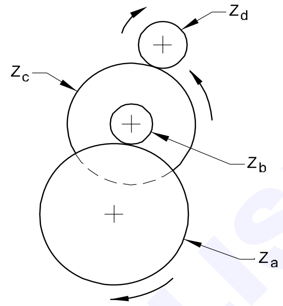


Fig C

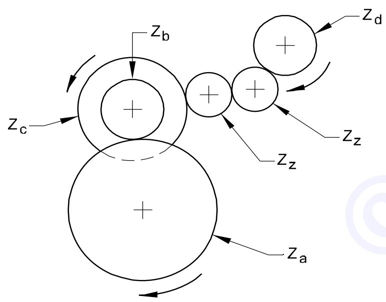


Fig H

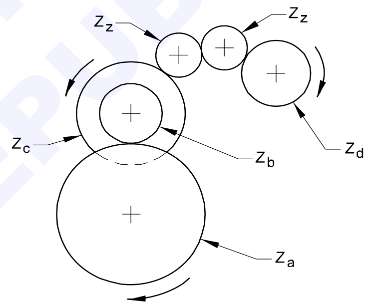


Fig D

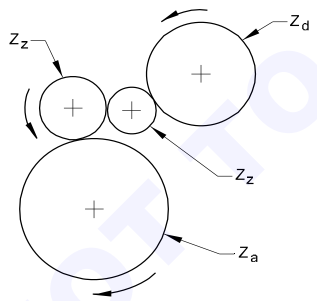


Fig I

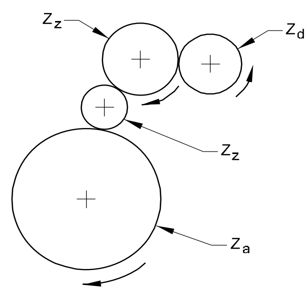


Fig E

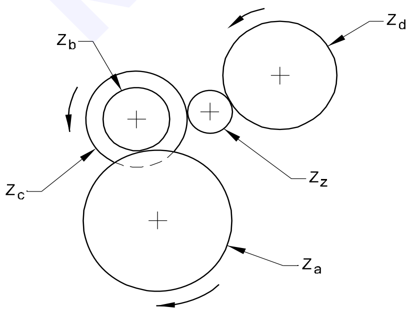
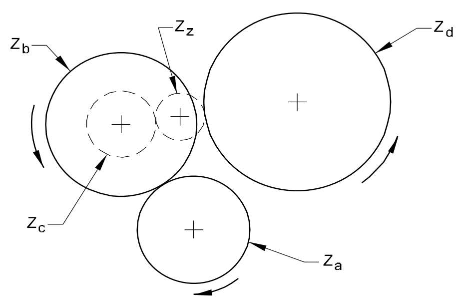


Fig J



MA20N22130H7

TABLE 1

CHANGE GEARS FOR SPIRAL MILLING

Lead (m m)	Number of teeth in change gears: Za	Zb	Zc	Zd	Type of Dividing Head: 130 Spiral R-H (Figure)	130 Spiral L-H (Figure)	180 Spiral R-H (Figure)	180 Spiral L-H (Figure)
24.56	100	24	86	44	B	E	G	K
26.06	100	28	86	40	B	E	G	K
26.79	100	24	86	48	B	E	G	K
28.66	100	28	86	44	B	E	G	K
29.33	100	24	72	44	C	E	H	K
31.11	100	40	72	28	C	E	H	K
31.26	100	48	86	28	B	E	G	K
32.75	100	44	86	32	B	E	G	K
34.23	100	44	72	28	B	E	H	K
35.56	100	40	72	32	B	E	H	K
35.72	100	48	86	32	B	E	G	K
36.00	100	48	64	24	C	E	H	K
36.47	100	56	86	28	B	E	G	K
37.33	100	48	72	28	B	E	G	K
38.15	100	44	64	28	C	E	H	K
39.12	100	44	72	32	B	E	G	K
40.00	100	40	64	32	C	E	H	K
42.00	100	48	64	28	C	E	H	K
42.67	100	48	72	32	B	E	G	K
43.41	86	56	72	24	C	E	G	K
44.00	100	44	64	32	C	E	H	K
44.65	100	48	86	40	B	E	G	K
46.89	100	72	86	28	B	E	G	K
47.63	100	64	86	32	B	E	G	K
48.00	100	48	64	32	B	E	H	K
49.00	100	56	64	28	B	E	G	K
49.65	86	48	72	32	C	E	H	K
49.78	100	56	72	32	B	E	G	K
50.65	86	56	72	28	C	E	H	K
52.10	100	56	86	40	B	E	G	K
53.34	100	48	72	40	B	E	G	K
54.00	100	24	64	40	B	E	G	K
55.00	100	44	64	72	B	E	G	K
56.89	100	64	72	32	B	E	G	K
57.89	86	56	72	32	B	E	G	K
58.67	100	48	72	44	B	E	G	K
59.54	100	64	86	40	B	E	G	K
60.00	100	48	64	40	B	E	G	K
61.72	100	24	56	72	B	E	G	K
62.22	100	56	72	40	B	E	G	K
62.78	86	24	64	72	B	E	G	K
63.00	100	28	64	72	B	E	G	K
64.00	100	-	-	32	A	D	F	I
64.50	100	24	64	86	B	E	G	K
65.12	86	56	64	32	C	E	H	K
66.00	100	48	64	44	B	E	G	K
66.67	72	-	-	24	A	D	F	I
68.22	86	48	72	44	B	E	G	K
69.77	86	48	64	40	C	E	H	K
70.00	100	56	64	40	B	E	G	K
71.11	100	64	72	40	B	E	G	K
72.00	100	72	64	32	B	E	G	K
73.14	100	64	56	32	B	E	G	K
73.67	100	72	86	44	B	E	G	K
74.42	86	-	-	32	A	D	F	I
75.00	64	-	-	24	A	D	F	I
75.43	100	48	56	44	B	E	G	K
76.44	100	32	72	86	B	E	G	K
77.00	100	56	64	44	B	E	G	K
78.55	100	24	44	72	C	E	H	K
79.59	86	56	72	44	B	E	G	K
80.00	100	-	-	40	A	D	F	I
81.40	86	56	64	40	B	E	G	K
82.69	86	64	72	40	B	E	G	K
83.33	100	64	86	56	B	E	G	K
84.00	100	56	64	48	B	E	G	K
85.05	86	32	56	64	C	E	H	K
86.00	100	32	64	86	B	E	G	K
86.82	86	56	72	48	B	E	G	K
88.00	100	-	-	44	A	D	F	I
88.90	72	-	-	32	A	D	F	I
89.54	86	56	64	44	B	E	G	K
90.00	100	72	64	40	B	E	G	K
90.96	86	64	72	44	B	E	G	K
91.64	100	28	44	72	B	E	H	K
93.34	100	56	48	40	B	E	H	K
95.56	100	86	72	40	B	E	G	K
96.00	100	-	-	48	A	D	F	I
97.67	86	56	64	48	B	E	G	K
98.29	100	32	56	86	B	E	G	K
99.00	100	72	64	44	B	E	G	K
99.56	100	64	72	56	B	E	G	K
100.00	64	-	-	32	A	D	F	I
100.59	100	64	56	44	B	E	G	K
101.75	86	28	64	100	B	E	G	K
102.33	86	-	-	44	A	D	F	I
103.20	100	24	40	86	B	E	G	K
104.51	72	28	64	86	B	E	G	K
105.12	100	86	72	44	B	E	G	K
106.30	86	64	56	40	B	E	G	K

Lead	Number of teeth in change gears				Type of Dividing Head			
					130 Spiral		180 Spiral	
m m	Za	Zb	Zc	Zd	R-H	L-H	R-H	L-H
					Figure			
107.15	100	72	86	64	B	E	G	K
108.00	100	72	64	48	B	E	G	K
109.10	44	-	-	24	A	D	F	I
109.01	100	64	56	48	B	E	G	K
111.11	72	-	-	40	A	D	F	I
111.63	86	-	-	48	A	D	F	I
112.00	100	-	-	56	A	D	F	I
113.14	100	72	56	44	B	E	G	K
114.67	100	48	72	86	B	E	G	K
115.76	86	64	72	56	B	E	G	K
116.28	86	32	64	100	B	E	G	K
117.34	100	64	48	44	B	E	G	K
118.25	100	44	64	86	B	E	G	K
119.07	72	24	56	100	B	E	G	K
120.00	100	40	48	72	B	E	G	K
121.53	72	28	64	100	B	E	G	K
122.86	100	40	56	86	B	E	G	K
123.43	100	72	56	48	B	E	G	K
124.03	86	64	48	40	B	E	H	K
125.00	64	-	-	40	A	D	F	I
125.58	86	72	64	48	B	E	G	K
126.00	100	72	64	56	B	E	G	K
126.85	86	24	44	100	B	E	G	K
128.00	100	-	-	64	A	D	F	I
129.00	100	86	64	48	B	E	G	K
130.23	86	-	-	56	A	D	F	I
130.91	100	40	44	72	B	E	G	K
131.56	86	72	56	44	B	E	G	K
132.00	100	44	48	72	B	E	G	K
132.89	86	32	56	100	B	E	G	K
133.78	100	86	72	56	B	E	G	K
134.38	100	56	40	43	B	E	H	K
135.66	86	28	48	100	B	E	G	K
136.51	72	32	56	86	B	E	G	K
137.50	64	-	-	44	A	D	F	I
138.89	72	32	64	100	B	E	G	K
139.54	86	40	48	72	B	E	G	K
140.80	100	44	40	64	B	E	G	K
142.86	56	-	-	40	A	D	F	I
143.34	100	40	48	86	B	E	G	K
144.00	100	-	-	72	A	D	F	I
145.35	86	40	64	100	B	E	G	K
146.55	86	72	64	56	B	E	G	K
147.43	100	48	56	86	B	E	G	K
147.98	86	28	44	100	B	E	G	K
148.84	86	-	-	64	A	D	F	I

Lead	Number of teeth in change gears				Type of Dividing Head			
					130 Spiral		180 Spiral	
m m	Za	Zb	Zc	Zd	R-H	L-H	R-H	L-H
					Figure			
149.34	100	64	48	56	B	E	G	K
150.00	64	-	-	48	A	D	F	I
150.50	100	86	64	56	B	E	G	K
151.52	71	24	44	100	C	E	H	K
152.38	72	64	56	48	C	E	H	K
152.89	100	86	72	64	B	E	G	K
153.49	86	44	48	72	B	E	G	K
155.04	86	32	48	100	B	E	H	K
155.56	72	-	-	56	A	D	F	I
156.36	100	40	44	86	B	E	G	K
157.67	100	44	48	86	B	E	G	K
158.40	100	44	40	72	B	E	G	K
158.73	72	32	56	100	B	E	G	K
159.25	72	32	48	86	B	E	G	K
160.00	100	40	32	64	C	E	H	K
162.04	72	28	48	100	B	E	G	K
162.97	100	64	44	56	B	E	G	K
164.58	100	72	56	54	B	E	G	K
166.1	86	40	56	100	B	E	G	K
166.67	48	-	-	40	A	D	F	I
167.45	86	-	-	72	A	D	F	I
168.00	100	72	48	56	B	E	G	K
169.30	86	32	44	100	B	E	G	K
171.43	56	-	-	48	A	D	F	I
172.00	100	-	-	86	A	D	F	I
172.80	100	48	40	72	B	E	G	K
173.64	72	40	64	100	B	E	G	K
174.42	86	48	64	100	B	E	G	K
175.00	64	-	-	56	A	D	F	I
176.00	100	44	32	64	C	E	H	K
176.78	64	72	56	44	C	E	H	K
177.78	72	-	-	64	A	D	F	I
178.58	64	32	56	100	B	E	G	K
179.17	72	48	64	86	B	E	G	K
180.00	100	40	32	72	B	E	H	K
181.82	44	-	-	40	A	D	F	I
182.28	64	28	48	100	C	E	H	K
182.72	86	44	56	100	B	E	G	K
183.27	100	72	44	56	B	E	G	K
184.14	86	44	40	72	C	E	H	K
185.19	72	32	48	100	B	E	G	K
186.05	86	32	40	100	B	E	G	K
187.64	100	48	44	86	B	E	G	K
189.20	100	44	40	86	B	E	G	K
190.00	72	44	64	100	B	E	G	K
192.00	100	72	48	64	B	E	G	K
192.86	64	72	56	48	B	E	H	K

Lead	Number of teeth in change gears				Type of Dividing Head			
					130		180	
mm	Za	Zb	Zc	Zd	Spiral		Spiral	
					R-H	L-H	R-H	L-H
					Figure			
193.50	100	72	64	86	B	E	G	K
195.35	86	56	48	72	B	E	G	K
196.57	100	64	56	86	B	E	G	K
198.00	100	44	32	72	B	E	G	K
199.07	72	40	48	86	B	E	G	K
199.34	86	48	56	100	B	E	G	K
200.00	24	-	-	24	A	D	F	I
200.67	100	56	48	86	B	E	G	K
201.60	100	56	40	72	B	E	G	K
202.02	72	32	44	100	B	E	G	K
203.49	86	56	64	100	B	E	G	K
204.76	72	48	56	86	B	E	G	K
206.40	100	48	40	86	B	E	G	K
207.41	72	56	48	64	C	E	H	K
208.34	72	48	64	100	B	E	G	K
209.03	72	56	64	86	B	E	G	K
211.17	64	44	56	86	B	E	G	K
213.11	86	56	44	72	B	E	G	K
215.00	100	40	32	86	B	E	G	K
216.00	100	48	32	72	B	E	G	K
217.17	72	40	44	86	C	E	H	K
218.18	44	-	-	48	A	D	F	I
218.91	100	56	44	86	B	E	G	K
219.43	100	48	28	64	C	E	H	K
220.00	40	-	-	44	A	D	F	I
221.14	100	72	56	86	B	E	G	K
222.22	72	32	40	100	B	E	H	K
223.22	64	40	56	100	B	E	G	K
223.96	64	40	48	86	B	E	H	K
224.00	100	56	32	64	B	E	G	K
225.00	64	-	-	72	A	D	F	I
226.28	72	56	44	64	B	E	H	K
227.28	64	32	44	100	C	E	H	K
228.58	56	-	-	64	A	D	F	I
229.33	100	64	48	86	B	E	G	K
230.36	64	48	56	86	B	E	G	K
231.49	72	40	48	100	B	E	G	K
232.56	86	-	-	100	A	D	F	I
233.33	48	-	-	56	A	D	F	I
234.40	86	56	40	72	B	E	G	K
236.50	100	44	32	86	B	E	G	K
238.08	72	48	56	100	B	E	G	K
238.89	72	-	-	86	A	D	F	I
240.00	40	-	-	48	A	D	F	I
240.80	100	56	40	86	B	E	G	K
243.06	72	56	64	100	B	E	G	K
245.54	64	44	56	100	B	E	G	K

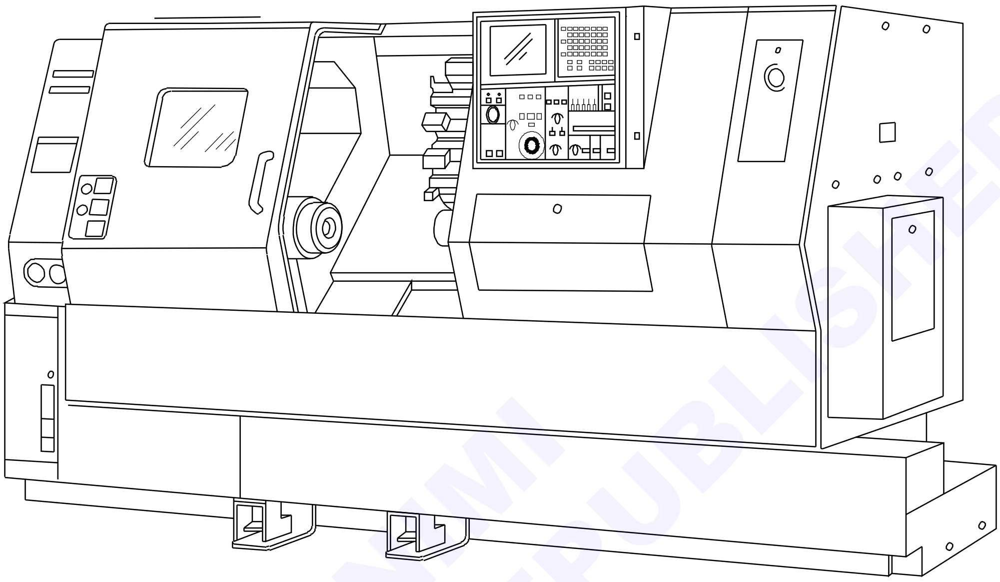
Lead	Number of teeth in change gears				Type of Dividing Head			
					130		180	
mm	Za	Zb	Zc	Zd	Spiral		Spiral	
					R-H	L-H	R-H	L-H
					Figure			
246.86	100	48	28	72	B	E	G	K
250.00	32	-	-	40	A	D	F	I
251.16	86	48	32	72	C	E	H	K
252.00	100	56	32	72	B	E	G	K
252.53	72	40	44	100	B	E	G	K
254.63	72	44	48	100	B	E	G	K
255.81	86	44	40	100	B	E	G	K
256.00	100	56	28	64	B	E	G	K
257.15	56	-	-	72	A	D	F	I
258.00	100	72	48	86	B	E	G	K
260.61	72	48	44	86	B	E	G	K
261.63	86	72	64	100	B	E	G	K
262.50	64	56	48	72	C	E	H	K
264.00	100	44	24	72	C	E	H	K
265.78	86	64	56	100	B	E	G	K
266.67	48	-	-	64	A	D	F	I
267.86	64	48	56	100	B	E	G	K
268.75	64	-	-	86	A	D	F	I
270.29	100	44	28	86	B	E	G	K
271.32	86	56	48	100	B	E	G	K
273.02	72	64	56	86	B	E	G	K
275.00	32	-	-	44	A	D	F	I
277.78	72	-	-	100	A	D	F	I
278.71	72	56	48	86	B	E	G	K
279.07	86	48	40	100	B	E	G	K
280.00	40	-	-	56	A	D	F	I
281.45	100	86	44	72	B	E	G	K
284.08	64	40	44	100	B	E	G	K
285.72	28	-	-	40	A	D	F	I
286.46	64	44	48	100	B	E	G	K
288.00	100	64	32	72	B	E	G	K
290.68	86	40	32	100	B	E	G	K
293.02	86	56	32	72	B	E	G	K
294.86	100	48	28	86	B	E	G	K
295.98	86	56	44	100	B	E	G	K
297.62	56	40	48	100	B	E	H	K
298.67	100	56	24	64	B	E	H	K
299.01	86	72	56	100	B	E	R	K
300.00	48	-	-	72	A	D	F	I
301.00	100	56	32	86	B	E	G	K
303.04	72	48	44	100	B	E	G	K
304.05	72	56	44	86	B	E	G	K
305.56	72	44	40	100	B	E	G	K
307.14	56	-	-	86	A	D	F	I
309.60	100	72	40	86	B	E	G	K
310.08	86	64	48	100	B	E	G	K
312.05	64	-	-	100	A	D	F	I
313.55	64	56	48	86	B	E	G	K
314.29	28	-	-	44	A	D	F	I

Lead	Number of teeth in change gears				Type of Dividing Head			
					130		180	
mm	Za	Zb	Zc	Zd	Spiral		Spiral	
					R-H	L-H	R-H	L-H
					Figure			
315.34	100	44	24	86	B	E	G	K
317.46	72	64	56	100	B	E	G	K
318.52	72	64	48	86	B	E	G	K
319.77	86	44	32	100	B	E	G	K
320.00	40	-	-	64	A	D	F	I
322.50	64	48	40	86	C	E	H	K
324.08	72	56	48	100	B	E	G	K
325.58	86	56	40	100	B	E	G	K
327.38	56	44	48	100	B	E	G	K
329.18	100	64	28	72	B	E	G	K
332.23	86	40	28	100	B	E	G	K
333.33	72	48	40	100	B	E	G	K
334.45	72	56	40	86	B	E	G	K
336.00	100	56	24	72	B	E	G	K
338.27	86	64	44	100	B	E	G	K
340.91	64	48	44	100	B	E	G	K
342.86	56	64	48	72	C	E	H	K
343.75	64	44	40	100	B	E	G	K
344.00	100	64	32	86	B	E	G	K
345.54	64	72	56	86	B	E	G	K
347.47	72	64	44	86	B	E	G	K
350.00	32	-	-	56	A	D	F	I
353.54	72	56	44	100	B	E	G	K
357.15	56	-	-	100	A	D	F	I
358.34	72	48	32	86	C	E	H	K
360.00	40	-	-	72	A	D	F	I
364.58	64	56	48	100	B	E	G	K
365.45	86	44	28	100	B	E	G	K
366.67	24	-	-	44	A	D	F	I
370.38	72	64	48	100	B	E	G	K
372.08	86	64	40	100	B	E	G	K
373.27	72	86	64	100	B	E	G	K
375.00	64	48	40	100	B	E	G	K

Personal and CNC machine safety, safe handling of tools safety switches & material handling

Objectives : At the end of this exercise you shall be able to
- **follow personal safety in CNC workshop**
- **maintain safety of CNC machine .**

Fig 1



Do's

- A well trained operator should operate the CNC Machine.
- Only one operator should operate the machine at a time.
- Check the lubrication oil and Hydraulic oil level before starting the machine.
- Ensure doors are closed before switching ON the Machine.
- Keep less speed while operating in JOG mode, especially when the tool is near the chuck/Job.
- Operator should ensure the machine zero point while starting the machine.
- Operator should check the work offset for every required set tool and the same to be entered in the program.
- Special care should be taken while changing the tool.
- Check the part program for correction before operating.
- Learn all G codes, and M codes, of the control installed in your machine .
- Learn all offset, Referrence points pertaining to your machine.
- Learn the basic maintenance schedule for your machine.
- Ensure that the stabilizer is ON before starting.

Don'ts

- Do not operate machine without the working knowledge of the machine.
- Do not operate the machine when covers are removed.
- Do not insert any bar or tool holder in the spindle while rotation.
- Do not open the control panel, without switching OFF power.
- Do not operate the machine without trying in simulation.
- Do not attend electrical fault, without removing the main fuse carriers.

CNC machine safety system

The built-in safety system on a CNC machine includes guards and protective devices which should be securely fitted and always kept in position while the machine is being used. It may include.

Emergency Stop Button

Used to shut down the machine immediately. It is located on the control panel and at other points on the machine, for example the hand held unit.

Soundproof Casing

Reduces noise emission generated by the operating section and protects the operator from the risk of flying objects or tool fragments.

Curtain Guards

Made of PVC and designed to protect the operator from the risk of airborne chips or tool fragments.

Guard Fence

The fence marks the working area in which the machine moves. It protects the operator from the risk of interference with moving parts. The guard may be of an open type or made of mesh.

Below mention points are some general personal safety rules that you can use as a guide only. You might like to add any other rules that apply to you.

Tool Safety

Below are some general tooling safety rules that you can use as a starting guide. You might like to add any other rules that apply to you.

Do:

- Always check that the machine is not operating when loading a tool magazine.
- Always check that tools are in good condition, for example, sharp and free of cracks.
- Always check that tools are set correctly.
- Always check that the correct tool data is entered into the CNC program.
- Always test tools before use.
- Always check that the seating surfaces are clean before installing tools.
- Always check that spindle direction is correct for right-hand or left-hand operation.
- Only use tools within the limits specified by the manufacturer.
- Only tighten tools to recommended torque values.

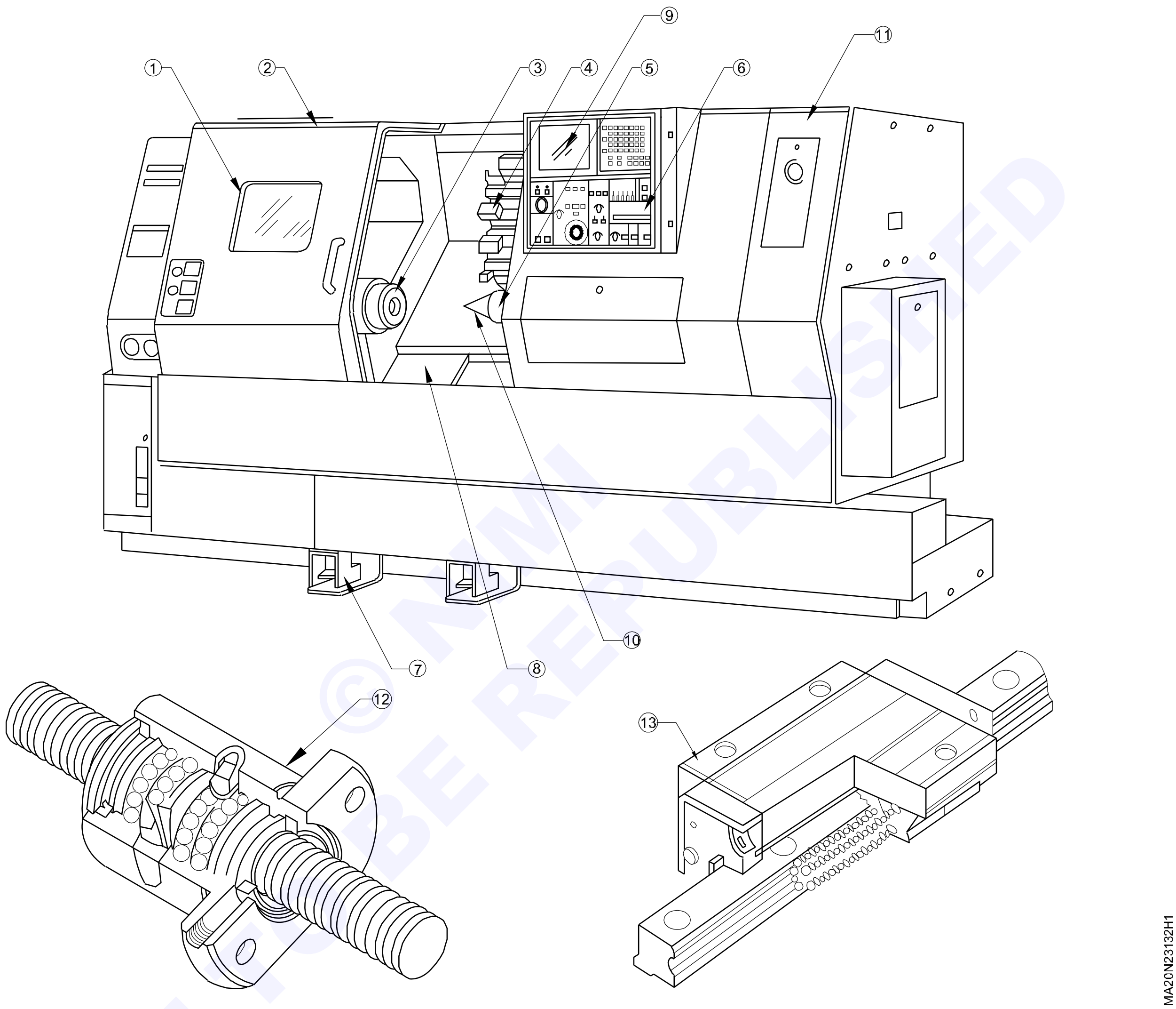
Material handling

Material handling now features technology that matches the sophistication of the machines that prepare the pieces. Swing arm panel feeders ensure continuous operation; panel stackers provide many options for storage and retrieval; robotics transfer pieces from machine to machine; conveyors and lift systems speed the transfer of materials with a minimum of human involvement; fully automated stacking/destacking machines easily handle input from multiple conveyors and trolleys.

Demonstration of CNC lathe machine, its parts and function

Objectives : At the end of this exercise you shall be able to
- **identify the parts of CNC lathe machine**
- **list out the functions of each part of the CNC lathe machine.**

Fig 1



Job Sequence

- Identify the parts of CNC lathe machine and its function.
- List out the name of the parts in the given table 1
- Instructor will demonstrate the parts and its functions.

Instructor to guide to identy CNC machine parts and axis control.

Table 1

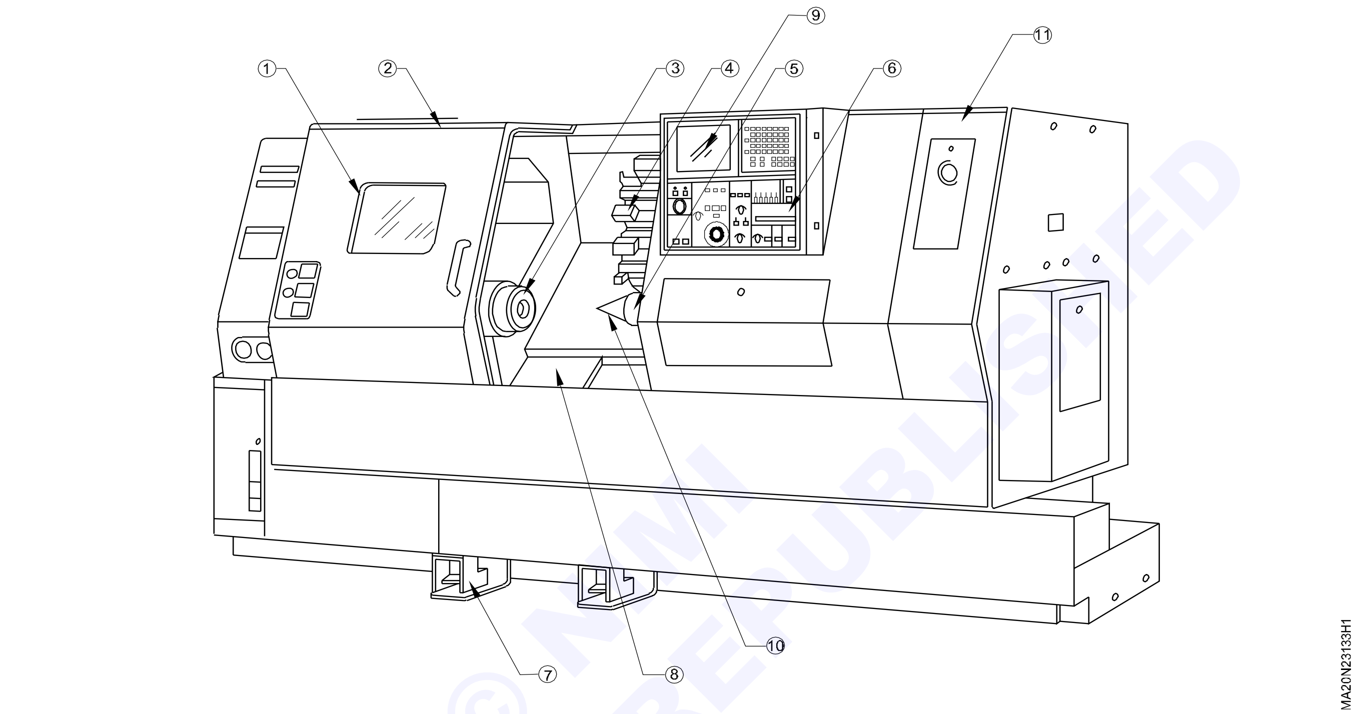
Part No.	Name of the parts	Function of the part
1		
2		
3		
4		
5		
6		
7		
8		
9		
10		
11		
12		
13		

Get it checked by the instructor.

Working of CNC machine parts by using multimedia based simulator

Objectives: At the end of this exercise you shall be able to
- **operate the multimedia based simulator**
- **identify the CNC machine parts on simulator**

Fig 1



- **Instructor will show the C.N.C Machine parts by using multimedia basd simulator Trainees should identify and understand the working parts and write in the parts given below table - 1**
- **Instructor to refer the previous exercise.**

Table - 1

SI No.	CNC parts Identified by trainee in simulator
1	
2	
3	
4	
5	
6	
7	
8	
9	
10	
11	

Identify machine over travel limits and emergency stop on the machine

Objectives: At the end of this exercise you shall be able to
- **identify the over travel limit switches and x and z axes in CNC turning center**
- **identify emergency stop and operate it.**

Job sequence

TASK 1 : **Indentification of machine over travel limit switches in x and y axis**

There are two types of over travel limit

1 Soft over travel

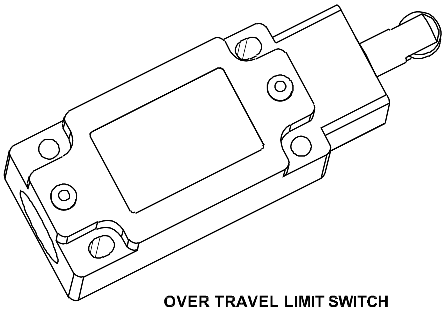
2 Hardware over travel

Software over travel can be controlled by the specific parameter

Hardware over travel limit is controlled by limit switch.

Identification of hardware over travel switch (Fig 1)

Fig 1



OVER TRAVEL LIMIT SWITCH

- Position the X axis in the middle of the x axis movement
- Remove the telescopic cover at both ends.
- Observe the over travel limit switched in both direction that is x - and x +.
- Similarly position the z axis movement in the middle of the lathe bed.
- Open the telescopic cover at both the ends and observe the observer travel limit switches.
- Refix the telescopic cover , take care no dust practical should enter into the remove slide.
- Fig 2 shows the x axis dog and limit switch
- Fig 3 shows the z axis dog and limit switch

Fig 2

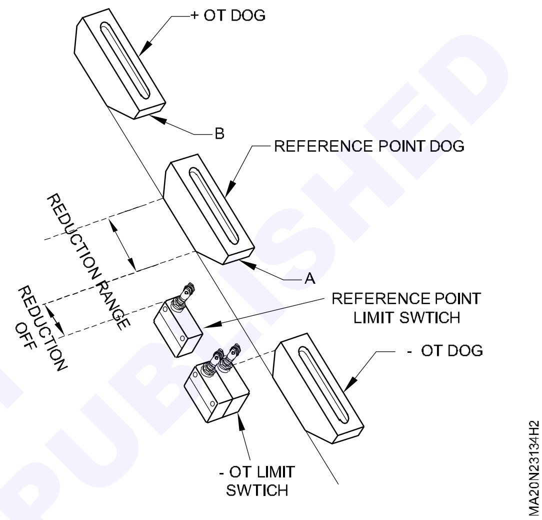
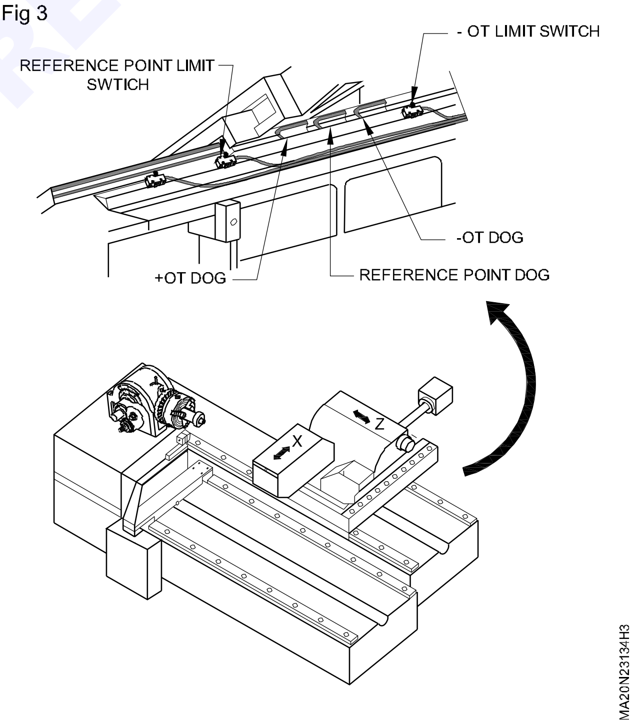


Fig 3



— — — — — — — —

TASK 2 : **Identification emergency stop bush button**

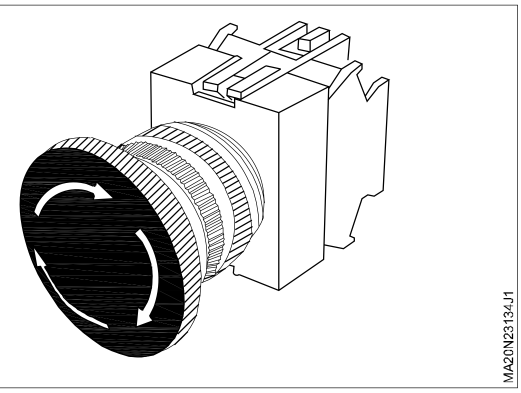
Emergency stop button are designed in such a manner in which their role is more physical, such as interrupting a power supply to the machine control system. It is a basic big red pushbuttons fixed on machine control panel.

Emergency stop pushbutton that has mechanical plastic or metal tabs and grooves internally such that when you push it (interrupting the circuit), it is held in that position until you twist it. They are designed to be large, hard to miss, and easy to push sample is given Fig 1.

Note : Pratice to put off and relace the emergency switch.

Caution: Do not try to rotate in anticlockwise direction.

Fig 1



— — — — — — — —

Tool path for turning, facing, grooving, threading and drilling

Objective: At the end of this exercise you shall be able to
- **layout the tool path for various turning operation.**

Job Sequence

TASK 1: **Tool path for end facing operation (Fig 1&2)**

- P1 positioning facing tool from tool reference point in rapid movement.
- P2 positioning tool for facing on work piece in rapid movement.

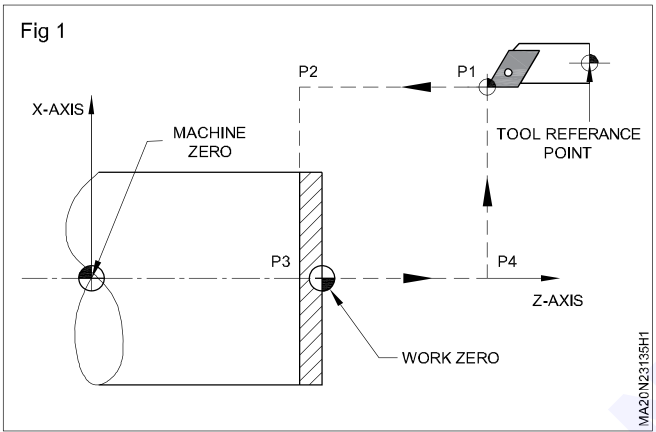


Fig 1

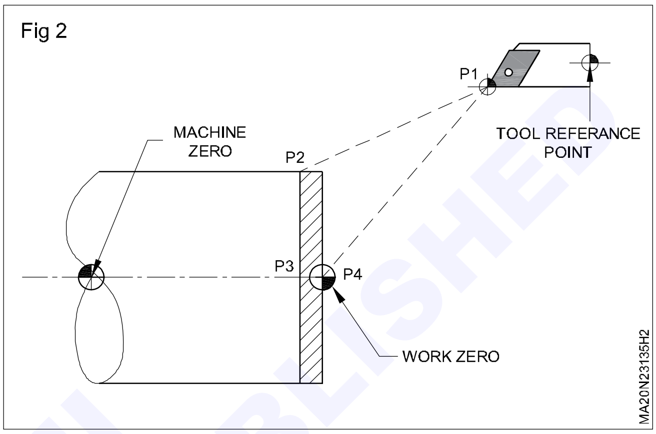


Fig 2

- P3 End facing operation in feed movement.
- P4 with drawing tool from work piece in rapid movement
- P5 positioning the tool in reference point in rapid movement.

TASK 2: **Turning tool path (Fig 1)**

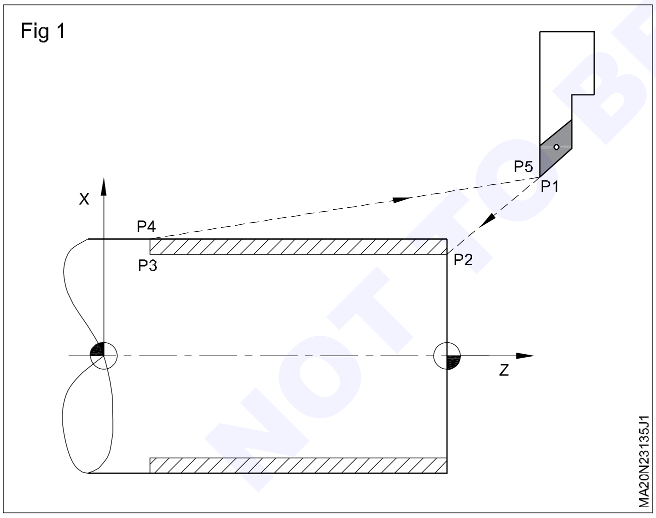
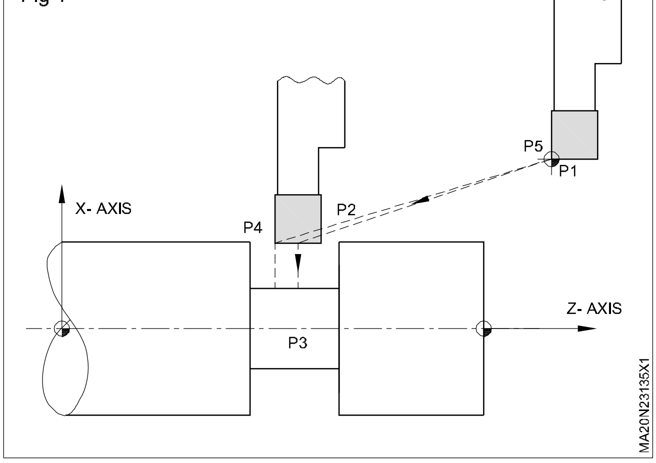


Fig 1

- P1 to P5 - indicate the tool movement
- Dotted line indicate tool movement is rapid
- Thick line indicate the tool movement in cutting feed.

TASK 3: **Grooving tool path (Fig 1)**

Fig 1

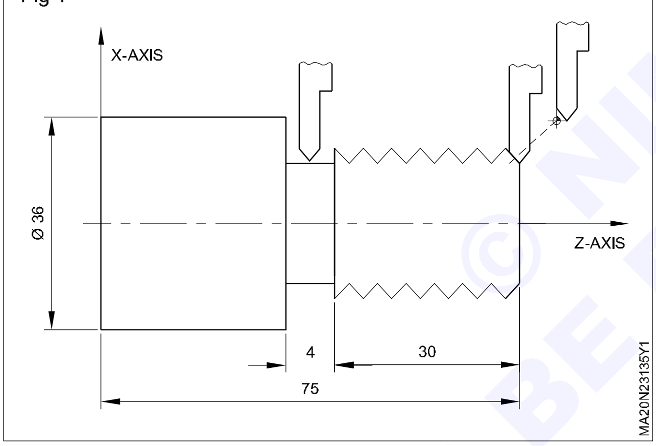


- P1 to P5 - Indicate the tool movement for grooving operation.
- Position the tool at required length in rapid movement.
- Feed the tool to the required depth in cutting feed
- With draw the tool just away from the outside diameter
- Send back the tool to the reference point

TASK 4: **Threading tool path (Fig 1 & 2)**

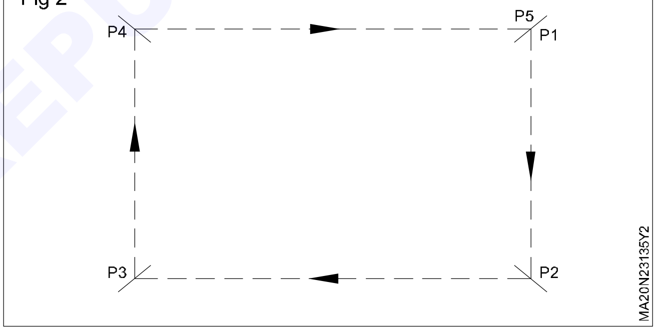
- P1 = positioning of threading tool from reference point in rapid movement
- P2 = giving depth of cut to cut thread

Fig 1



- P3 = start cutting thread to required length
- P4 = with draw the tool in X axis just away from the threading diameter.
- P5 = Once again position the tool rabidly in p1.
- Repeat the same until threading depth is completed.

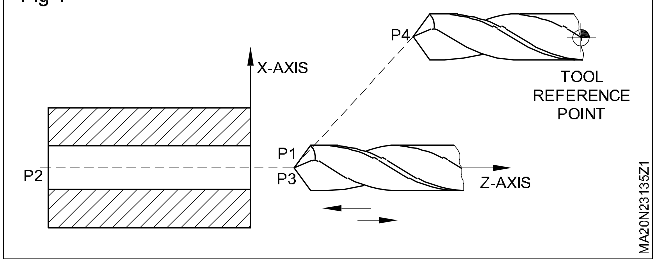
Fig 2



TASK 5: **Tool path for drilling operation (Fig 1)**

- P1 = position the tool at x zero and z 2 to 3 mm away from work zero in rapid movement.
- P2 = move the tool in z - direction to the required depth
- P3 = with draw the tool to start positions in rapid movement.
- P4 = send back the tool to reference point/tool change position.

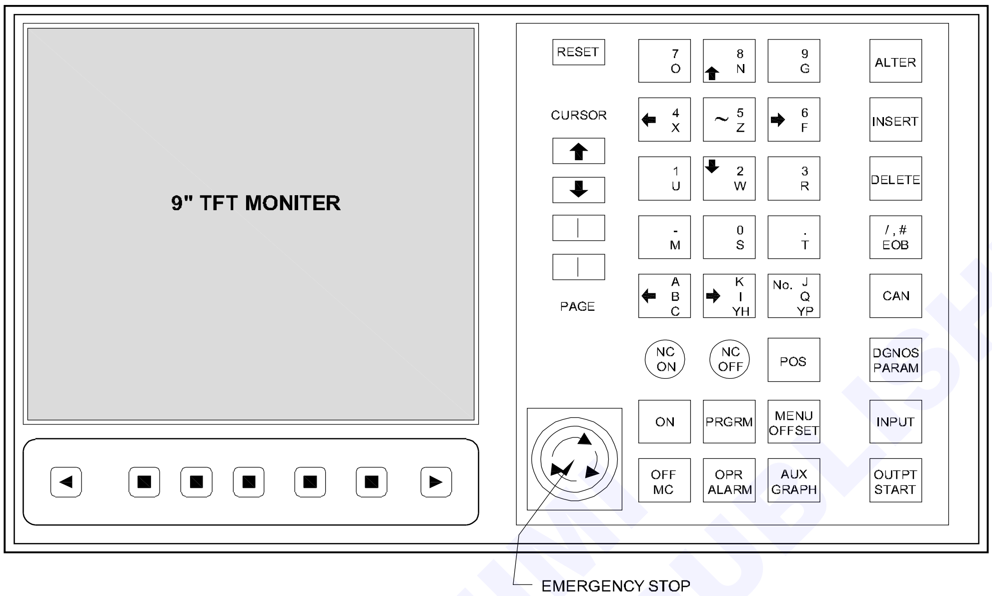
Fig 1



Identification of safety switches and interlocking of DIH modes

Objectives: At the end of this exercise you shall be able to
- **identify safety switches in CNC lathe**
- **identify inter locks used in CNC lathe.**

Fig 1



MA20N23136H1

Job sequence

1 **Trainer shall demonstrate the safety switches are provided in CNC lathe.**

2 **Identify and demonstrate the various interlocking system provided in CNC lathe.**

3 **Ask the trainees to identify, observe and record in the table 1.**

Table 1

Sl.No.	Condition	Operation	Observation
1	Edit key is OFF position	Enter the part program	......
2	Keep door in open position	Press cycle start	......
3	Chuck, open position	Start spindle rotation in MDI mode	......
4	Emergency stop switch in	Homing the axis pressed condition	......
5	Close the feed rate over ride	Move the axes switch to 'Zero' position	......
6	"Machine lock" switch in 'ON' position	Start the auto cycle to machine the job	......
7	Optional stop switch in 'OFF' position	Start the Auto cycle to machine job with various operation	......
8	Block skip switch in OFF position (part program with / blocks)	Start the auto cycle to machine job	

Get it verified by the trainer.

Identify common tool holder and insert shapes by ISO nomenclature

Objectives: At the end of this exercise you shall be able to
- **identify the tool depending upon the feed movement**
- **identify the clamping systems and design on inserts and tool holders**
- **identify the shapes of insert commonly used.**

Job Sequence

TASK 1: **Identify the hand of tools in Fig 1 and recondition Table 1**

Fig 1

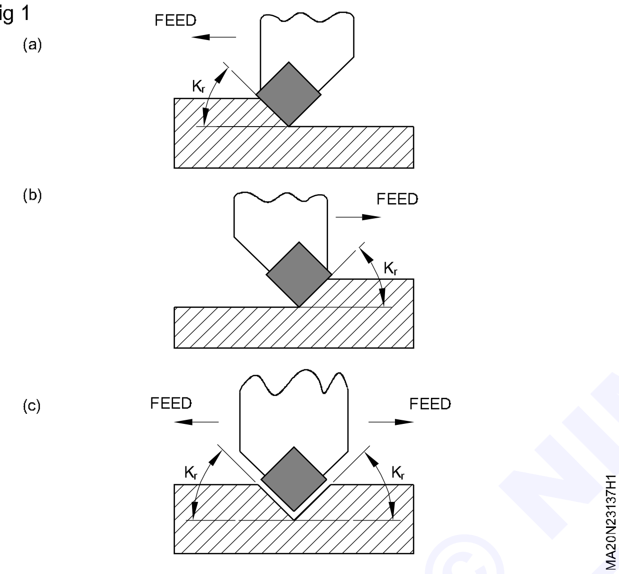


Table 1

	Hand of tool
a	
b	
c	

— — — — — — — —

TASK 2 : **Identify the clamping system shown in Fig 1 and record it in Table 2.**

Fig 1

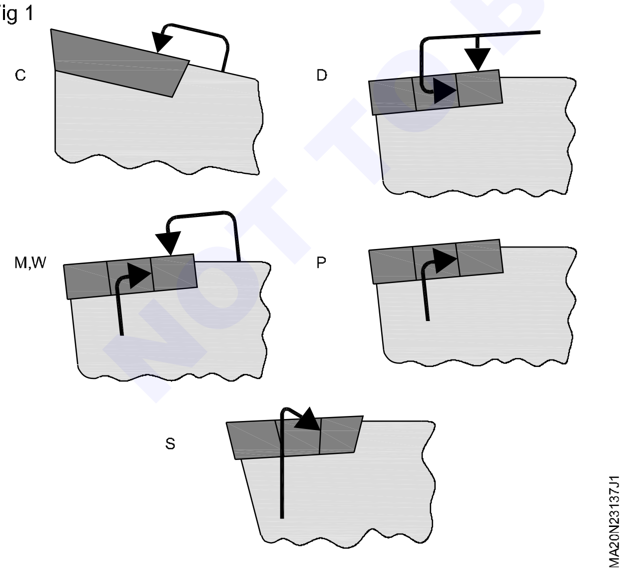


Table 2

	Clamping system
C	
D	
M.W	
P	
S	

— — — — — — — —

TASK 3: **Identify the clamping design shown in Fig 1 and record it in Table 3**

Fig 1

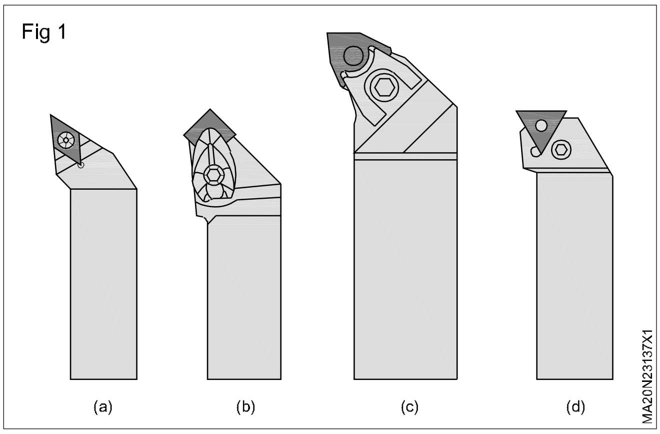


Table 3

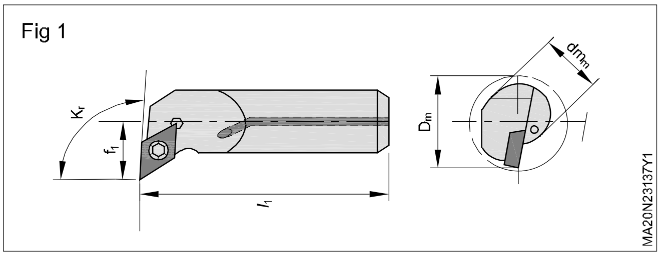
	Clamping design
a	
b	
c	
d	

— — — — — — — —

TASK 4: **Identify the tool shown in Fig 1 and record it.**

Name of tool...........

Fig 1



— — — — — — — —

TASK 5: **Identify the insert shape shown in Fig 1 and record it Table 4.**

Fig 1

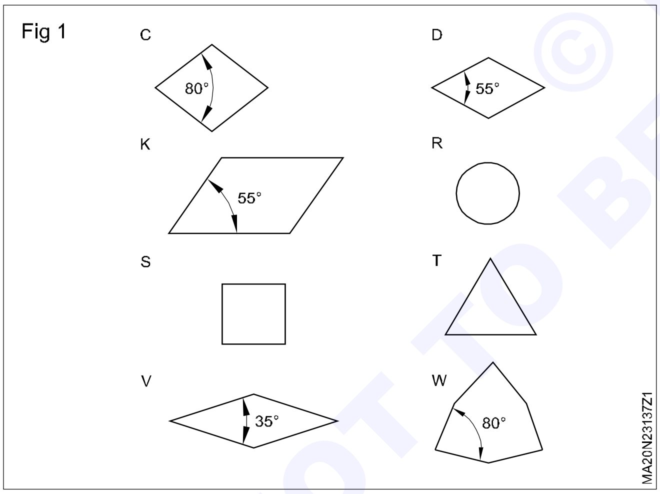


Table 4

	Insert shape
C	
D	
K	
R	
S	
T	
V	
W	

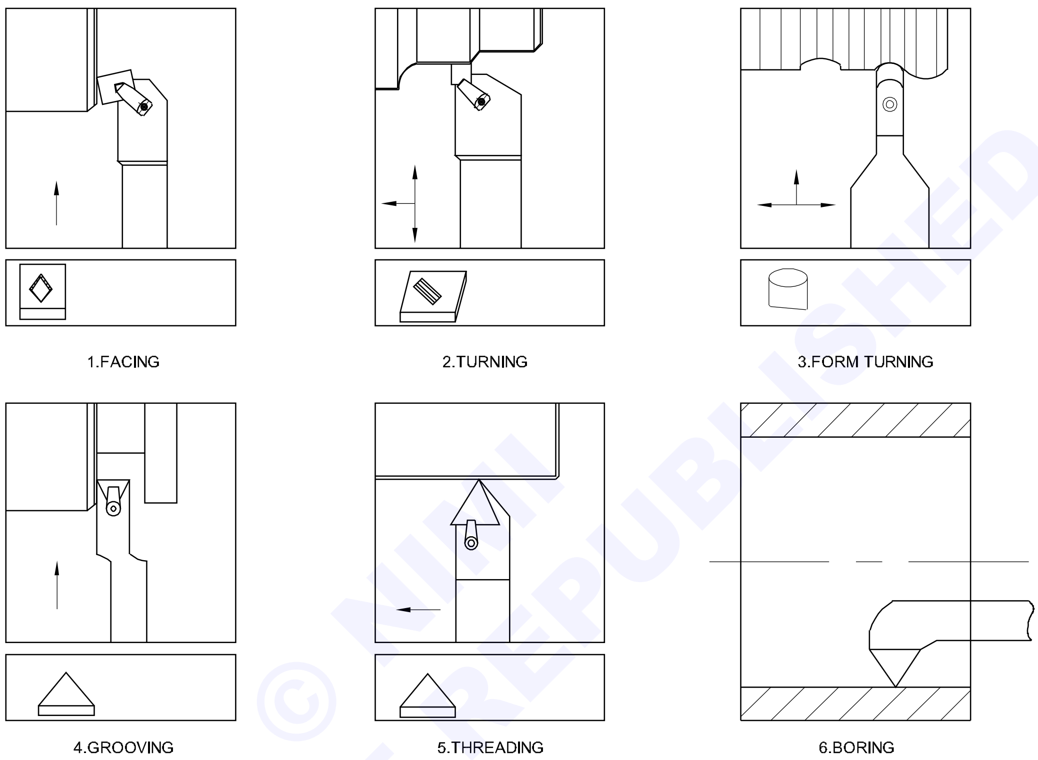
Note: Trainer shall display all the tool holders, inserts and explain the purpose and its technical names.

— — — — — — — —

Select cutting tool holder and insert for each operation

Objectives: At the end of this exercise you shall be able to
- **select suitable insert for various finish turning operation for machining alloy along steel**
- **select the corresponding tool holders for the operation.**

Fig 1



Note: trainer shall provide required catalogue and guide the trainees in selecting inserts and its tool holders.

Job Sequence

- Select material according to ISO from the catalogue S/NO 1to 6
- Identify the operation from the drawing and select the shape of the inserts
- Determine the machining condition
- Select the clamping system.
- Select the corresponding tool holders.

Skill Sequence

Selection turning tool from tool catalogues

Objective: This shall help you to

- **select turning insert and its tool holder for particular operation.**

First step

Define material and type of operation,

Define material according to ISO 'P' , 'M' and 'K' and identify the operation from the contents.

Second step

Define application and machining conditions.

Locate first choose of insert geometry and grade by application.

F- Finishing

M- Medium

R- Rough

Conditions

 - Good

 - Normal

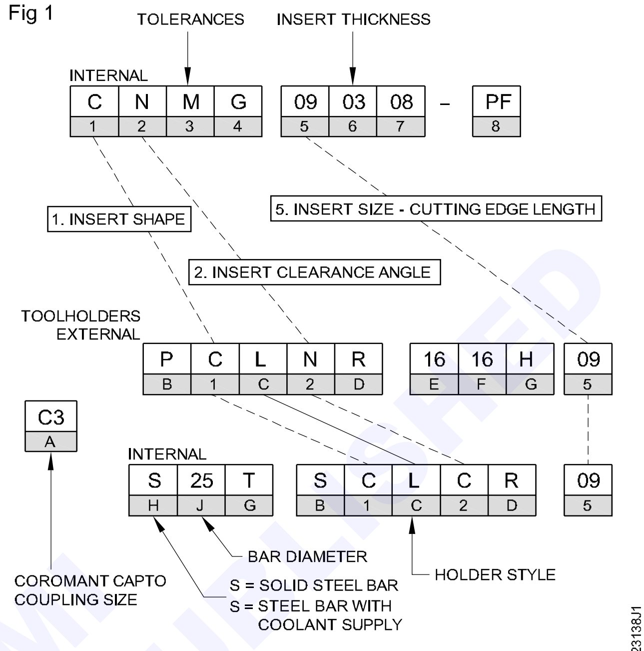
 - difficult

Third step

Choose insert with recommended cutting data.

- Select the insert from the ordering page and note down the speed, feed and depth of cut recommended.

Fig 1



TURNING TOOLS
CODE KEY FOR INSERTS AND TOOLHOLDERS

Fourth step

Choose tool holder

- Select the tool holder using the insert shape and size.

Fix inserts on the tool holder

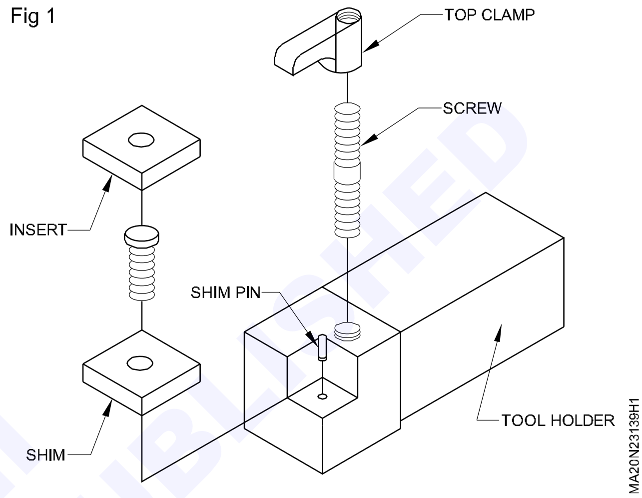
Objectives: At the end of this exercise you shall be able to
- **remove the insert from the tool holder**
- **fix the insert on the tool holder.**

Job Sequence

- Replace the insert. (Fig 1)
- Clean the insert and tool holder.
- Check the condition of the insert and shim.
- If damaged remove the insert from the tool holder with proper allenkey.
- Check the shim for damage. If it is damaged replace or index to next corner.
- Clean the seat of insert and shim.
- Place the shim in proper location.
- Place the insert with sharpe cutting edge or new one against the supporting points.
- Clamp the insert with proper allenkey.

Note: Pay full attention while removing or tightening - otherwise you may damage the screw.

Fig 1



Cutting tool material for various applications

Objective: At the end of this exercise you shall be able to
- **decide cutting tool material for various applications.**

Job Sequence

Write the suitable tool material for the job material mentioned in the table 1

Table 1

Sl No.	Job material	Tool material
1	Low carbon steel	
2	High carbon steel	
3	Stainless steel	
4	Grey cast iron	
5	Brass & Bronze	
6	Aluminium	
7	Aluminium casting	
8	Copper	
9	Gun metal	
10	Tool steel	

Note: Trainer shall provide any tool catalogue for the selection of cutting tool material in accordance with ISO.

Cutting parameters from tool manufacturer's catalogue, cutting speed and feed

Objective: At the end of this exercise you shall be able to
- **select the recommended cutting parameters by using catalogue.**

Job Sequence

Trainer shall collect catalogue from tool manufacturer and ask the trainees to fill the data in table 1 according job material.

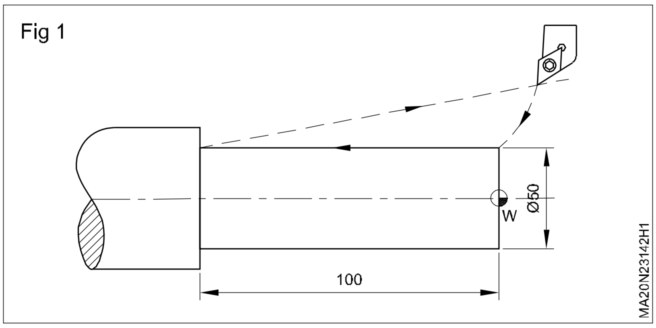
Table

	Speed / feed /depth of cut for general turning			
Part Material	**Cutting speed**	**Feedrate /mm/rev**		**Depth of cut mm**
	Vc m/min	**Rough**	**Finish**	**Finish** **0.4 R**
Mild steel				
High carbon steel				
Aluminium				
Copper				
Cast Iron				
Tool steel				

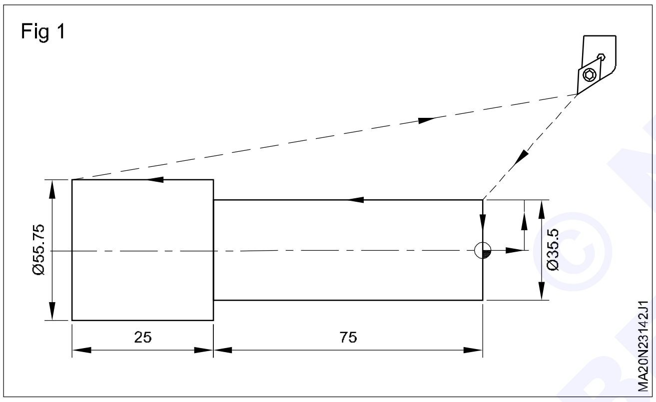
Linear interpolation and circular interpolation assignments and simulation on software

Objectives: At the end of this exercise you shall be able to
- **write CNC program with G00 & G01**
- **write CNC program with G02 & G03**
- **enter and verify the program in Simulator.**

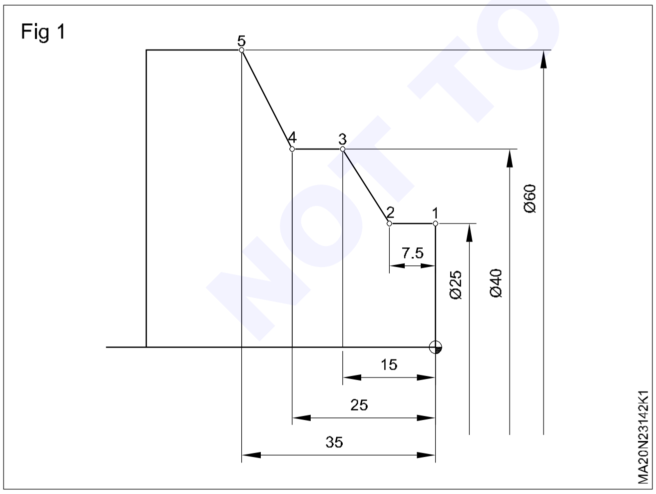
Task 1: Plain turning



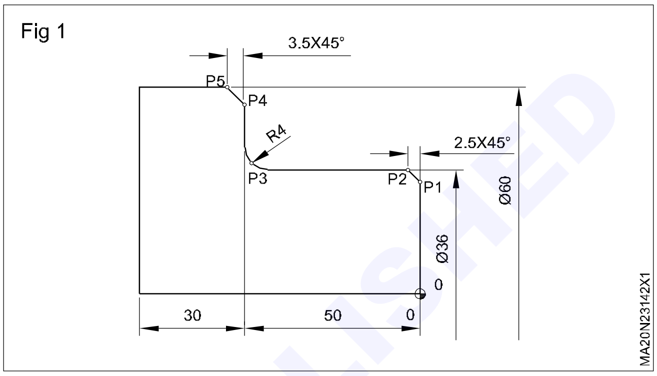
Task 2: Facing, plain turning and step turning



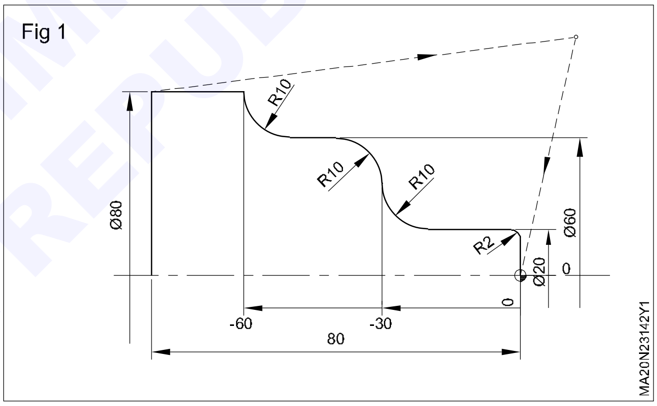
Task 3: Programming with linear interpolation



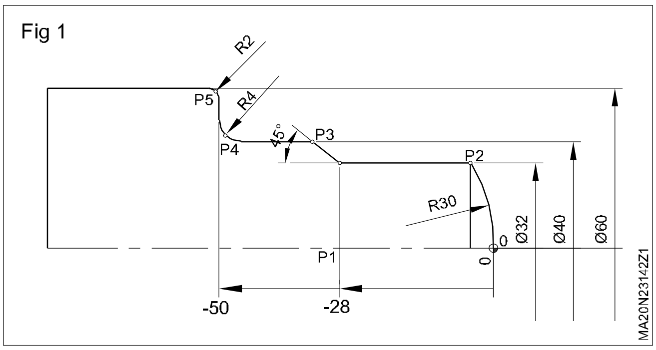
Task 4: Programming with G01, G02



Task 5: Programming with G01, G02 & G03



Task 6: Part program



Job Sequence

TASK 1: **CNC Programming for Plain turning - (Fanuc control)**

N5 G90, G55, G95;

N10 T0101, S500 M04;

N15 G00 X55.00 Z2.00;

N20 G01 X-0.1 Z0.00;

N25 G00 X-0.1 Z5.00;

N30 G00 X50 Z 5.00;

N35 G01 X50 Z-100.0;

N40 G00 X100 Z100.0;

N50 G28 G91 X0 Z0 T0100;

N55 M05;

N60 M30;

— — — — — — — —

TASK 2 & TASK 3: **Write the CNC Program in fanuc control**

— — — — — — — —

TASK 4 : **Programming with G01& G02**

N5 G90 G55 G95;

N10 T0404 S500 M04 ;

N15 G00 65 Z0.0;

N20 G01 X-0.1 Z0.0;

N25 G00 X-0.1 Z5.0;

N27 G00 X62.0 Z5.0;

N30 G01 X62.00 Z-80;

N35 G00 X64.00 Z2.00;

N40 G01 X31.00 Z0.00;

N45 G01 X36.00 Z-2.50;

N50 G01 X36.00 Z-46.00;

N55 G02 X44.00 Z-50 I4.0 K0.00;

N60 G01 X53.00 Z-50;

N65 G01 X60.00 Z-53.50;

N70 G01 X60.00 Z-80;

N75 G00 X100.00 Z-100.00;

N80 G28 G91 X0 Z0T0400 M05;

N85 M30;

— — — — — — — —

TASK 5 & TASK 6: **Write the CNC program using G01, G02, G03 with I, J and R**

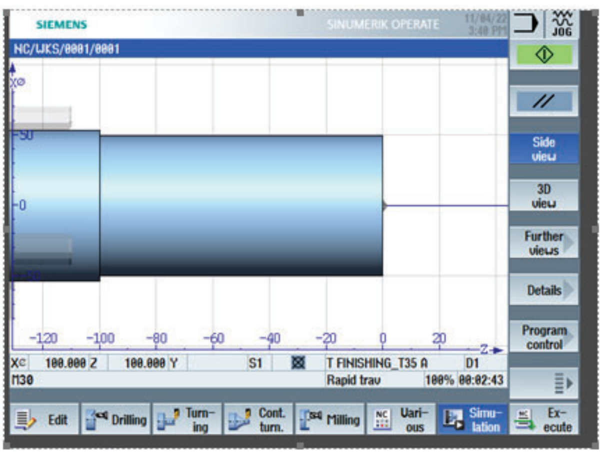
- Enter the CNC program task 1 to task 6 in CNC simulator
- Verity the program by simulating with CNC simulator.

— — — — — — — —

TASK 7 _-**SIENUMERIC PROGRAM IN SIMULATOR**

G90 G55 G95
WORKPIECE(,,,"CYLINDER",0,0,150,110,54)
T="FINISHING_T35 A"
S500 M04
G00 X55.00 Z2.0
G01 X-0.1 Z0.0 F0.1
G00 X-0.1 Z5.0
G00 X50.0 Z5.0
G01 X50.0 Z-100.0
G00 X100 Z100
M04
M30

Fig 1



— — — — — — — —

TASK 8 : **SINUMERIC PROGRAM IN CNC SIMULATOR**

N10 G90 G55 G95

N15 WORKPIECE(,,,"CYLINDER",0,0,150,110,60)

N20 T="FINISHING_T35 A"

N25 S500 M04

N30 G00 X65 Z0

N35 G01 X-0.1 Z0 F0.1

N40 G00 X-0.1 Z5

N45 G00 X62 Z5

N50 G01 X62 Z-80

N55 G00 X64 Z2

N60 G01 X31 Z0

N65 G01 X36 Z-2.5

N70 G01 X36 Z-46

N75 G02 X44 Z-50 I4 K0

N80 G01 X53 Z-50

N85 G01 X60 Z-53.5

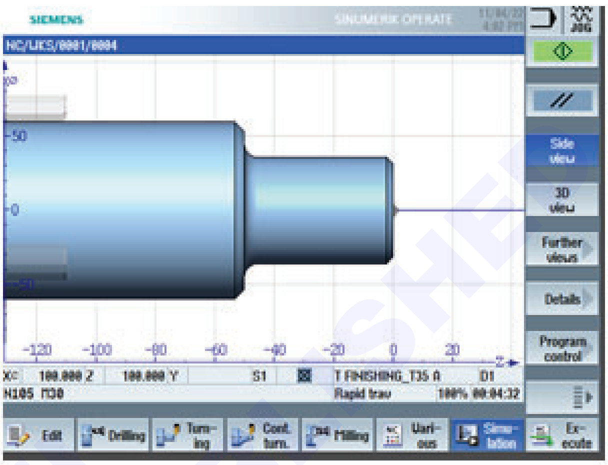
N90 G01 X60 Z-80

N95 G00 X100 Z 100

N100 M05

N105 M30

Fig 1



— — — — — — — —

Skill Sequence

Entering and simulatating program in sinutrain

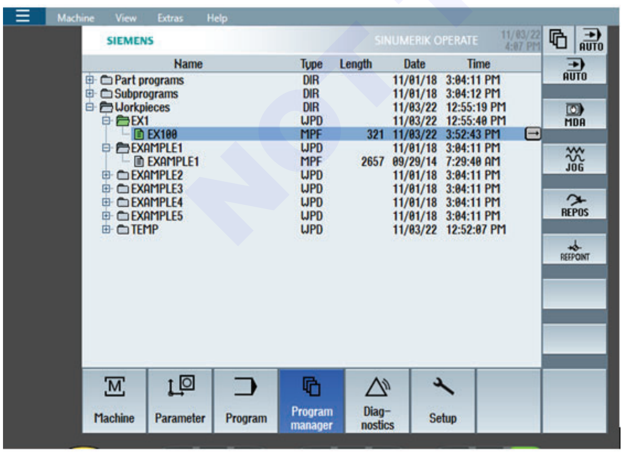
Objectives: This shall help you to

- **open the simutor**
- **enter new program**
- **verify the program by simulation method**

Open simulator

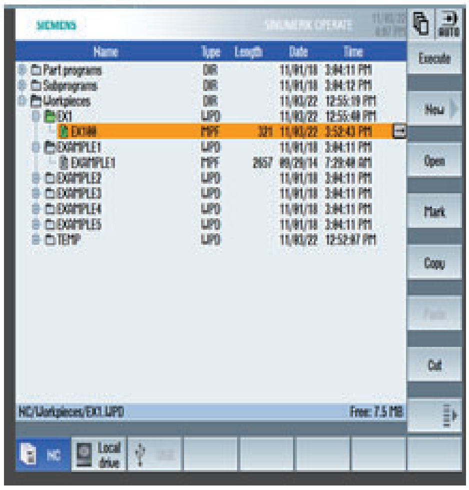
Press program manger (Fig1)

Fig 1



The screen will display as in Fig 2.

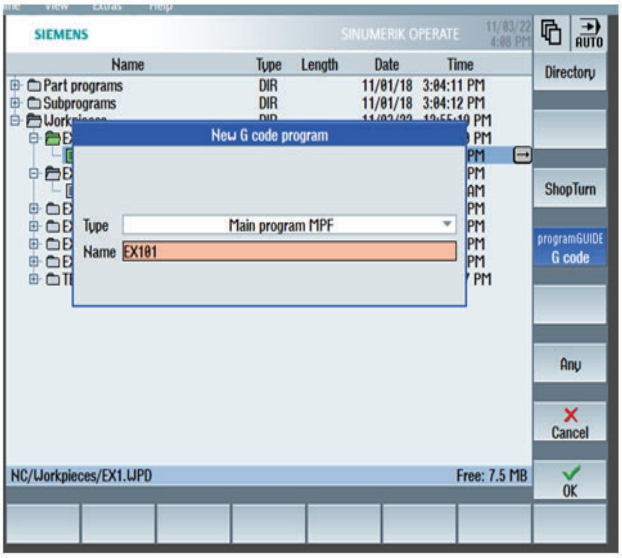
Fig 2



Press new

The screen will display as in Fig 3.

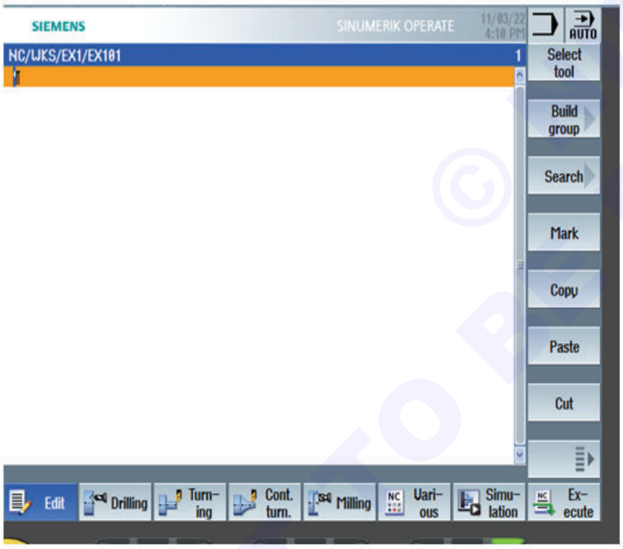
Fig 3



Enter the name of the program and press ok

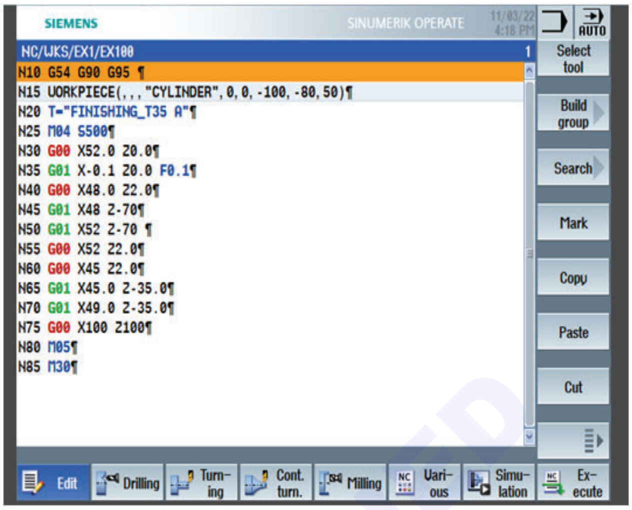
The screen will display as in Fig 4

Fig 4



Type the program using computer key board as in Fig 5

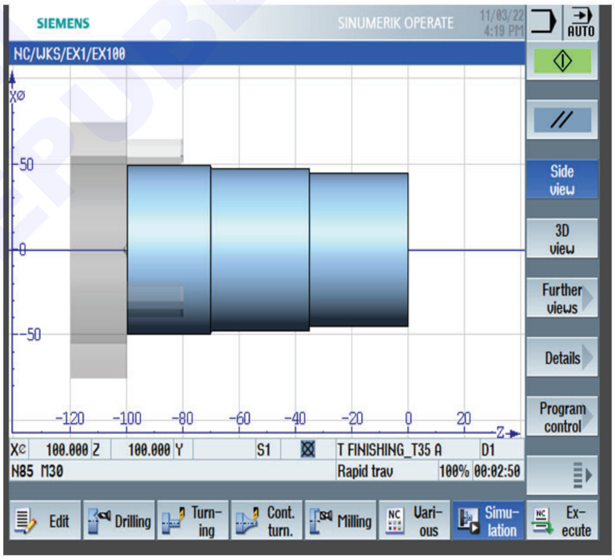
Fig 5



Press simulation as in Fig 6.

The screen will display the simulation.

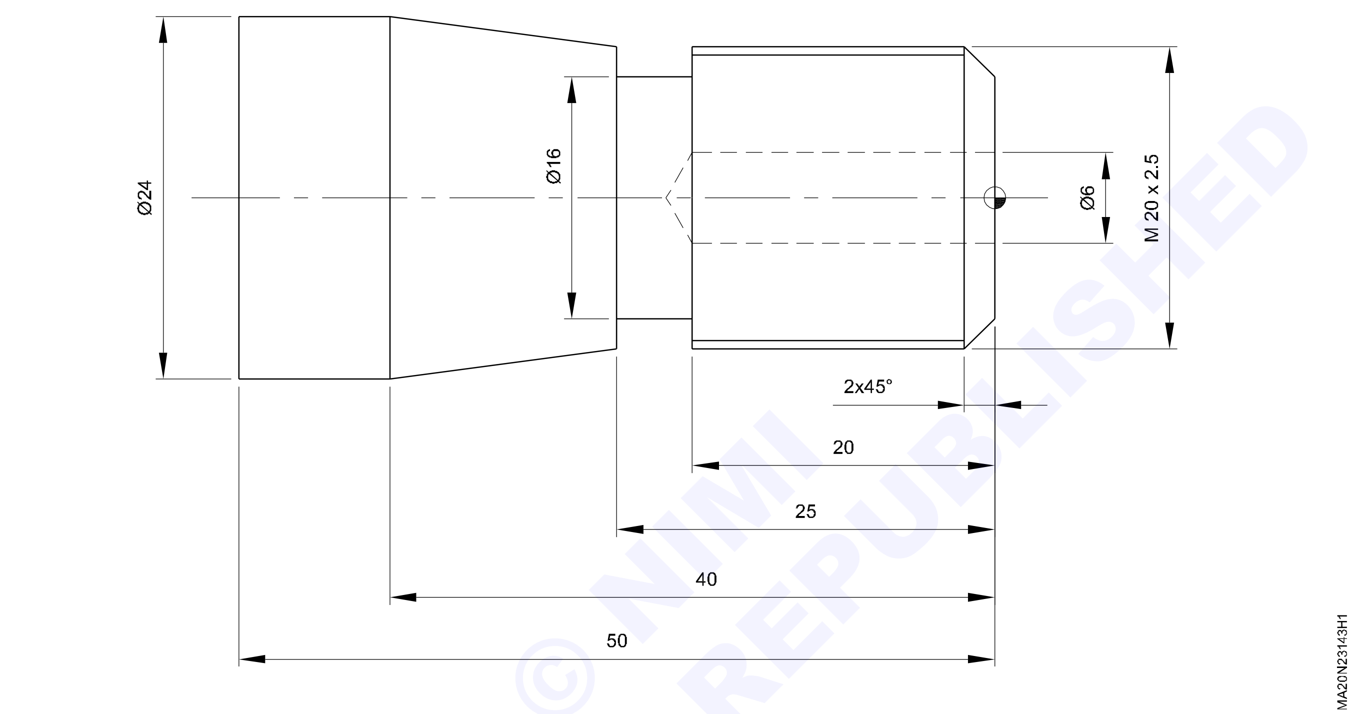
Fig 6



Write, simulate CNC part programming using simulator with canned cycles

Objectives : At the end of this exercise you shall be able to
- **switch on the CNC machine simulator**
- **write part program in fanuc control**
- **input the program & simulate.**

Fig 1



Job Sequence

- Write the part program in fanuc system.
- Check up with the instructor.
- Input the part program in the simulator.
- Run and check the correctness of the program using the simulator.

Control system

Fanuc - series oimate TD

Part program

```
O0009;
N1 ; (Turning)
G28    U0     W0;
G92    S2000  T0101 ;
G96    S200   M03 ;
G00    X30.0  Z5.0   M08 ;
Z0.0 ;
G01    X-1.0  F0.1 ;
G00    X28.0  Z2.0;
G71    U1.0   R1.0 ; ;
G71    P10    Q20    U0.0   W0.0   F0.1 ;
N10    G01    Z0.0   F0.1 ;
X16.0 ;
X20.0  Z-2.0 ;
Z-25.0 ;
X24.0  Z-40.0;
N20    G01    Z-50.0 F0.1 ;
G00    X30.0  Z5.0   M09 ;
G28    U0     W0 ;
M05 ;
M01 :
N2 ; (Centre drill)
G28    U0     W0 ;
T0202 ;
G97    S800   M04 ;
G00    X0.0   Z5.0   M08 ;
G01    Z-6.0  F0.08 ;
G00    Z5.0   M09 ;
G28    U0     W0 ;
M05 ;
M01 ;
N3 ; (∅6 Drill)
G28    U0     W0 ;
T0404 ;
G97    S1000  M04 ;
G00    X0.0   Z5.0   M08 ;
G74    R2.0 ;
G74    Z-20.0 Q10000 F0.06 ;
G00    Z5.0   M09 ;
G28    U0     W0 ;
M05 ;
M01 ;
N4 ; (OD Grooving 3 mm width)
G28    U0     W0 ;
T0303 ;
G97    S600   M03 ;
G00    X22.0  Z5.0   M08 ;
G01    Z-23.0 F0.1 ;
G75    R2.0 ;
G75    X16.0  Z-25.0 P500   Q2000  F0.06 ;
G00    X25.0 ;
Z5.0   M09 ;
G28    U0     W0 ;
M05 ;
M01 ;
N5 ; (Threading)
G28    U0     W0 ;
T0505 ;
G97    S600   M04 ;
G00    X22.0  Z5.0   M08 ;
G01    Z3.0   F0.1 ;
G76    P030060 Q150  R20 ;
G76    X16.755 Z-22.0 P1622 Q300  F2.5 ;
G00    X25.0  Z5.0   M09 ;
G28    U0     W0 ;
M05 ;
M01 ;
M30 ;
%
```

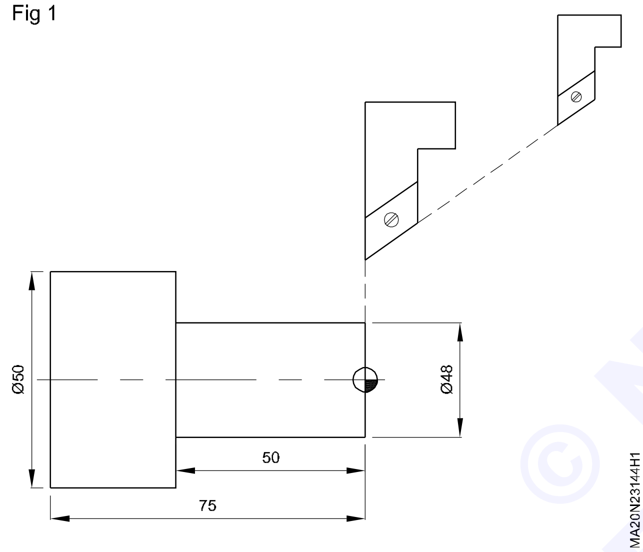
Avoiding collisions caused by program errors, by making deliberate program errors and verification/simulation software

Objectives: At the end of this exercise you shall be able to
- **find various causes for collisions**
- **rectification, rectify the collision.**

Job sequence

- Plain turning and Facing operation program is written.
- Verify the program on simulation software.
- Identify the errors if any.
- Rectify the errors.

Fig 1



Wrong program

```
O5555
N1    G28    U0       W0;
N2    G97    S1000    M03;
N3    T0101;
N4    G00    X60.0    Z0.0    M08;
N5    G00    X0.0     F0.1;
N6    G00    X48.0    Z2.0;
N7    G00    Z-50.0   F0.1;
N8    G00    X60.0    Z0.0    M09;
N9    G28    U0       W0;
N10   M05;
N11   M30;
```

Correct program

```
      O5555;
N1    G28    U0       W0;
N2    G97    S1000    M03;
N3    T0101;
N4    G00    X60.0    Z0.0    M08;
N5    G01    X0.0     F0.1;
N6    G00    X48.0    Z2.0;
N7    G01    Z-50.0   F0.1;
N8    G00    X60.0    Z0.0    M09;
N9    G28    U0       W0;
N10   M05;
N11   M30;
```

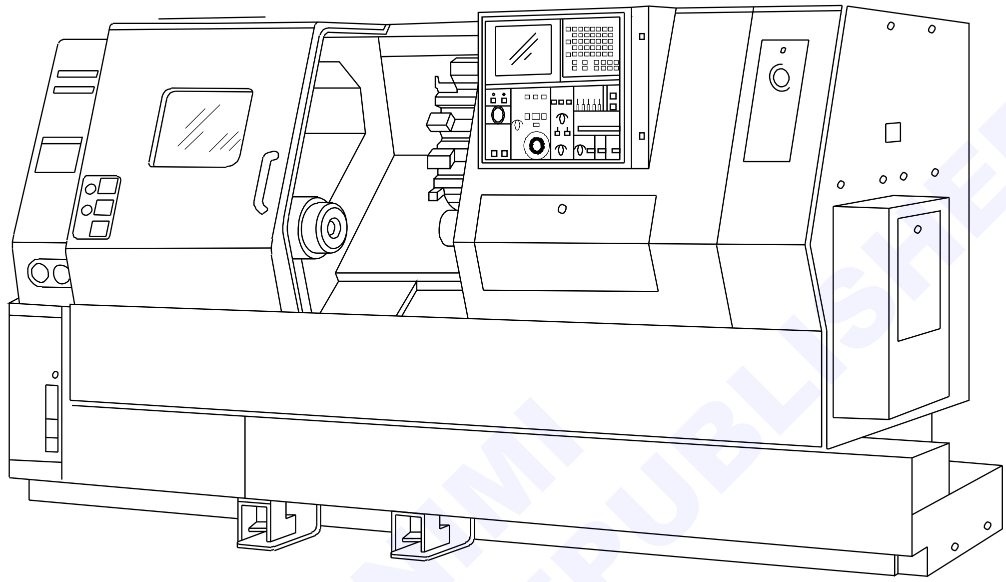
Collisions and rectifications

Sl No	Identification of errors	Effects	Rectification of errors
1	G00 X0.0 F0.1;	Will damage turret & Tool holder, Machine Alignment, and affect Accuracy	G01 X0.0 F0.1;
2	G00 Z-50.0 F0.1;	- do -	G01 Z-50.0 F0.1;
3	Operator alter the programme in wrong format.	- do -	Check the programme and correct it.
4	Improper holding the job and tool.	- do -	Check the reason for inproper holding and correct it.
5	Wrong selection and mismatch the tool for operation	- do -	Select proper tool for the particular operation.
6	Wrong offset for Job and Tool	- do -	Check offset for Job and Tool.

Conduct a preliminary check of readiness of the CNC turning centre

Objectives: At the end of this exercise you shall be able to
- **check cleanliness of the machine**
- **check oil levels**
- **check correct working of lubrication system**
- **send the turret to machine reference point.**

Fig 1



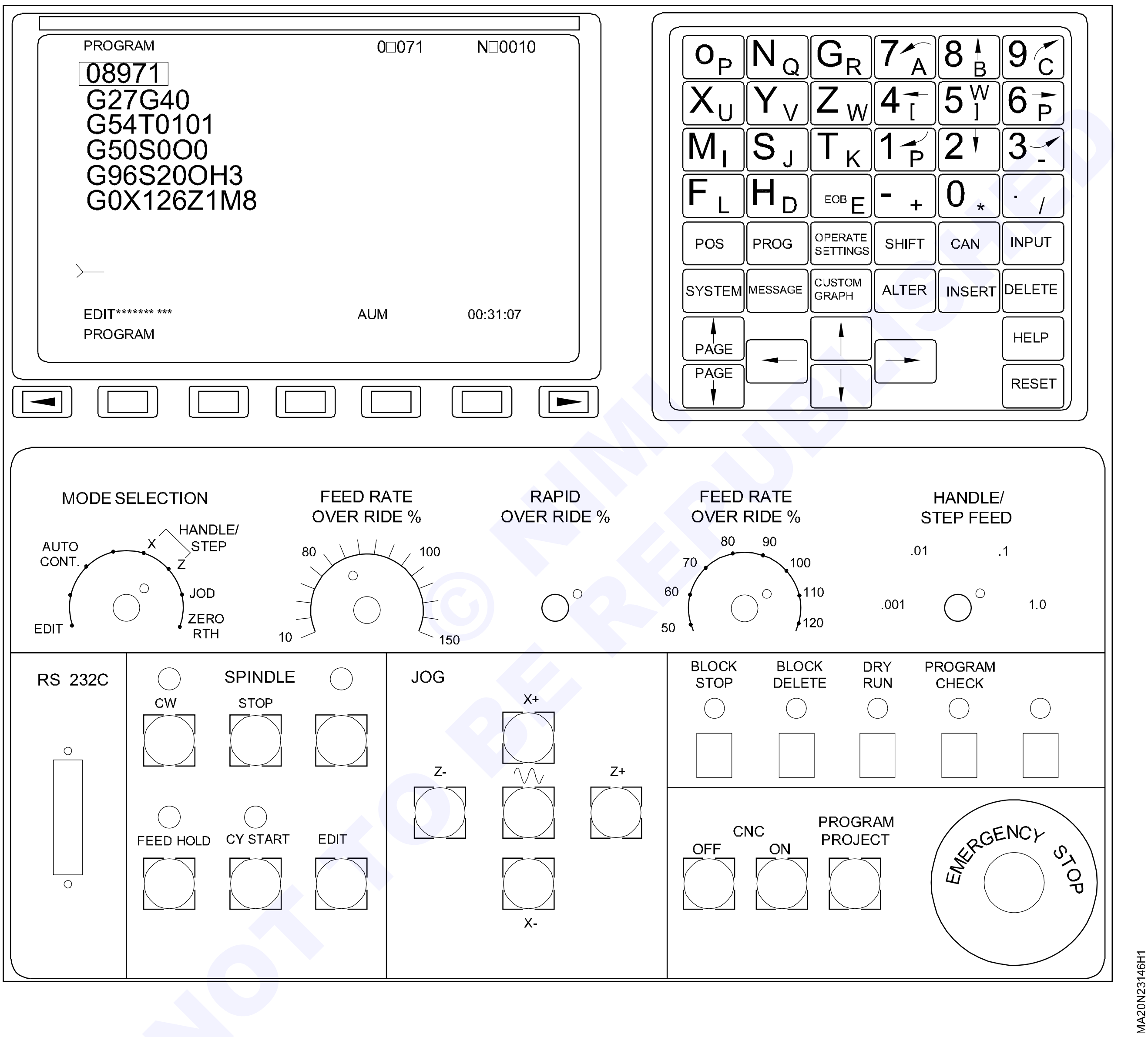
Job sequence

- Ensure the cleanliness of machine.
- Use banian waste to clean.
- Ensure there is no oil spill around the machine.
- Switch 'ON' the Machine.
- Move the tool in JOG mode to a safe place.
- Send the turret to machine Reference point.
- Check the lubrication oil level and ensure it is within the acceptable level.
- Check hydraulic oil level.
- Check coolant oil level.
- Check correct working of lubrication system by manual operation.

Starting the machine, do homing on CNC simulator and machine

Objectives: At the end of this exercise you shall be able to
- **perform homing in CNC simulator**
- **perform homing on CNC machine using JOG and MPG modes.**

Fig 1



Job sequence

TASK 1: **"Machine ON" and Homing on simulator**

Switch ON (System)

↓

Select the simulator software for turning

↓

Select the fanuc oit CNC system

↓

Release emergency button

↓

Unlock the edit key using cursor

↓

Give little movement by pressing X -, Z -

↓

Pres reference switch

↓

Press 'X' button to move X axis to the origin

↓

Press 'Y' button to move 'Y' axis to the origin

↓

CNC machine simulator 'Zero Ref' homing reached

— — — — — — — —

TASK 2: **"Machine ON" and Homing on machine**

Practice starting referencing and manual mode operations till you become familiar.

- Switch on the main power connection to the machine.
- Switch on the voltage stabilizer.
- Switch on the isolation switch.
- Press "NC ON" push button.
- Wait for the screen to indicate module setting status. (Fig 1)

Fig 1

D6A1-01
SLOT 01
SLOT 02

MA20N23146X1

- Display of software configuration (Fig 2)

Fig 2

D6A1-01
OMM-
PMC

MA20N23146X2

- Now press " CONTROL ON" push button LCD display will be on.
- The machines will be in "MDI" mode by default.
- Note "release EMG switch, if pressed and press control on
- Press "reset" switch on the control panel.
- Press "jog" mode switch.
- Press X-
- Press Z-
- Press "reference" mode switch
- Select X axis and press + button.
- Wait till the display indicate the completion of X axis referencing.
- X = 260 mm
- Select Z axis press + button wait till the display indicate the completion of Z axis referencing.
- Z = 450 mm.

The display and the X and Z value may be different machine.

Skill Sequence

Jog incremental and MDI mode operation

Objectives: This shall help you to
- **operate in JOG mode**
- **operate in (manual puls generator)**
- **operate in manual data input mode.**

MDI MODE operation

- Pictorial representation of JOG mode operation is shown in Fig 1.
- Pictorial representation of incremental mode operation is shown in Fig 2.
- Set mode switch to MDI selection
- Select program soft key the new empty screen appear.
- Enter G0 G91 X 100.0. Then press insert button
- Press the Cycle start button.
- The Axis X will move 100 mm in (+) direction from the previous tool position
- Repeat the step and give X-100.0 then
- The Axis X will move 100mm in (-) direction
- Now the tool will reach the programmed position.

Fig 1

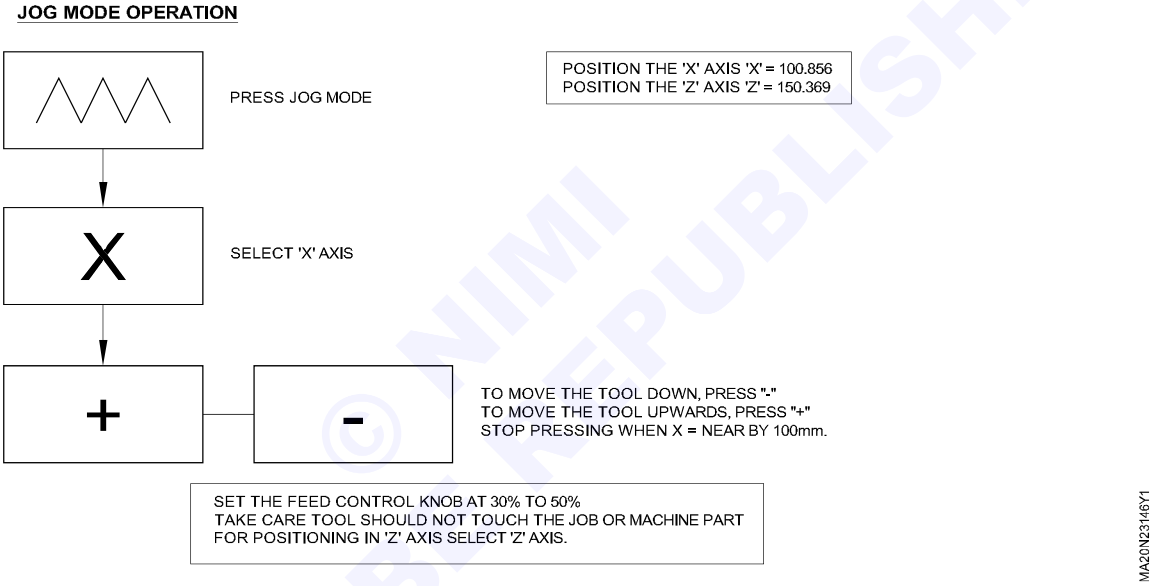
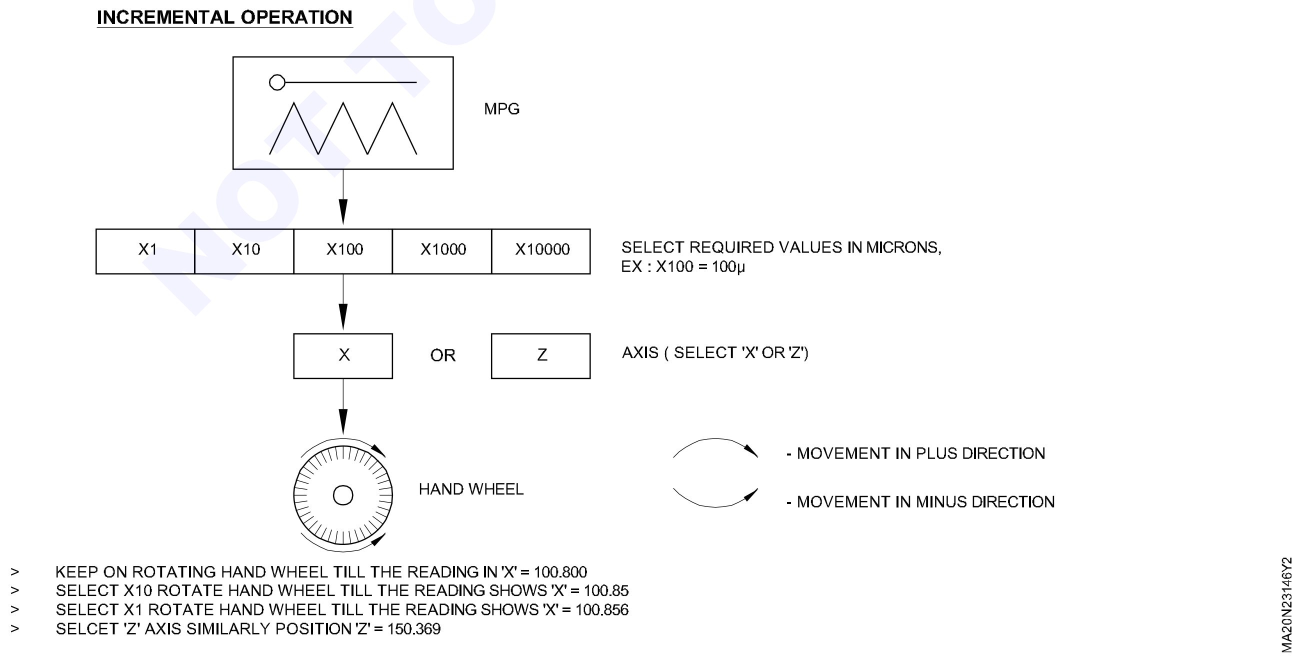
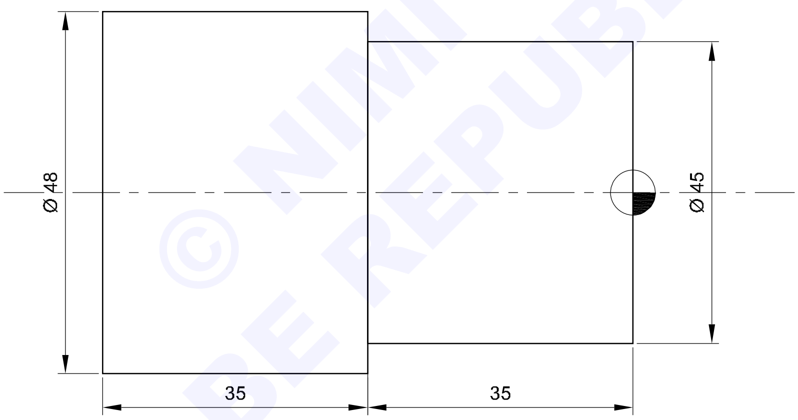


Fig 2



Facing and turning program

Objectives: At the end of this exercise you shall be able to
- **prepare CNC program for the given drawing**
- **enter the program in CNC simulator using edit mode**
- **verify the program by simulation on CNC simulator.**



01		-		-	-	2.3.147
NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.

SCALE 1:1	FACING AND TURNING PROGRAM	DEVIATIONS	TIME :
		CODE NO. MA20N23147E1	

Job Sequence

- Write the CNC program for facing operation
- Write the CNC program for plain turning operation.
- Write the CNC program for step turning
- Enter the program in CNC simulator using edit mode
- Verify the program by simulation in simulator

Program (facing and turning)

03001 - program number

N5 G90 G55 G95; - preparatory functions

N10 T0505 M04; - Tool change with spindle on ccw

N15 G00 X52.00 Z0.00; - Positioning for facing

N20 G01 X-0.1 Z0.00 F0.1;

M25 G00 X48.00 Z5.00;

N30 G01 X48.00 Z -70.00;

N35 G01 X52.00 Z-70.00;

N40 G00 X52.00 Z2.00;

N45 G00 X45.00 Z2.00;

N50 G01 X45.00 Z - 35.00;

N55 G01 X49.00 Z 35.00

N60 G00 X100 Z100;

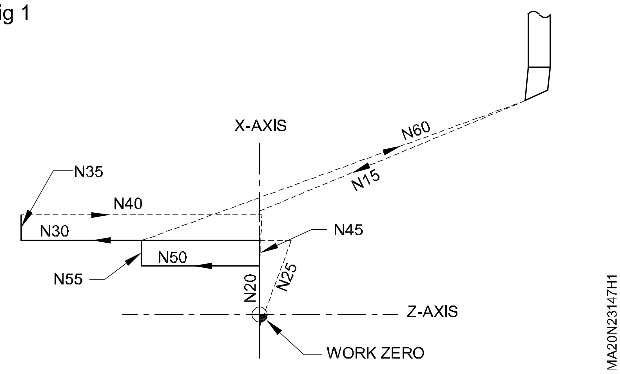
N65 G28 G91 X0.00 Y0.00 T0500 M05;

N70 M90;

N75 M30;

Tool path shown in Fig 1.

Fig 1



SIEMENS CNC SIMILATION PROGRAM,

N10 G54 G90 G95

N15 WORKPIECE(,,, "SYLINDER", 0,0,-100,-80,50)

N20 T= "FINISHING_T35A"

N25 M04 S500

N30 G00 X52.0 Z0.0

N35 G01 X-0.1 Z0.0 F0.1

N40 G00 X48.0 Z2.0

N45 G01 X48 Z-70

N50 G01 X52 Z-70

N55 G00 X52 Z2.0

N60 G00X45 Z2.0

N65 G01X45.0Z-35.0

N70 G01 X49.0 Z -35.0

N75 G00 X100 Z100

N80 M05

N85 M30

Skill Sequence

Enter CNC program in edit mode

Objective: This shall help you to
- **enter the programme in fanue control.**

Fanuc control

Go to JOG mode

↓

Press part program

↓

Press edit

↓

Enter % and new program number of job

↓

Press Select program

↓

(blank space open for entry of program)

↓

Write program using numerical key and edit key

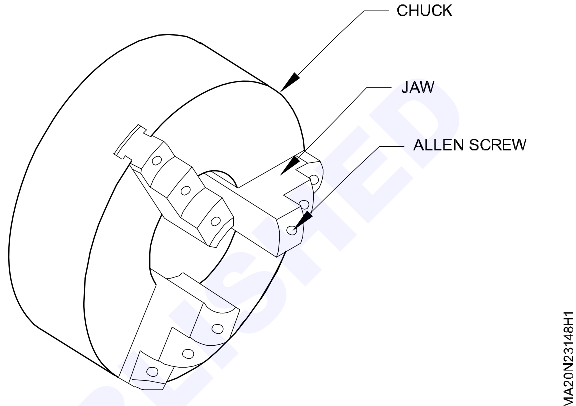
Mounting Jaws to suit the part holding area on CNC machine

Objectives : At the end of this exercise you shall be able to
- **remove the jaw and reload it**
- **check the concentricity with dial test indicator.**

Job Sequence

- Loosen allen screws of all the three jaws
- Count the number of serration radially outwards
- Leave equal number of serration (rack) outwards and hold jaws firmly
- Ensure the space slightly higher than the diameter of job
- Tighten the allen screw using suitable allen key.
- Check the clamping of job.

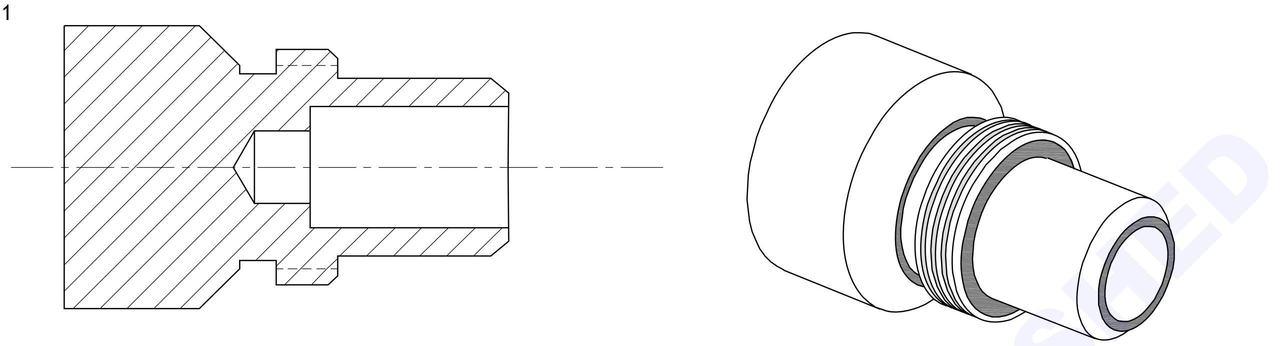
Fig 1



Mounting tools on the turret according to part & process required

Objectives : At the end of this exercise you shall be able to
- **prepare tooling sheet as per part drawing**
- **select suitable insert and its holder**
- **assemble the tool assembly in turret.**

Fig 1



Job Sequence

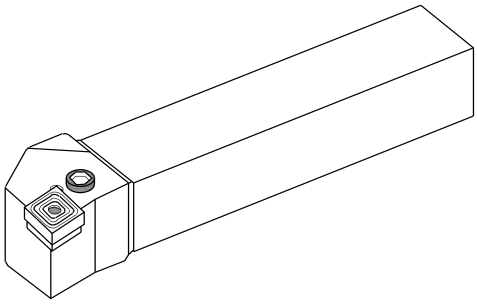
- Read the job drawing and list out the turning operation and arrange the operation simple to complex
- Prepare the tooling sheet as per the operation including insert and tool holder example as in chart 1

CHART 1

PART NO		DATE	
PART NAME		PROGRAMMER:	
MACHINE: CNC LATHE		MATERIAL / ALUMINUM	
SEQ	**OPERATION**	**TOOL**	**STATION**
1	FACING (ROUGH)	CNMG-432 INSERT, 30 GRADE DCLN HOLDER, -5°, RH, 1" X 1"X6" LONG	1
2	O.D TURNING (ROUGH)	CNMG-432 INSERT, 30 GRADE DCLN HOLDER, -5°, RH, 1"X1"X6" LONG	1
3	O.D TURNING (FINISH)	VNMG -432 INSERT DVJN HOLDER,-3°, RH, 1"X1"X6" LONG	2
4	O.D GROOVING	GROOVING INSERT, SEAT SIZE 3, CUTTING WIDTH 0.125" MIN TO 0.187" MAX CUTOFF/GROOVING HOLDER, SEAT SIZE 3 75"X. 75"X 5" LONG	3
5	O.D THREADING	TRIANGLE LAY DOWN INSERT , 60° 8 TPI EXTERNAL THREAD HOLDER, RH , 75" x75" X4" L	4
6	DRILLING	5/8 DIAMETER HSS TWIST DRILL NUMBER 35 DOVETAIL CHUCH HOLDER	5
7	BORING (ROUGH)	DPGT-211 55° INSERT SDUP HOLDER, STEEL, RH, ∅3/8" SHANK X 6" LONG, 0.6" MIN BORE DIAMETER	6
8	BORING (FINISH)	DPGT-211 55° INSERT SDUP HOLDER, CARBIDE, RH, ∅3/8 SHANK X6" LONG, 0.6" MIN BORE DIAMETER	7

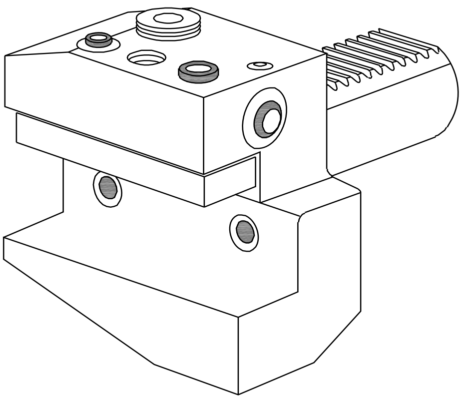
Fix insert in the appropriate tool holder (Fig 1)

Fig 1



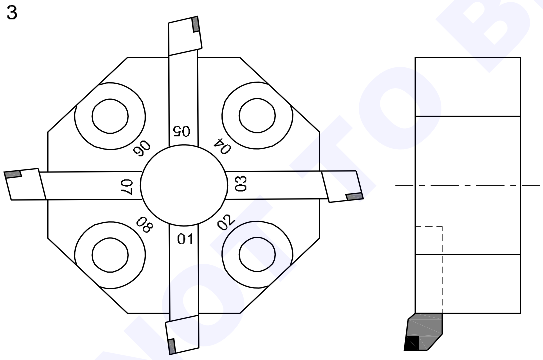
- Fix tool holder shank in respective tool holder for turret supplied along with the machine. (Fig 2)

Fig 2



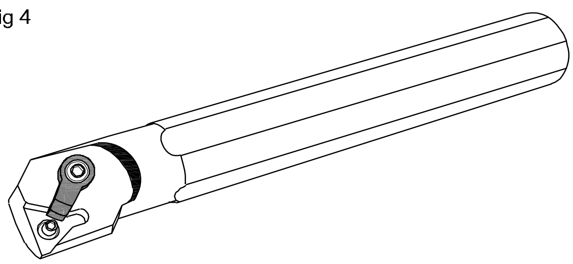
- Insert the turret tool holder assemble in the turret, alining rack matching with internal rack in turret and butting against the turret.
- Clamp the tool holder assembly with suitable allenkey.
- Similary assemble and mount the external turning tool in turret as shown in Fig 3, asper tooling sheet.

Fig 3



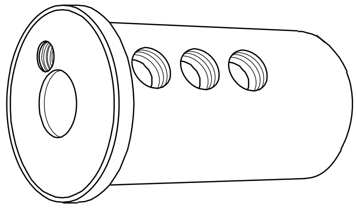
- For internal boring operation
- Select appropriate boring insert and it box bar (Fig 4)

Fig 4



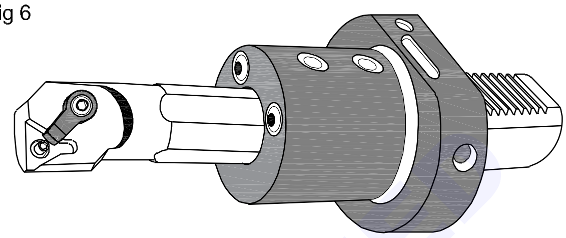
- Select suitable sleeve(Fig 5) and tighten it

Fig 5



- Insert the boring bar in Fig 6 and tighten it

Fig 6



- Mount the assembly in the turret
- For centre drilling and drilling operation select correct size tool asper drawing assemble Fig 7 in turret and Fig 8 in holder and Fig 9 in drill chuck.

Fig 7

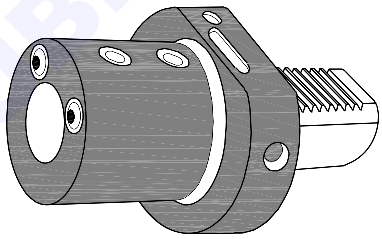


Fig 8

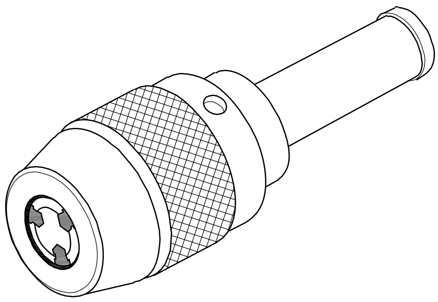
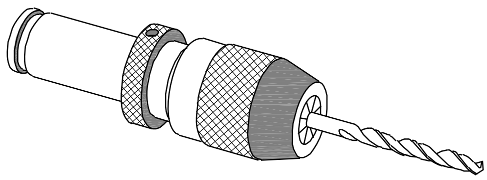


Fig 9



Perform work and tool setting

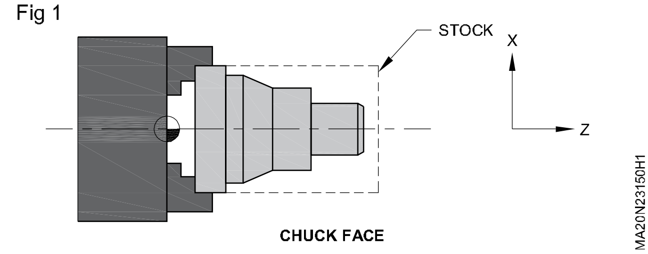
Objectives : At the end of this exercise you shall be able to
- **set the work co-ordinate system**
- **set the tool with respect to the work co-ordinate system**
- **set the live tool datum.**

PROCEDURE

TASK 1: **Work co-ordinate setting on the face of chuck (Fig1)**

- Set the work piece zero at the chuck face

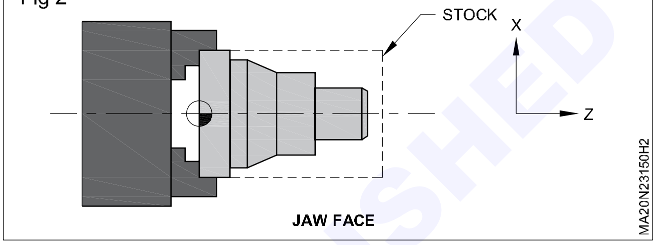
Fig 1



> **The work piece will be in 1st quardrant that is z and x values in positive. Generally fixtures are fitted on the face of the jaw.**

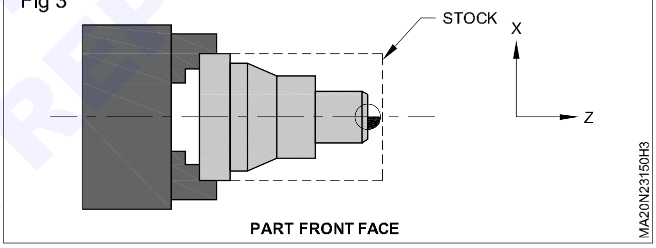
- Work co-ordinate on face of jaw (Fig 2)
- Set the work piece on the face of the jaw as shown in Fig 2.
- In this case the x zero and z zero is work piece seating side is/or in the face of jaw

Fig 2



- Work piece coordinate on front face (Fig 3)
- Set the work piece zero as shown in Fig 3
- In this case the work piece in second quardrant, that is x is positive z axis is negative

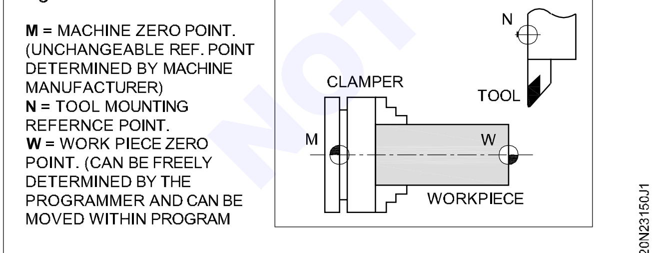
Fig 3



TASK 2 : **Tool reference point and commanding point**

- Turret reference point in relation to the work co-ordinate system is shown in Fig 1

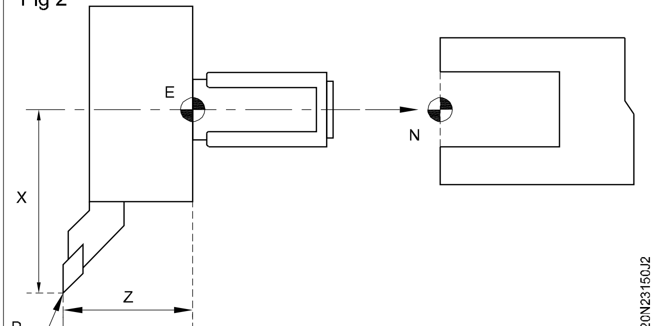
Fig 1



REFERENCE POINTS

- Tool reference point in relation to the turret reference point is shown in Fig 2
- X=tool length in x axis from reference point.
- Z=tool length in z axis from reference point.
- P =commanding point on tool in relation to the work coordinate system

Fig 2



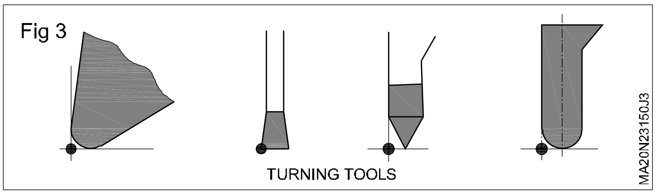
TOOL REFERENCE POINTS

- Measure x and z distance and enter in tool offset page, then 'p' will become commanding point

> **Measurement of tool offset differ control to control, the above if methods is sinumeric control.**

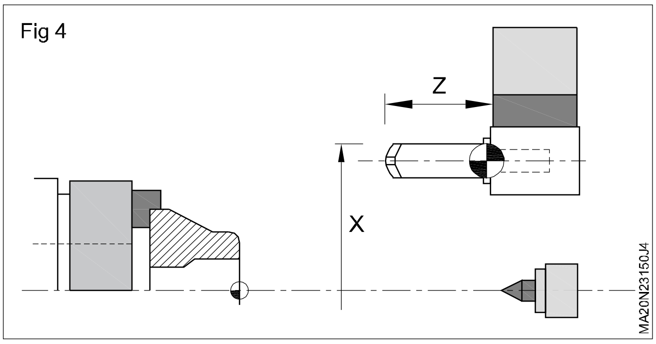
- For various tool the commanding point are shown in Fig 3

Fig 3



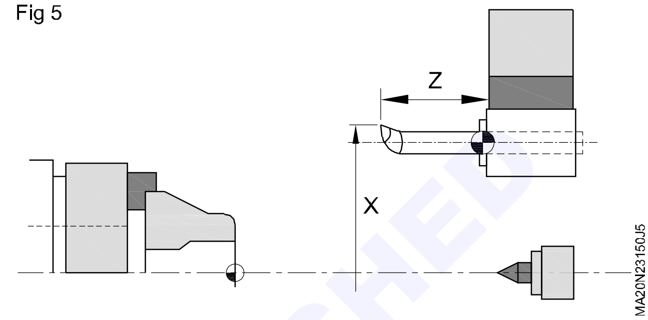
- For tools such as drill and other point to point tool used in milling or turning the reference point is always the extreme tip of the tool measured along z axis . x will always zero (Fig 4)

Fig 4



- Measure the tool centre distance from the machine centre in x axis and that will be x axis tool offset.
- Measure the distance from the reference point and the tool tip in z and that is tool offset in z axis
- For boring tool measure the distance from machine centre to the boring to tool tip that is x axis tool offset
- Similarly measure the length from the reference point to tool the in z direction. That is the tool offset in z direction as shown in Fig 5

Fig 5



— — — — — — — —

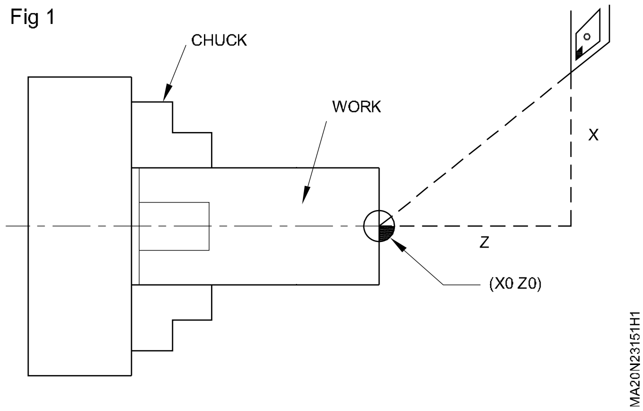
CNC turning centre operation in various modes (JOG, MDI, MPG & AUTO Mode)

Objectives : At the end of this exercise you shall be able to
- **measurement work offset values in z and x axis**
- **measure the tool offset values in x and z axis.**

Job Sequence

TASK 1: **Measurement of work offset (Fig1)**

Fig 1



Ensure the work secured firmly in chuck

Index the tool in MDI mode with tool offset cancel and set tool offset X0,Z0, and tool type

Switch 'ON' spindle

Carry out slight facing of the job

After the finish cut move the tool back in X direction only

Now switch off spindle

Go to Tool off set mode

Press GEOM soft key and position the cursor using cursor movement button and select the offset number G54

Enter the Z-axis valveZ0.0

Press soft key

Now rotate the spindle in appropriate direction and machine the outside diameter ('OD')

Do not disturb X-axis

Take tool away in Z-direction only

Stop the spindle

Measure the outer diameter of the job using micrometer

Go to OFFSET soft key

Press GEOM soft key

Position the cursor to the required work offset number and enter the measured value. (eg: X32.62)

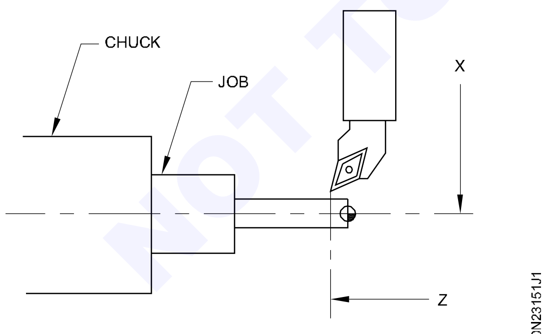
Press soft key

Note: The tool used for measuring work off set the tool offset is zero in X and Z direction.

— — — — — — — —

TASK 2: **Tool offset measurement (Fig 1) in fanuc control**

Fig 1



TOOL OFFSET METHOD

X axis tool offset method

Reference tool is T01 and offset is zero in X and Z axis.

Clamp job in chuck.

Select MDI mode. Press in MDI prog-screen.

Enter tool number: T0200 (Turning tool).

Press insert button, Press cycle start button.

Tool cutting edge position with spindle ON CW or CCW in MDI mode.

Enter MO3 SI500 Press reset button, Press cycle start button.

To select jog mode or MPG mode to move x and z axis.

Touch the job in x axis just clean OD turning to ensure no disturbance in x axis. (Fig 1)

Measure the outside diameter.

Using cursor in geomentry screen select Tool no : 2 x axis select.

Enter job diameter

Ex: x28.62 Press measure button in soft key

Now tool cutting edge in job centre is OK.

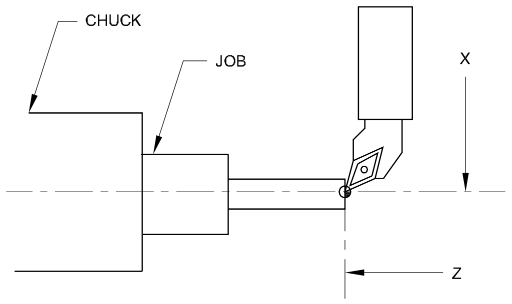
Tool offset in X axis is saved

Z axis offset method

Spindle ON rotate the job.

Select jog mode or MPG mode to move axis. Manually turning job facing position no disturbance Z axis. (Fig 2).

Fig 2



Z AXIS OFFSET METHOD

Select offset button press in geomentry mode.

Use cursor select tool no 2 and z axis.

Enter Z0.

Enter Z0 press Measure in soft key

Now z axis tool offset OK.

Tool offset is Z axis is saved

Second tool offset

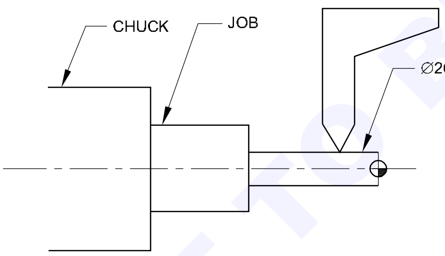
Select MDI mode Press MDI Prog Screen.

Enter tool no (Threading tool) T0300 Press Reset button Press Cycle start.

Select jog mode or MPG mode then move axis.

Same procedure MPG mode incremental touch job in x axis with piece of paper Do not distrub X axis (Fig 3).

Fig 3



Z AXIS OFFSET METHOD

Select offset button press in geomentry mode.

Use cursor to select tool no 3 and x axis.

Enter constant same dia.

Ex: x28.62 Press Measure button in soft key

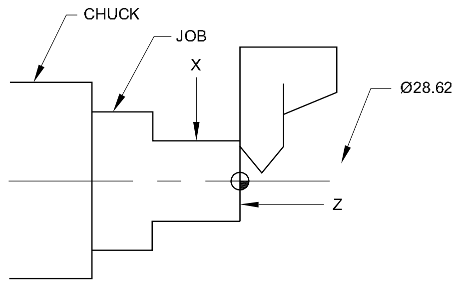
Threading tool offset measurement

Z axis offset

Select MPG mode in incremental variation. To move axis z position.

To check by inserting a piece of paper between tool and the job ensuring that there is no disturbance in Z axis (Fig 4).

Fig 4



Z AXIS OFFSET METHOD

Select offset button in geomentry mode.

To use cursor select tool no 3 and z axis.

Enter Z0.

Ex : Z0 Press Measure in soft key

Now second tool z axis offset saved.

Internal drilling tool x axis offset method

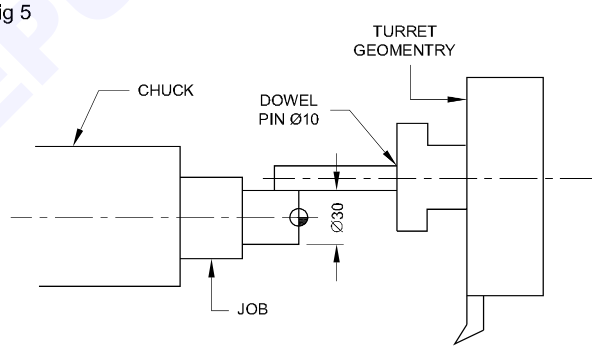
Fix the turned job in chuck

Select MDI mode press enter tool no T0400 and cycle start.

Select jog or MPG mode to move axis on the job of top side.

Check with piece of paper whether contact of dowel pin with the job is proper. (Fig 5)

Fig 5



DIAMETER OF JOB + DIAMETER OF DOWEL PIN = TOTAL DIA
INTERNAL DRILLING TOOL X AXIS OFFSET METHOD

Select offset screen in geomentry mode.

Use cursor to select x mode and tool number.

Enter dia.

Ex : Job dia + Dowel pin dia = Total dia

30 + 10 = 40

Enter dia x40 Press Measure button in soft key

Drill cutting point in job centre point.

Then fix the drill.

Touch the job face a piece of paper whether contact of drill with the job in proper.

Enter Z0 Press Measure in soft key

Drilling tool off set is saved

Tools nose radius shall get automatically added in the tool offset.But in programming, TNC is considered through G codes.

Entering the tool offset, tool nose radius and orientation for TNRC in offset page on CNC simulator

Objectives : At the end of this exercise you shall be able to
- **identify the tool offset page**
- **enter tool data in offset page.**

Job Sequence

- Press function key OFFSET SETTING.
- Press chapter selection soft key [OFFSET] or press OFFSET SETTING several times until the tool compensation screen is displayed.
- Pressing soft key [GEOM] displays tool geometry compensation values as in Fig 1.

Fig 1

OFFSET/GEOMETRY 00001 N00000

NO.	X	Z	R	T
G 001	0.000	1.000	0.000	0
G 002	1.486	-49.561	0.000	0
G 003	1.486	-49.561	0.000	0
G 004	1.486	0.000	0.000	0
G 005	1.486	-49.561	0.000	0
G 006	1.486	-49.561	0.000	0
G 007	1.486	-49.561	0.000	0
G 008	1.486	-49.561	0.000	0

ACTUAL POSITION (RELATIVE)

U 101.000 W 202.094

>

MDI **** *** *** 16:05:59

[WEAR] [GEOM] [WORK] [ ] [QPRT]

WITH TOOL GEOMETRY OFFSET

MA20N23152H1

- Pressing soft key [WEAR] displays tool wear compensation values. (Fig 2)
- Move the cursor to the compensation value to be set or changed using page keys and cursor keys, or enter the compensation number for the compensation value to be set or changed and press soft key [NO.SRH].
- To set a compensation value, enter a value and press soft key [INPUT]. To change the compensation value, enter a value to add to the current value (a negative value to reduce the current value) and press soft key [+INPUT]. Or, enter a new value and press soft key [INPUT].
- TIP is the number of the virtual tool tip (see Programming).

Fig 2

OFFSET/GEOMETRY 00001 N00000

NO.	X	Z	R	T
W 001	0.000	1.000	0.000	0
W 002	1.486	-49.561	0.000	0
W 003	1.486	-49.561	0.000	0
W 004	1.486	0.000	0.000	0
W 005	1.486	-49.561	0.000	0
W 006	1.486	-49.561	0.000	0
W 007	1.486	-49.561	0.000	0
W 008	1.486	-49.561	0.000	0

ACTUAL POSITION (RELATIVE)

U 101.000 W 202.094

>

MDI **** *** *** 16:05:59

[WEAR] [GEOM] [WORK] [ ] [QPRT]

WITH TOOL WEAR OFFSET

MA20N23152H2

- TIP may be specified on the geometry compensation screen or on the wear compensation screen.

Note:A decimal point can be used when entering a compensation value.

Procedure for entering tool offset data in CNC simulator and on machine will be same.

- Dimensions and annotations (2 methods): Select the import (Show model annotations) dimensions used to create the model
- Create (New reference) dimentsions (Note: reference dimensions cannot be changed) (Fig 3)
- When importing dimensions try using the feature/view option versus inserting all the dimesnions for the mode as it will cluster all them togerther. Feature helps reduce the cluster and yet the dimensions are editable, providing the benfit to edit the actual parts and assemblies in a bi-directional fashion from the drawing.
- Editing the sheet: Use the "Note" tool to enter your name and part number. (Fig 4)

Fig 3

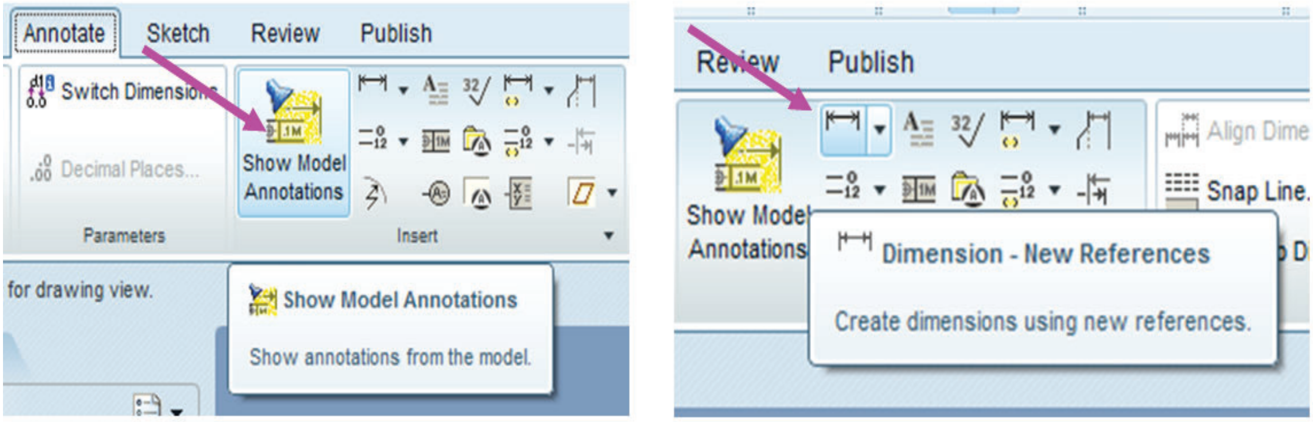
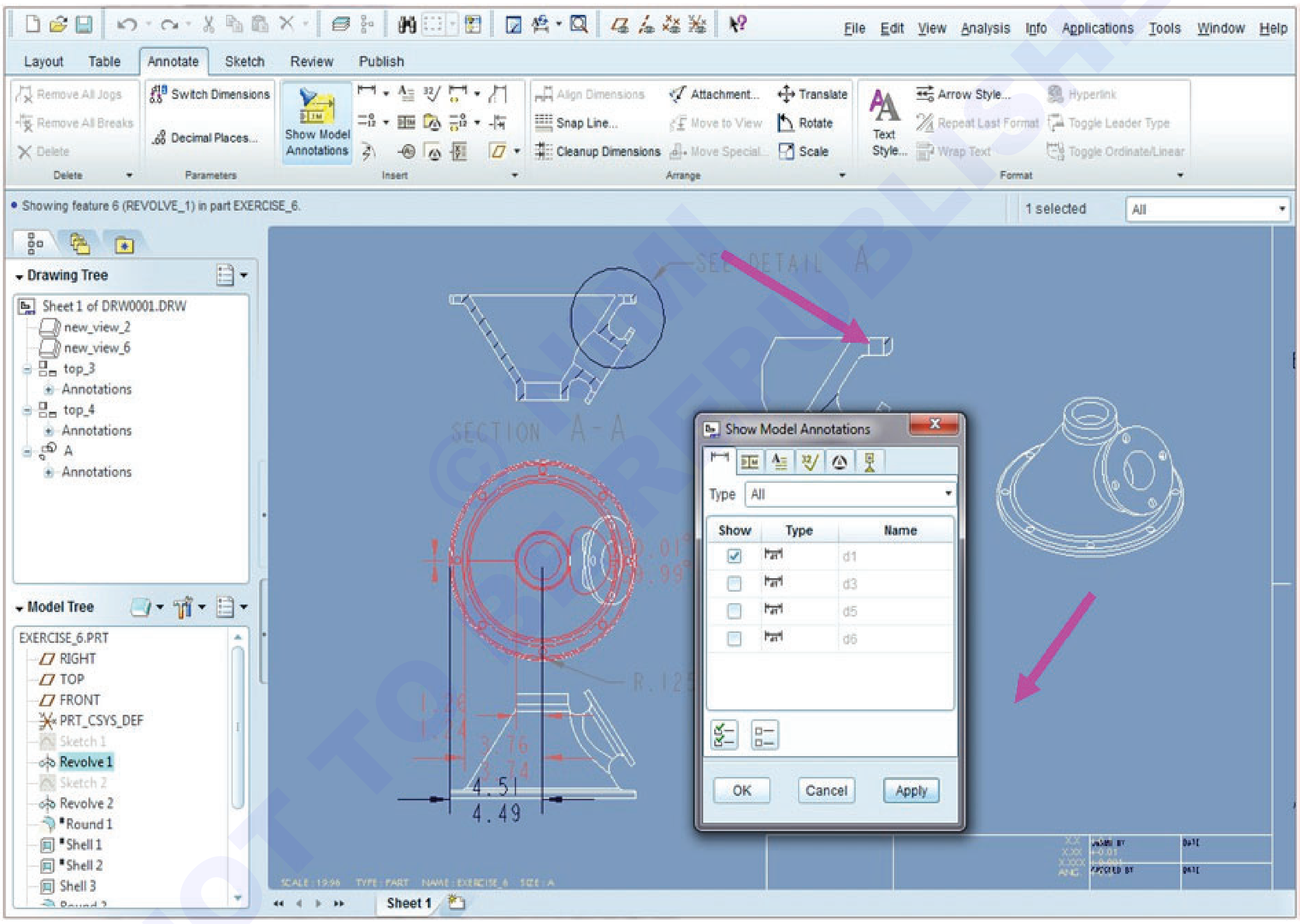


Fig 4



Skill Sequence

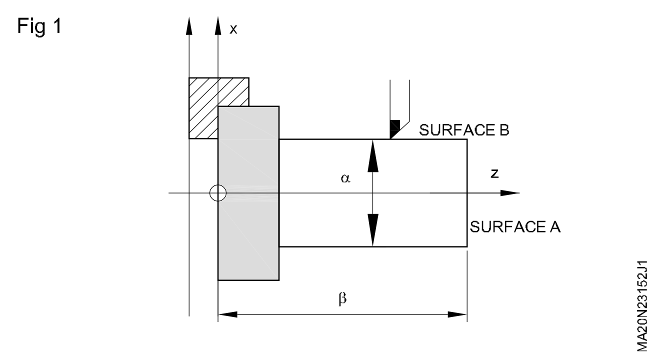
Direct input of tool offset value

Objective: This shall help you to
- **measure and direct input tool datas.**

Setting of Z axis offset value

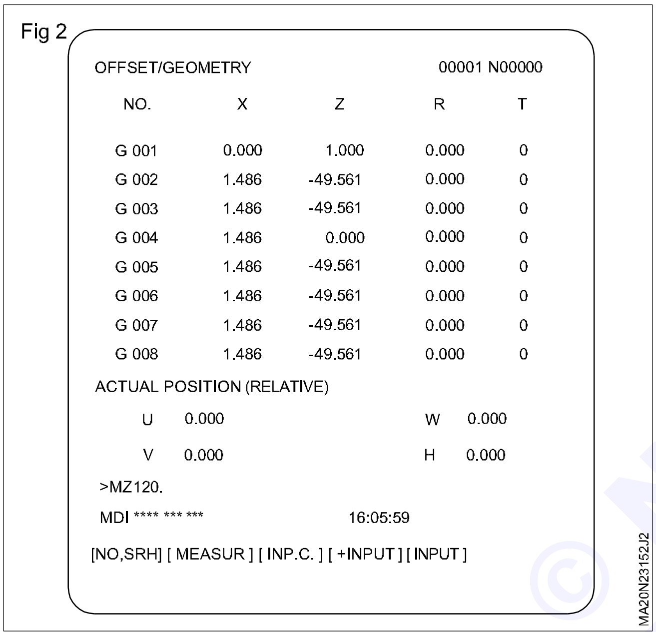
- Cut surface A in manual mode with an actual tool. Suppose that a workpiece coordinate system has been set. (Fig 1)
- Release the tool in X axis direction only, without moving Z axis and stop the spindle.
- Measure distance B from the zero point in the workpiece coordinate system to surface A.

Fig 1



- Set this value as the measured value along the Z–axis for the desired offset number, using the following procedure. (Fig 2)

Fig 2



- Press the function key OFFSET SETTING or the soft key [OFFSET] to display the tool compensation screen. If geometry compensation values and wear compensation values are separately specified, display the screen for either of them.
- Move the cursor to the set offset number using cursor keys.
- Press the address key Z to be set.
- Key in the measured value (β).
- Press the soft key [MESURE].
- The difference between measured value â and the coordinate is set as the offset value.

Setting of X axis offset value.

- Cut surface B in manual mode.
- Release the tool in the Z–axis direction without moving the X–axis and stop the spindle.
- Measure the diameter á of surface B. Set this value as the measured value along the X–axis for the desired offset number in the same way as when setting the value along the Z–axis.
- Repeat above procedure for other necessary tools. The offset value is automatically calculated and set.
- For example, in case α=69.0 when the coordinate value of surface B in the diagram above is 70.0, set 69.0 [MEASURE] at offset No.2.
- In this case, 1.0 is set as the X–axis offset value to offset No.2.

Compensation values for a program created in diameter programming

Enter diameter values for the compensation values for axes for which diameter programming is used.

Tool geometry offset value and tool wear offset value.

If measured values are set on the tool geometry compensation screen, all compensation values become geometry compensation values and all wear compensation values are set to 0. If measured values are set on the tool wear compensation screen, the differences between the measured compensation values and the current wear compensation values become the new compensation values.

Program checking in dry run single block mode

Objectives : At the end of this exercise you shall be able to
- **load the program to run in auto mode operation**
- **check the program in dry run using single block mode.**

Job sequence

- Load the program to run in auto mode operation
- Keep the feed rate and rapid knobs to zero position.
- Press cycle start
- Press dry run and single block mode
- Open the rapid switch to 30%
- Press cycle start button, the execution of the program is stopped after the current block is executed.
- Press cycle start button to executeive next block.
- Similarly continued until the end of program that is M30;

Skill Sequence

Running program in auto mode/memory operation

Objectives: This shall help you to
- **load the program to run in auto mode**
- **execute the program in auto mode.**

Memory operation

Programs are registered in memory in advance. When one of these programs is selected and the cycle start switch on the machine operator's panel is pressed, automatic operation starts, and the cycle start LED goes on.

Steps in memory operation

- Press the MEMORY mode selection switch.
- Select a program from the registered programs. To do this, follow the steps below.
- Press [PROG] to display the program screen.
- Press address [O]
- Enter a program number using the numeric keys.
- Press the [O SRH] soft key.
- Press the cycle start switch on the machine operator's panel. Automatic operation starts, and the cycle start LED goes on. When automatic operation terminates, the cycle start LED goes off.

To stop or cancel memory operation midway through, follow the steps below.

Stopping memory operation

- Press the feed hold switch on the machine operator's panel. The feed hold LED goes on and the cycle start LED goes off. The machine responds as follows.
- When the machine was moving, feed operation decelerates and stops.
- When dwell was being performed, dwell is stopped.
- When M, S, or T was being executed, the operation is stopped after M, S or T is finished.
- When the cycle start switch on the machine operator's panel is pressed while the feed hold LED is on, machine operation restarts.

Terminating memory operation

- Press the [RESET] key on the MDI panel.

Automatic operation is terminated and the reset state is entered.

- When a reset is applied during movement, movement decelerated the stops.

Dry run and single block mode

Objective: This shall help you to

- **run the program in dry run mode and single block mode.**

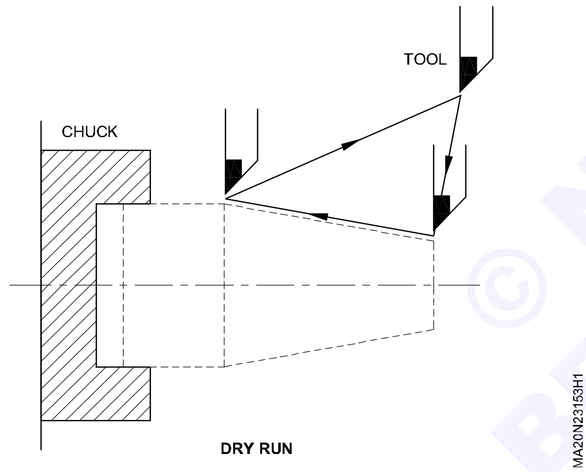
Dry run

The tools is moved at the federate specified by a parameter regardless of the federate specified in the program. This function is used for checking the movement of the tool under the state that the workpiece is removed from the table.

Steps for dry run operation (Fig 1)

- Load the program
- Select auto mode operation
- Press the dry run switch on the machine operator's panel during automatic/memory operation.
- Press cycle start. The tool moves at the feed rate specified in a parameter.
- Rapid traverse switch can also be used for changing the feed rate.

Fig 1



Single block operation

Pressing the single block switch, starts the single block mode. When the cycle start button is pressed in the single block mode, the tool stops after a single block in the program is executed. Check the program in the single block mode by executing the program block by block.

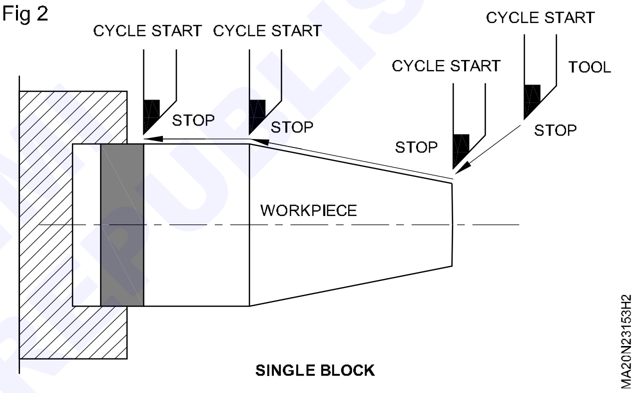
Steps for single block (Fig 2)

Press the single block switch on the machine operator's panel. The execution of the program is stopped after the current block is executed.

Press the cycle start button to execute the next block. The tool stops after the block is executed.

Refer to the appropriate manual provided by the machine tool builder for single block execution.

Fig 2



Absolute & incremental programming assignments by simulation

Objective: At the end of this exercise you shall be able to
- **program with absolute and incremental system.**

TASK 1 : **Method of Programming 1 (Fig 1)**

There are two methods of dimensioning .

1 Absolute system of programming (or) fixed

2 Incremental system of Programming (or) floating zero system of dimensioning. (or) previous point zero system of dimensioning.

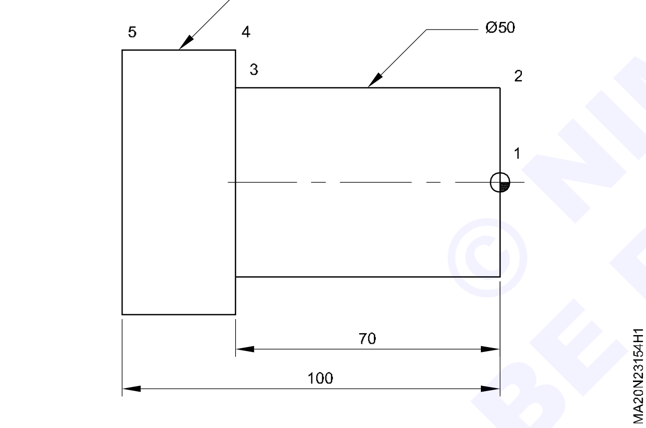
Absolute Programming

In absolute dimensions programming all the point of the tools is coming from the datum point (or) zero point.

Incremental Programming

In this system, tool move form the previous point.

Fig 1



Example 1 : The points 1 to 5 in the drawing indicates the absolute in Table 1 and Incremental in Table 2

Absolute program in table 1.

TABLE 1

Absolute		
Position	**X**	**Z**
1	0. 0	0. 0
2	50.0	0. 0
3	50.0	-70.0
4	70.0	-70.0
5	70.0	-100.0

Incremental program in table 2.

TABLE 2

Incremental		
Position	**U**	**W**
1	0.0	0.0
2	50.0	0.0
3	0.00	- 70.0
4	20.0	0.0
5	0.0	- 30.0

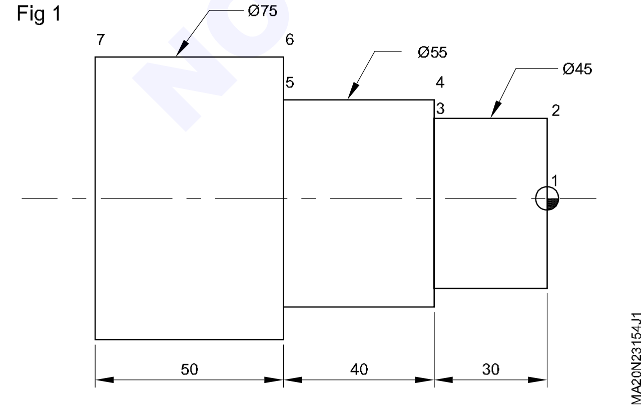
Exercise for Absolute & Incremental Methods

Write the points for the following figures in absolute & incremental programming.

— — — — — — — —

TASK 2 : **Method of Programming 2 (Fig 1)**

Fig 1



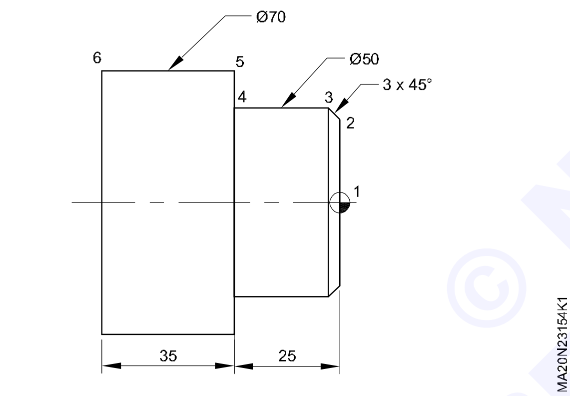
Absolute		
Position	**X**	**Z**
1		
2		
3		
4		
5		
6		
7		

Incremental		
Position	U	W
1		
2		
3		
4		
5		
6		
7		

TASK 3 : **Method of Programming 3 (Fig 1)**

Trainees to indicate the co-ordinate values in the given tables.

Fig 1

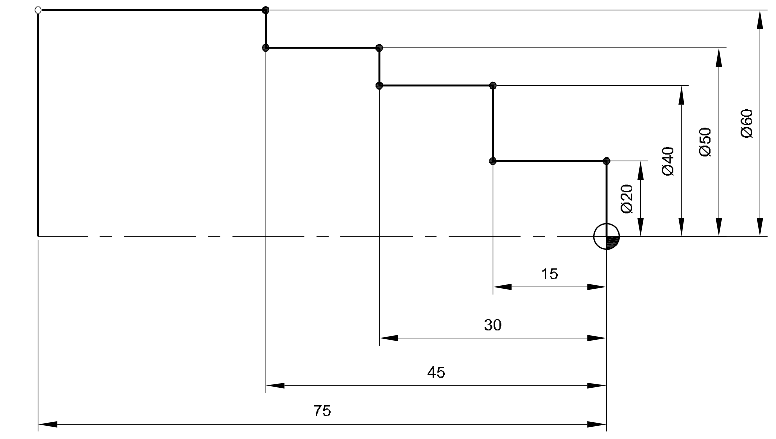


Absolute		
Position	X	Z
1		
2		
3		
4		
5		
6		

Incremental		
Position	U	W
1		
2		
3		
4		
5		
6		

TASK 4 : **Plot the points in absolute and incremental modes for Figs 1, 2 & 3.**

Fig 1



(G90)

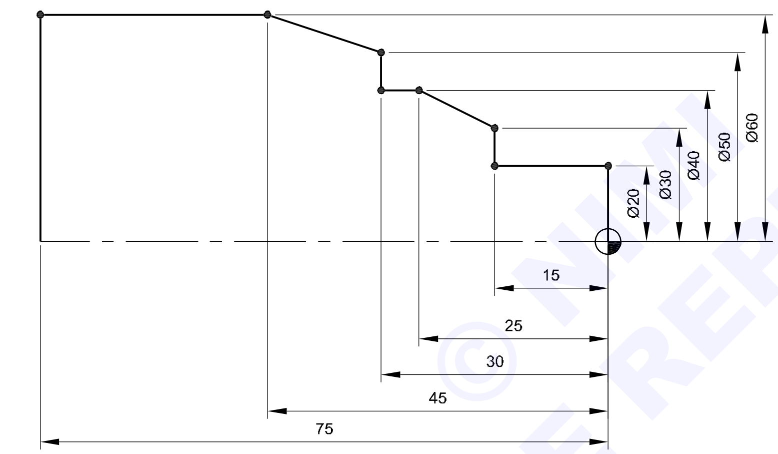
SL NO	X axis	Z axis
1		
2		
3		
4		
5		
6		
7		
8		
9		

(G91)

SL NO	X axis	Z axis
1		
2		
3		
4		
5		
6		
7		
8		
9		

MA20N23154X1

Fig 2



(G90)

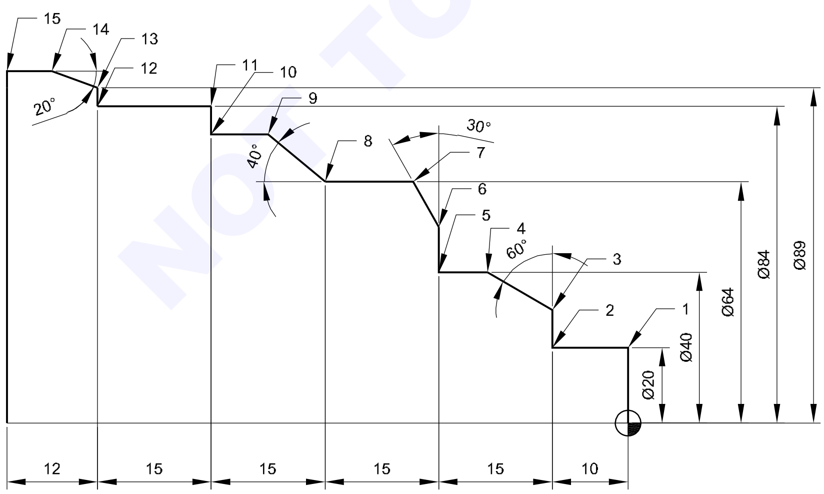
SL NO	X axis	Z axis
1		
2		
3		
4		
5		
6		
7		
8		
9		

(G91)

SL NO	X axis	Z axis
1		
2		
3		
4		
5		
6		
7		
8		
9		

MA20N23156H2

Fig 3



(G90)

SL NO	X axis	Z axis
1		
2		
3		
4		
5		
6		
7		
8		
9		
10		
11		
12		
13		
14		
15		

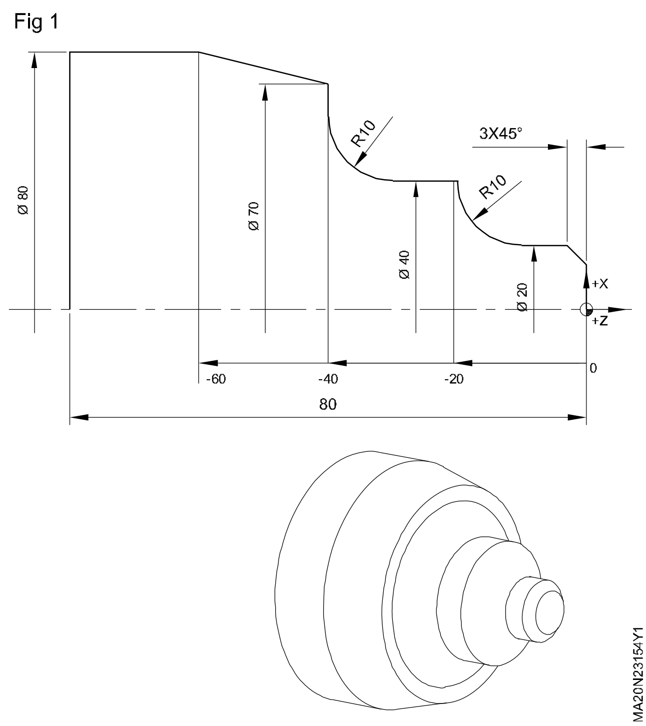
(G91)

SL NO	X axis	Z axis
1		
2		
3		
4		
5		
6		
7		
8		
9		
10		
11		
12		
13		
14		
15		

MA20N23156H3

TASK 5 **: (Fig 1)**

Write the tool path using G01, G02 & G03 with G90 +G91

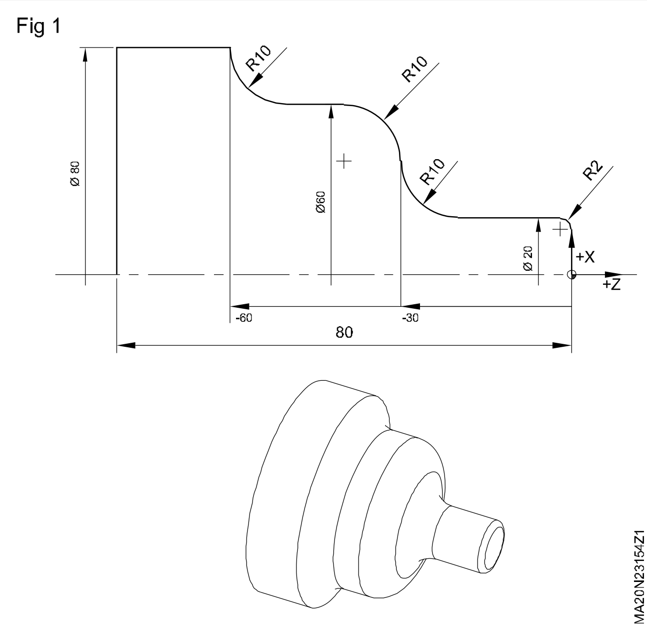


G91					
N	**G**	**X**	**Z**	**I**	**K**
N1					
N2					
N3					
N4					
N5					
N6					
N7					
N8					
N9					
N10					
N11					
N12					
N13					

G90					
N	**G**	**X**	**Z**	**I**	**K**
N1					
N2					
N3					
N4					
N5					
N6					
N7					
N8					
N9					
N10					
N11					
N12					
N13					

TASK 6 **: (Fig 1)**

Write the tool path using G01, G02, G03 with G90/G91.



G91					
N	**G**	**X**	**Z**	**I**	**K**
N1					
N2					
N3					
N4					
N5					
N6					
N7					
N8					
N9					
N10					
N11					
N12					

G90					
N	**G**	**X**	**Z**	**I**	**K**
N1					
N2					
N3					
N4					
N5					
N6					
N7					
N8					
N9					
N10					
N11					
N12					

Checking finish size by over sizing through tool offset on CNC simulator

Objective: At the end of this exercise you shall be able to
- **do the correction of size in tool offset.**

Job sequence

- Switch ON machine as per procedure
- Enter the program as per drawing
- Run the simulation on dry run with machine
- If there is no error run the machine in SBL or Auto mode
- Check the finished dimensions
- If there is any error compared to the required dimension, calculate the difference
- Add the difference in value to the respective axis in the tool offset
- Run and produce a sample to the correct measurement.

After completing the operation enter actual value of job in Table 1

Table 1

Tool Numbers	X axis Value	Z axis Value
1	20.02	45.05
2		
3		
4		

Note

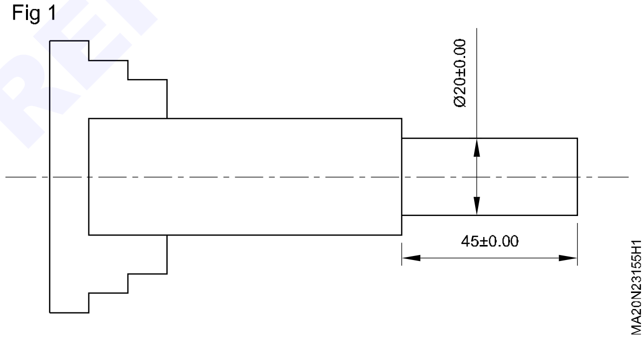
- Required size of the is job X 20 ± 0.00
  Z 45 ± 0.00
- Go to wear offset page and input the X,Z difference value
- Add the Tool wear offset difference value on the table column in Table 2.

Table 2

Tool Number	X axis Value	Z axis Value
1	- 0 .02	- 0.05
2		
3		
4		

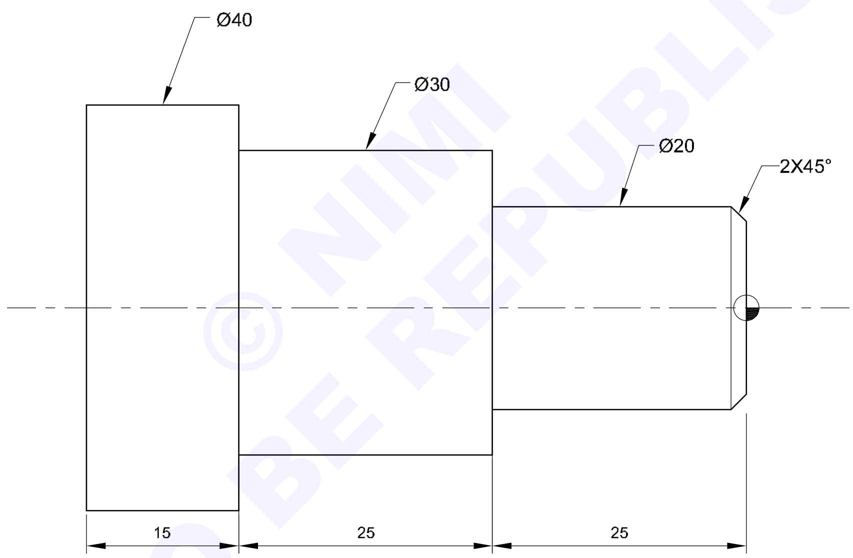
- To avoid the rejection of job the first piece is made, slightly increased in size in offset.
- After completion of the first piece,check all the dimension without removing the chuck.
- Observe the difference value of drawing size and Acutal size, and this difference, if any, should be 'input' in the wear off set.

Fig 1



Facing and step turning in auto mode

Objectives : At the end of this exercise you shall be able to
- **prepare CNC program for the given job drawing**
- **enter the program in CNC control**
- **verity the program by simulation**
- **run the program in auto mode.**



1	Ø 50 x 80	-	ALUMINIUM	-	1	2.3.156
NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	Ex. NO.

SCALE 1:1	FACING AND STEP TURNING IN AUTO MODE	TOLERANCE : ± 0.02	TIME :
		CODE NO : MA20N23156E1	

Job sequence

- Write the CNC program in sinumeric control
- Mount the work on 3 jaw chuck
- Mount the turning tool in tool turret
- Measure the work and tool offset
- Enter the work and tool offset in relevant page
- Verify the program by simulation.
- Check the program by dry run method .

Work piece should by removed or shift the work offset in z axis only during dry un method.

- Run the program in auto mode
- Check the a part dimensions and compare with drawing dimension
- If there is any error in dimension do the wear correction.

PRAGRAM FOR FACING AND STEP TURNING

N1 G90 G55 G94 G71

N2 WORKPIECE (,,, "CYLINDER" , 0,0,80,68,50)

N3 T= "ROUGHING_T80 A"

N4 S500 M03

N5 G00 G42 X55 Z0

N6 G01 X-0.1 Z0 F0.1

N7 G00 X-0.1 Z5

N8 G00 X42 Z5

N9 G01 X42 Z-64

N10 G01 X55

N11 G00 X55 Z5

N12 G00 X31 Z5

N13 G01 X31 Z-49

N14 G01 X42 Z-49

N15 G00 X42 Z5

N16 G00 X21 Z5

N17 G01 X21 Z-24

N18 G01 X32 Z-24

N19 G00 G40 X100 Z100

N20 T= "FINISHING_ T35 A"

N21 S700 M03

N22 G00 G42 X16 Z5

N23 G01 X16 Z0 F0.1

N24 G01 X20 Z-2

N25 G01 X20 Z-25

N26 G01 X30 Z-25

N27 G01 X30 Z-50

N28 G01 X40 Z-50

N29 G01 X40 Z-65

N30 G01 X42 Z-65

N31 G00 G40 X100 Z100

N32 M05

N33 M30

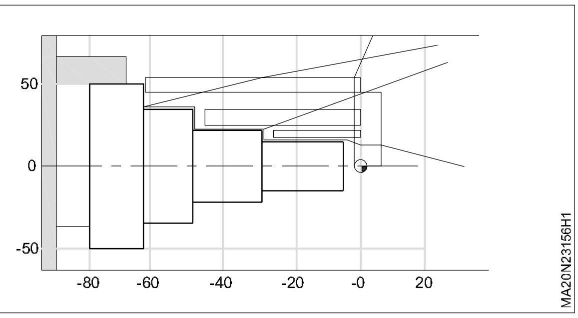
NOTE N1 TO N7 FACING

N8 TO N19 STEP TURNING ROUGHING

N20 TO N33 FINISH TURNING

Fig 1 shows the detailed tool path for roughing and finishing operation.

Fig 1



Recovering from axes over travel, on CNC simulator

Objective: At the end of this exercise you shall be able to
- **release the axes over travels.**

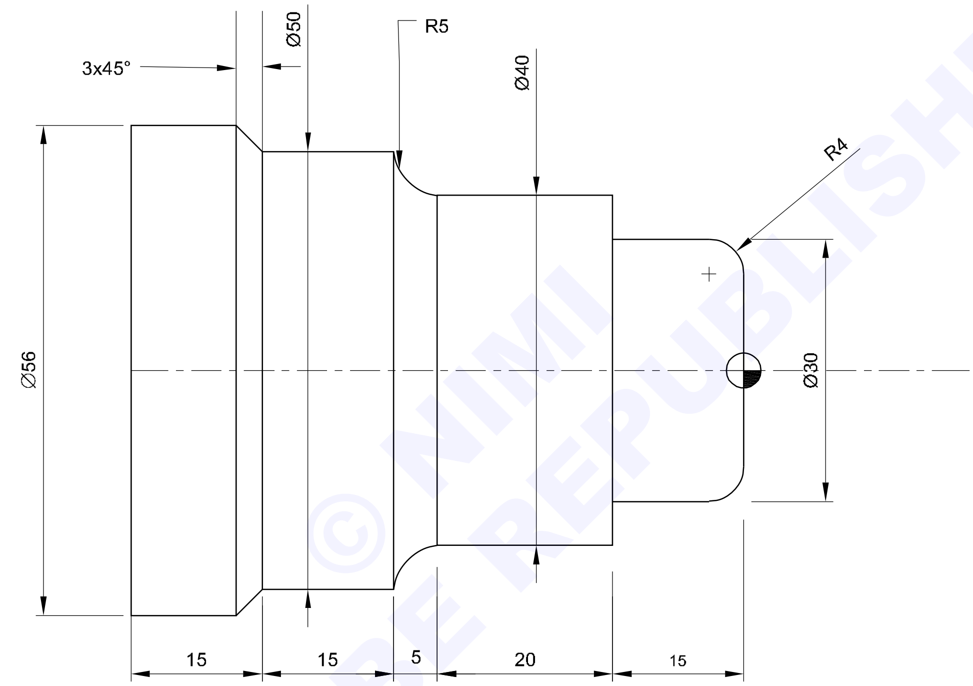
Job Sequence

- Observe on CNC simulator monitor screen the axes over travel in 'x' or 'z' axes
- Press reset button to clear over travel alarm
- Select MPG mode.
- Accordingly select the axis
- Move the axis in opposite direction to recover the over travel limit
- Continue the further operation (AUTO, MDI)

Note: The trainer shall demonstrate on how to recover the axes over travel in X and Z axis using CNC simulator and ask the trainees to practice it.

Part program preparation, for turning with chamfering & radius turning with TNRC

Objectives: At the end of this exercise you shall be able to
- **write the part program for given job in SINUMERIC / FANUC**
- **input the program into the computer / CNC machine**
- **operate the simultor**
- **transfer to machine and execute simulated programe.**



Job Sequence

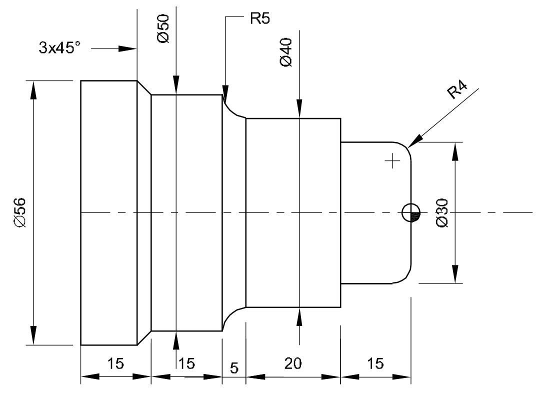
- Switch ON the CPU and Monitor.
- Select the software.
- Enter the program in edit mode.
- Select required tool and clamping device.
- Select the path.
- Run the Simulation in AUTO mode .
- Transfer the programe to the machine.
- Take the offset.
- Execute the program.
- Check the dimension and remove the job.
- Switch off the machine.

1	Ø 60 x 100	-	Fe310	-	-	2.3.158
NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.

SCALE 1:1	**TURNING OPERATION**	DEVIATIONS ± 0.1	TIME :
		CODE NO. MA20N23158E1	

PART PROGRAM (IN SINUMERIC)

Fig 1




```
% 225
N5   G71  G90  G95;
N10  G00  G53  X180  Z0;
N15  D5   T5
N20  M03  S900  F0.1;
N25  G00  X65   Z2;
N30  R20 =185 R21=65  R22=0  R23=0.3 R24=0.3
     R26 =1   R27=42  R29=41 L95 P1; (stock
     Removal cycle)
N35  D0   M05;
N40  G00  G90  G53  X180  Z0;
N45  M30;
```

L185 (SUB PROGRAM)

```
N5   G01  X0   F0.1;
N10  G01  Z0 F0.1;
N15  G01  G42  X22  F0.1;
N20  G90  G03  X30  Z-4  B4  F0.1;
N25  G01  Z-15  F01.;
N30  X40;
N35  Z-35;
N40  G90  G02  X50  Z-40  B5;
N45  G01  Z-55;
N50  X56  Z-58;
N55  G01  G40  Z-70;
N60  X65;
N65  M17;
```

R - PARAMETER FOR STOCK REMOVAL CYCLE(L95)

R 20 = Sub Program Number or Sub routine No.

R 21 = Starting position of X axis (Absolute)

R22 = Starting position of in Z axis (Absolute)

R23 = Finishing allowance of X (Incremental)

R24 = Finishing allowance of Z (Incremental)

R26 = Roughing cuts

R27 = Tool Nose Radius compensation

R29 = Type of turning (Roughing , Finishing)

L95 = Stock removal cycle

P1 = No.of Tool passes

PART PROGRAM [FANUC]

```
O0002
G21 G90 G95 G55
G28  U0  W0
T0101;
G97  S1500  M03;
G00  X65.0 Z5.0  M08;
G01  Z2.0  F.01;
G71  U1.0  R1.0;
G71  P10  Q20  U0.0  W0.0  F0.1;
N10  G01  X0.0  F0.1;
G01  Z0.0  F0.1;
G01  G42  x 22.0  F0.1;
     G03  X30.0  Z-4.0  R4  F0.1;
     G01  Z-15.0 F0.1;
     X40.0;
     Z-35.0;
     G02  X50.0  Z - 40  R5  F0.1;
     G01  Z-55.0 F0.1;
     X56.0  Z-58.0 ;
N20  G01  G40  Z-70.0 F0.1;
     G00  X65.0  Z5.0  M09;
     G28  U0  W0;
     M05;
     M30;
```

G71 - Turning Cycle

G71 U --- R --- ;

G71 P --- Q --- U --- W --- F --- ;

U : Depth cut per pas in ‘X’ axis (Radial Value)

R : Relief Amount

P : Starting block number

Q : Ending block number

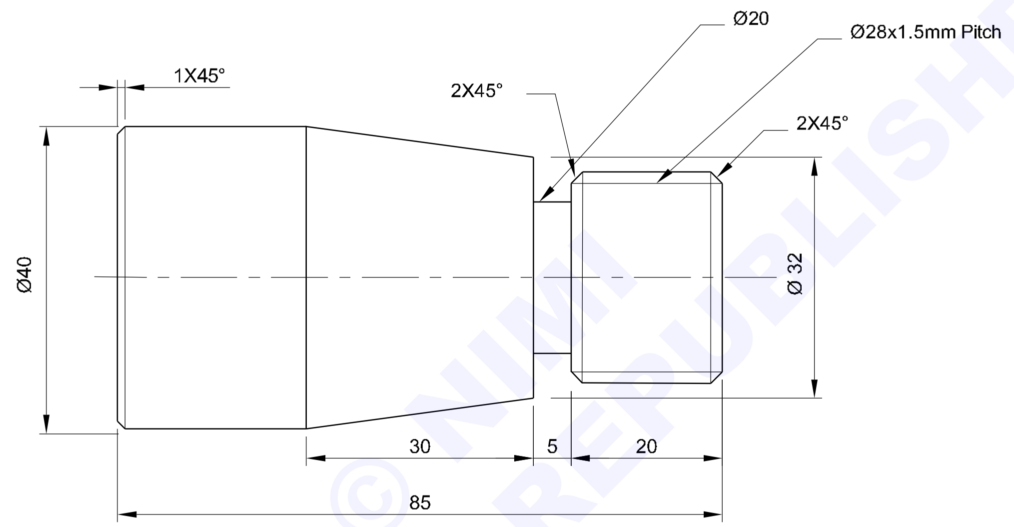
U : Finishing Allowance in ‘X’ axis

W : Finishing Allownace in ‘Z’ axis

F : Feed

Part program writing and execute

Objectives: At the end of this exercise you shall be able to
- **write the part program**
- **check the setup**
- **execution of turning operation in automatic mode with TNRC**
- **grooving and threading.**



Job Sequence

- Write the job sequence.
- Prepare the CNC program using canned cycles in both control (simens an fanuc).
- Verify the program using CNC simulator.
- Load the program in CNC machine.
- Set job and work offset and tool offset.
- Verify the program by simulation and dry run method.
- Execute the program in auto mode.

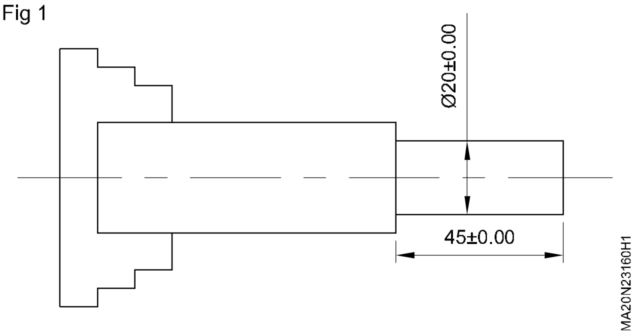
1	Ø 45 x 105	-	Fe310	-	-	2.3.159
NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.

SCALE 1:1	PART PROGRAM FOR CNC SIMULATOR AND CNC MACHINE FOR TURNING, GROOVING AND THREADING	DEVIATIONS ± 0.1	TIME :
		CODE NO. MA20N23159E1	

Checking finish size by over sizing through tool offset on the machine

Objective: At the end of this exercise you shall be able to
- **do the correction of size in tool offset.**

Fig 1



Job Sequence

- Switch ON machine as per procedure
- Enter the program as per drawing
- Run the simulation on dry run with machine
- If there is no error run the machine in SBL or Auto mode
- Check the finished dimensions
- If there is any error compared to the required dimension calculate the difference
- Add the difference in value to the respective axis in the tool offset
- Run and produce a sample to the correct measurement.

After complete the operation enter actual value of job

Tool Numbers	X axis Value	Z axis Value
1	20.02	45.05
2		
3		
4		

Note

- Required size of the is job X 20 ± 0.00
  Z 45 ± 0.00
- Go to wear offset page and input the X,Z difference value

Tool Number	X axis Value	Z axis Value
1	- 0 .02	0.05
2		
3		
4		

- Add the Tool wear offset difference value on the table column
- To avoid the rejection of job the first piece is made, slightly increased in size in offset.
- After completion of the first piece,check all the dimension without removing the chuck.
- Observe the difference value of drawing size and Acutal size, and this difference, if any, should be 'input' in the wear off set.

Skill Sequence

Correction of work piece diameter in CNC lathe

Objective: This shall help you to
- **turn the work piece to correct dimensions in diameter and its length (X and Y).**

1 if the work diameter measures more than the required value

Reduce the tool's x offset the difference between the measured value and the required value

Or

Enter the difference between the measured value and the required value against x wear with a minus sign

Example:	Required value measured value X tool offset	35.20mm 35.37mm 47.32mm
Solution (a)		
	Tool offset correction	= 35.37 - 35.20
		= 0.17
	New tool offset	= 47.32 - 0.17 =47.49
Solution (b)		
	Keep the tool offset unchanged (47.32) and enter -0.17 against X wear.	

2 If the work diameter measures less than the required value

Increase the tool's X offset the difference between the measured value and the required value.

Or

Enter the difference between the measured value and the requied value against X wear with a plus sign

Example:	Required value measured value X tool offset	40.37mm 40.15mm 57.732mm
Solution(a)		
	Tool offset correction	=40.37 -40.15=0.22
		=0.22
	New tool offset	= 57.732 +0.22 = 57.952
Solution (b)		
	Keep the tool offset unchanged 57.732 and enter +0.22 against X wear.	

3 If the work piece length is measured less than the required value

Reduce the tool's Z offset by the difference between the measured value and the programmed value

Example:

Solution (a)	Tool offset in Z	= 40.00mm
	Programmed length	= 50.00mm
	Measured length	= 49.50mm
	Difference	= 0.50 mm
	New tool offset in Z 40.00-0.50 =39.50	
Solution (b)	Keep the tool offset in Z unchanged (40.00) and enter -0.50 against Z wear.	

4 If the work piece length is measured more than the required value

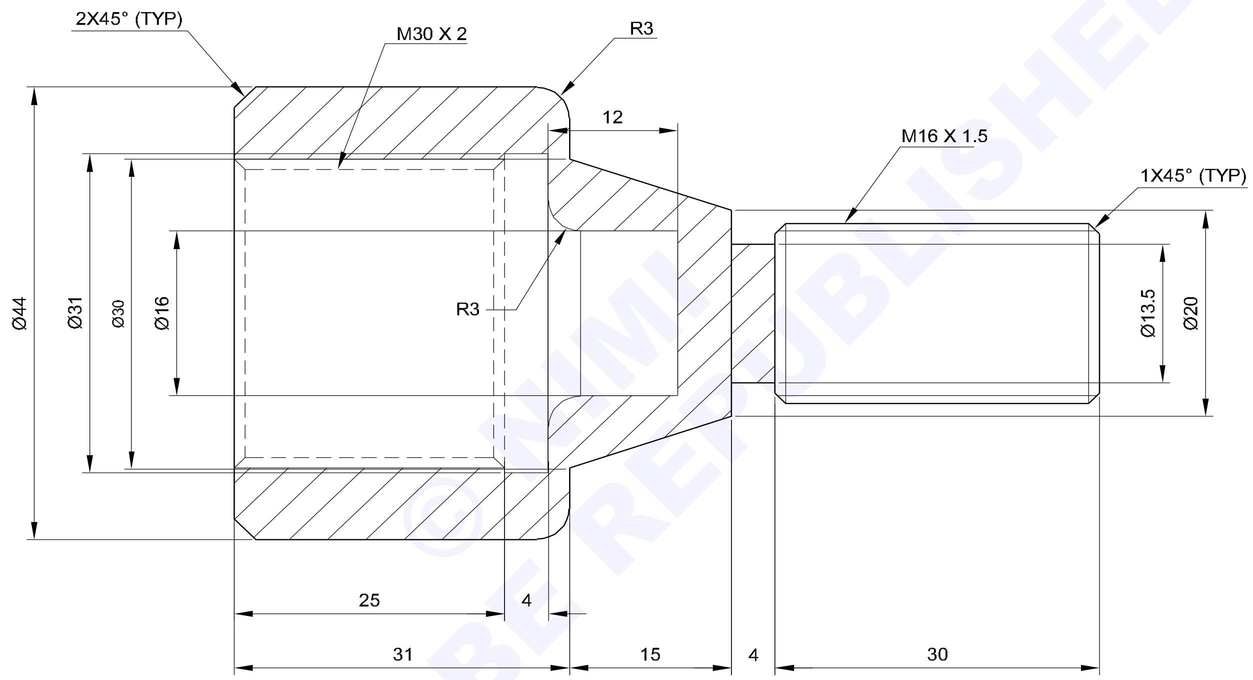
Add the tool Z offset by the difference between the measured value and programmed value

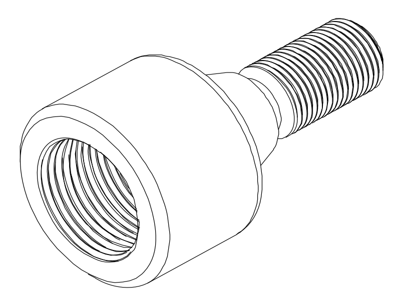
Example :

Solution (a)	Tool offset in Z	= 40.00mm
	Programmed length	= 50.00mm
	Measured length	= 50.50mm
	Difference	= 0.50mm
	New tool offset in Z = 40.00 +0.50 =40.50	
Solution b	Keep the tool offset in Z unchanged 40.00 and enter +0.50 against Z wear.	

Machine parts with external and internal-features

Objectives : At the end of this exercise you shall be able to
- **write the program for external features**
- **write the program for internal features**
- **verify the program**
- **execute the program in CNC turning centre**
- **maintain the dimensions as per drawing.**





1	Ø50 X 110	-	ALUMINUM	-	1	2.3.161
NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.

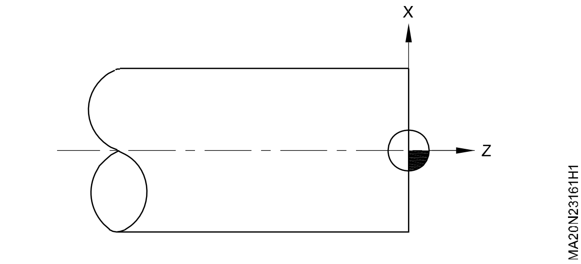
SCALE 1:1	**EXTERNAL AND INTERNAL TURNING OPERATION**	DEVIATIONS ± 0.02	TIME :
		CODE NO. MA20N23161E1	

Job Sequence

1st Operation (External)

- Write the CNC program for external operations in fanuc and siemens control using stock removal grooving and threading cyles
- Part of using parting cycle
- Enter the program in CNC machine and verify the program by simulation and dry run method.
- Set the workpiece on self centring chuck
- Set the tools in turret in accordance with program
- Fix the work zero at the front end face of work piece (Fig 1)

Fig 1

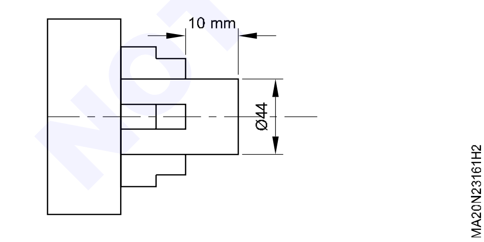


- Set the tool offset for rough turning finish turning grooving and threading tool.
- Check the tool offset
- Run the program in auto mode
- Check the dimensions of the work piece. It any error correct it by wear offset. And once again run the program and check the dimension
- Load the parting program and run the program and part off the work piece keeping 1mm on face is machine it

2nd Operation (Internal)

- Determine the facing allowance
- Hold the work piece on 44mm and projecting 10mm away from the jaw of self centering chuck. (Fig 2)

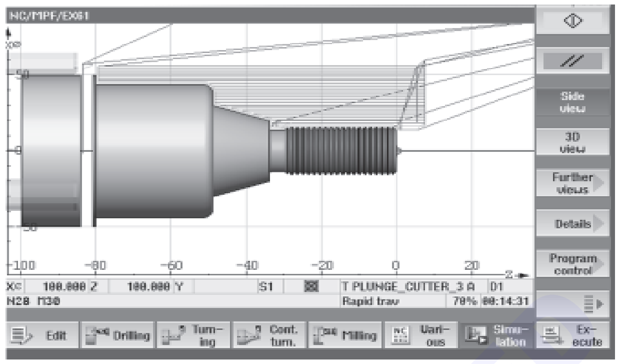
Fig 2



- Set the 'U' drill 16mm boring tool, grooving tool and threading tool in accordance with program
- Set tool offset and work offset
- Shift work zero in to the work piece as per the facing allowance determined and set new work zero
- Run the program, and check the dimension
- Go get it verified by your trainer

CNC external operation program in simens control (Fig 3)

Fig 3



N1 WORKPIECE(,,, "CYLINDER" ,0,0,100,85,50)

N2 T= "FINISHING _T35 A"

N3 M03 S1000

N4 G95 F100

N5 G00 X100 Z100

N6 G01 X55 Z5

N7 CYCLE62("111" , 0,,)

N8CYCLE952 ("111",,"",2101311,0.10,0,1,0.1,0.1, 0.5,0.1,0.1,0,1,0.1,0,,,,,2,2,,,0,2,,0,12 1100010,1,0)

N9 G00 X100 Z100

N10 G00 X13 Z5F100

N11G01 X13 Z0 F100

N12 G01 X16 Z-1.5

N13 G0.1 X16 Z-34

N14 G01 X20 Z-34

N15 G01 X30 Z-49

N16 G01 X38 Z-49

N17 G03 X44 Z-52 I0K-3

N18 G01X44 Z-80

N19 G01 X55 Z-80

N20 G00 X100 Z100

N21 M05

N22 T= "FINISHING _ T35A"

N23 S100 M03

N24 G00 X55 Z5

N25CYCLE940(16,-34, "A",1,1,0,1,1,,,,, 45,70,1,0.1,0. 1,13,,,,2,1100)

N26 G00 X100 Z100

N27 M05

N28 T= "THREADING_1.5"

N29 S500 M03

N30 G00 X55 Z 5

N3 CYCLE199 (0 16,32 ,,1, 2, 0.9201,0,30,0,4,0,1.5,1110101,4,2,0.3,0.5,0,0,1,0,0.53122,1,, "ISO_METRIC", "M10",102,1)

N32 G00 X100 Z100

N33 M05

N34 T= "PLUNGE_CUTTER_3 A"

N35 G00 X155 Z5

N36 CYCLE92 (55,-81,10,-2,1.5,1,2000,1000,3,0.1,2,200,032,0,,2,0)

N37 G00 X100 Z100

N38 M05

N39 M30

Fig 3 **STOCK REMOVAL**

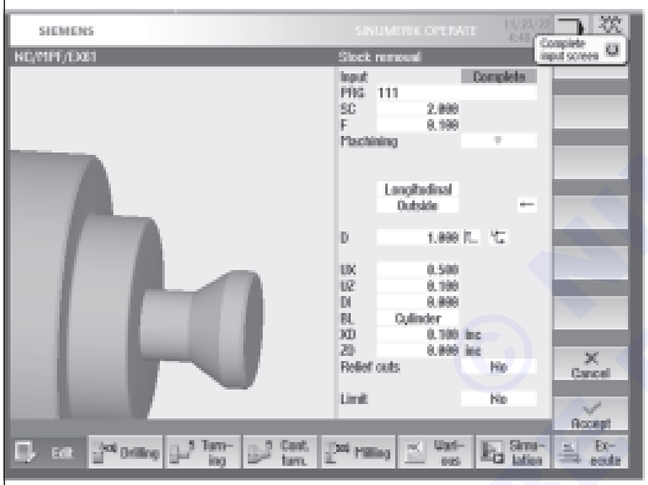


Fig 4 **UNDER CUT THREAD**

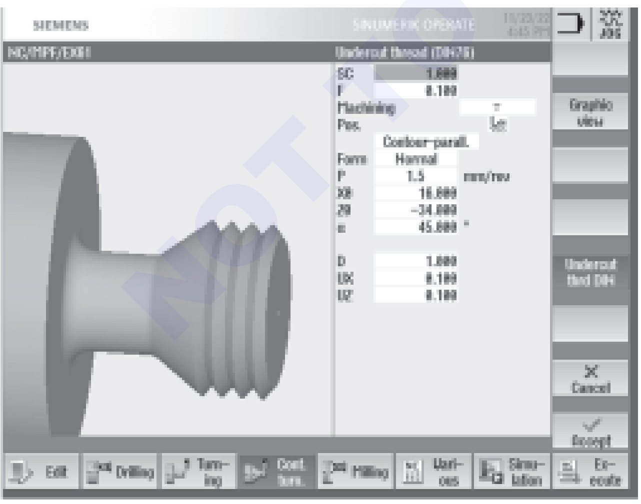


Fig 5 **THREAD LONGITUDINAL**

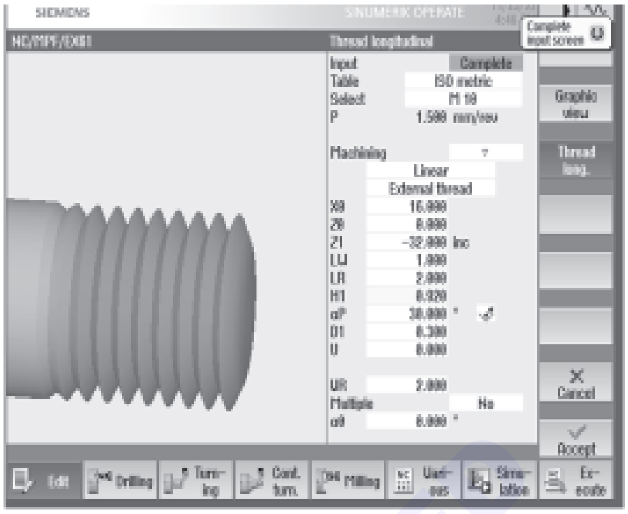


Fig 6 **CUT OFF**

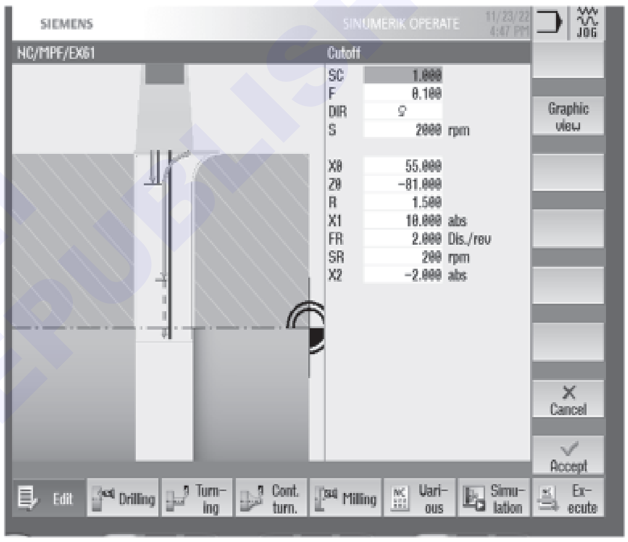
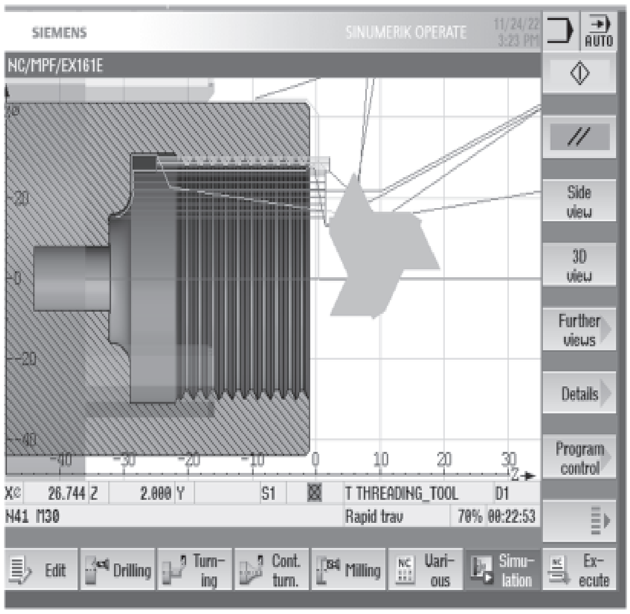


Fig 7 **INTERNAL OPERATION**

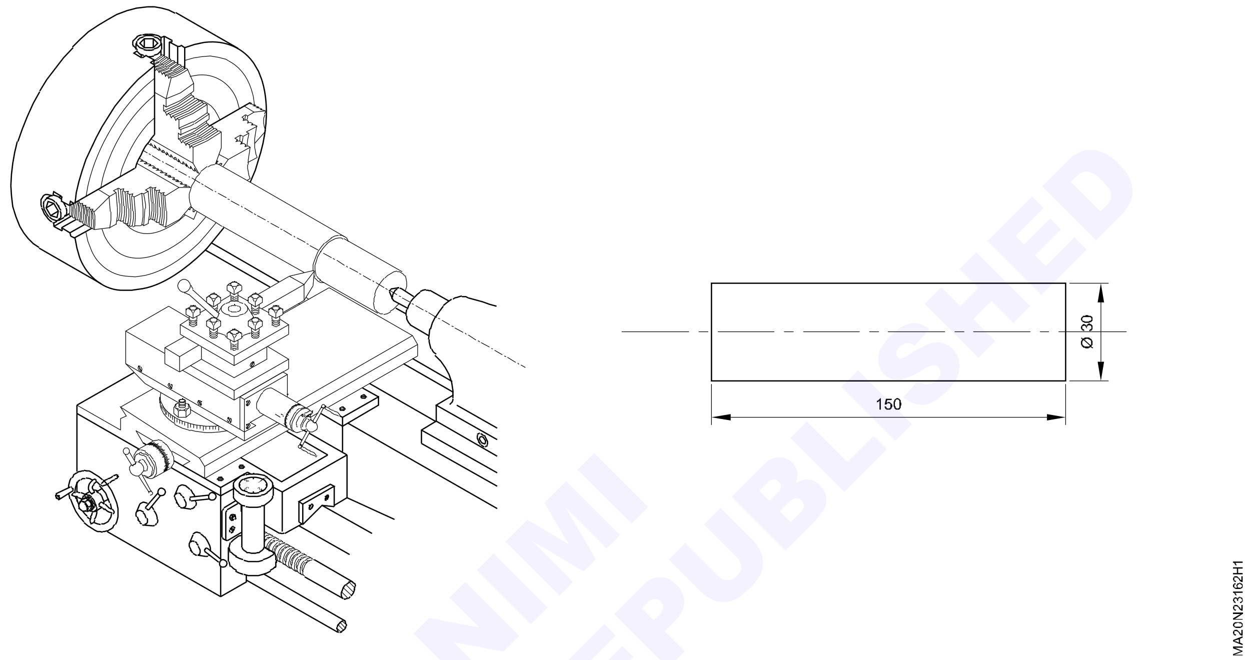


```
SUB PROGRAM 222
N1 G01 X30 Z0 F0.1
N2 G01 X27.744 Z-2
N3 G01 X27.744 Z-29
N4 X22 Z-29
N5 G02 X16 Z-32 CR=3
N6 G01 X16 Z-44
N7 M17
SUB PROGRAM 333
N1 G01 X30 Z0 F0.1
N2 G01 X27.744 Z-2
N3 G01 X27.744 Z-29
N4 X22 Z-29
N5 G02 X16 Z-32 CR =3
N6 G01 X16 Z-44
N7 M17
N1 G54 G90 G95 G71
N2 WORKPIECE(,,, "CYLINDER",0,-1,-80,-15,44)
N3 T= "FINISHING_T35 A"
N4 S100 M03
N5 G00 X-1 Z5
N6 G01 X-1 Z0 F0.1
N7 G01 X40 Z0
N8 G01 X44 Z-2
N9 G01 X44 Z-10
N10 G00 X100 Z100
N11 M05
N12 T= "CUTTER_8"
N13 S100 M03
N14 G00 X0 Z55
N15  CYCLE83(50,0,10,,-44,,5,90,0.6,0.6,15,  0,0,5,2,
0.6,1.6,0,1,11211111)
N16 G00 X100 Z100
 N17 M05
N18T="FINISHING_T35I"
N19S1000M03
N20G00X15Z5
N21CYCLE2("222",0,,)
N22CYCLE952("222",,"",2102311,0.1,1,2,1,0.1,0.1,0,0,0.1,0,
1,0,0,,,,,2,2,,,0,1,,0,12,1100010,1,0)
N23 G00 X100 Z100
N24 M05
N25 T="PLUNGE_CUTTER_3 I"
N26 S100 M03
N27 X15 Z10
N28 G01 X23 Z-23
N29 X23 Z-23
N30 X31 Z-25
N31 X31 Z-29
N32 X22 Z-29
N33 G00 X22 Z10
N34 M05
N35 G0 X100 Z100
N36 M05
N37 T="THREADING_TOOL"
N38 S150 M03
N39 G00 X15 Z10
N40CYCLE99(0,27.744,25,,2,0,1.228,0,30,0,5,3,2,111
0102, 4,0.5,0.3,0.5,0,0,1,0,0.708986,1,,,2,1)
N41 G00 X100 Z100
N42 M05
N43 M30
```

Machining long part holding in chuck and tailstock

Objectives : At the end of this exercise you shall be able to
- **set the work place in chuck and supporting with tail stock**
- **Prepare program, verify and execute the program.**

Fig 1

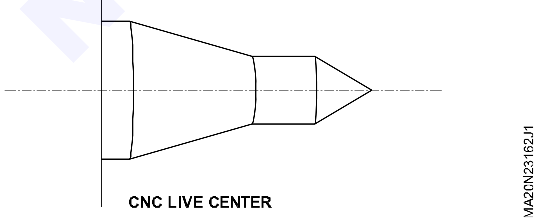


Job Sequence

CNC turning with chuck and tailstock

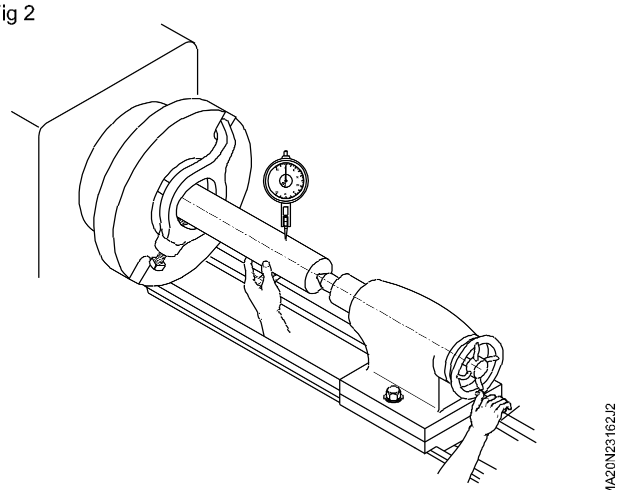
- Hold the work piece in 3 jaw chuck face and turn to remove the rough material for 40mm length.
- Reverse the work piece and face and maintain the total length of the work piece.
- Center drill the work piece at required depth.
- Hold the work piece at pre turned diameter
- Set the tailstock with CNC live centre (Fig 1)
- Move the tailstock closer to the work piece

Fig 1



- Move the tailstock with MPG so that the CNC live centre is supporting the work piece with out play and the chuck should rotate with out any friction.
- Lock/clamp the tail stock in position.
- Check for the workpiece deflection while supporting the workpiece using dial test indicator. (Fig 2)

Fig 2



- Set the work zero and tool offset
- Write the program for the given job
- Enter the program and verify it
- Execute the program in auto mode operation.
- Check the diameter and the parallel of work piece.
- Unclamp the tailstock and move away from the workpiece
- Remove the work from the chuck.
- Reverse it and turn the left out portion.

Skill Sequence

CNC lathe work holding - when to use tailstock and steady rest

Objective: This shall help you to

- **select the work piece supporting device depending upon length and diameter ratio.**

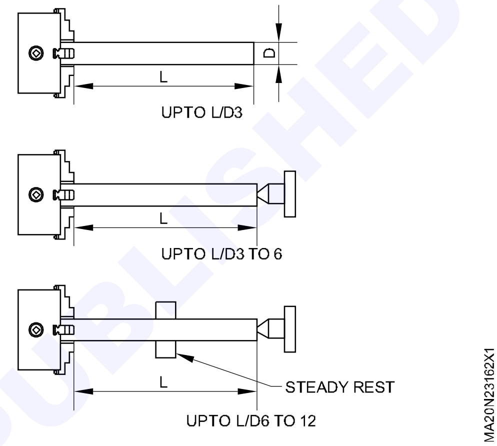
The part held in the chuck is a cantilever, and the radial cutting force of the tool tends to bend the part. You use a tailstock or steady rest to prevent the bending. Here's a CNC lathe work holding thumb rule that tells you when you can hold in a chuck, when to use tailstock,and when to use a steady, based on the L/D (Length to Diameter) ratio of the part.

Chuck only: Use if L/D is less than 3 . You can go up to L/D 5 with reduced cutting parameters, which reduce the cutting force.

Tailstock: Use if L/D is between 3 and 6 . You can go up to L/D 10 with reduced cutting parameters.

Steady rest: Use if L/D is between 6 and 12 . You can go up to L/D 20 with reduced cutting parameters.

Fig 1



Block search operation

Objective : At the end of this exercise you shall be able to
- **start from the interruption due to power shutdown or tool breakage.**

Job Sequence

Following operations are to be carried out when machining is interrupted due to a reason such as a damaged tool / power failure and is restarted.

- Remove the cause which interrupted the machining
- Locate the interrupted point in the program
- Restore the machine to the suitable status including the auxillary functions like coolant on, tool call and spindle status)
- Move the tool to the position suitable for restarting the machine
- Resume automatic operation from the interrupted block or before several blocks

Steps on siemens control

- Select the desired program
- Press reset mode
- Select the desired search mode
- Press "Block search" soft key
- Place the cursor on a particular program block
- Press the "Start search" softkey.
- The search starts. Your specified search mode will be taken into account

The current block will be displayed in the "Program" window as soon as the target is found.

Note: If the set point position is not approached automatically the follow the below steps.

- If the cycle start outputs the auxiliary function collected during the search. The program is in stop state.
- Press "over store" softkey . the overstore window opens
- Enter the required data and NC block. Press "cycle start" key observe execution in "overstore" window press the “back” soft key.
- Select jog, REPOS mode
- Select the axes to be traversed one after another (x,z)
- Press the "+" or '-' for the relevant direction till the axes are moved to the interrupted position.
- Press “cycle start” key
- Now the process is continued from the defined location.

Note: Generally, the program restart in easy steps

- Press the soft key 'Restart' restart point list screen will open
- Select a block from list, and press “SEARCH EXEC” key to rstart. It display the program where the program was interrupted
- Press "Cycle start" key
- Automatic restoration of model information and axes moving to restart point and resumes the automatic operation from the interrupted block.

Changing wear offset

Objectives : At the end of this exercise you shall be able to
- **determine the amount of wear in x and z direction**
- **input /change the wear offset values in x and z axes.**

Job Sequence

Determine the wear offset

> **Adjusting the tool wear offset is necessary because, as the cutting tool wear, the dimension they are machining may increase or decrease. The tool wear offset process allows you to change the position of the cutting tool compensate for the tool wear.**

- All 'X' setting will normally have diameter value and 'z' axis setting will be length.
- If the machined diameter is larger then the programed dimension, the wear offset is in minus direction and vice versa.
- Measure the size of the work and compare with the programed values. Tool number is 3

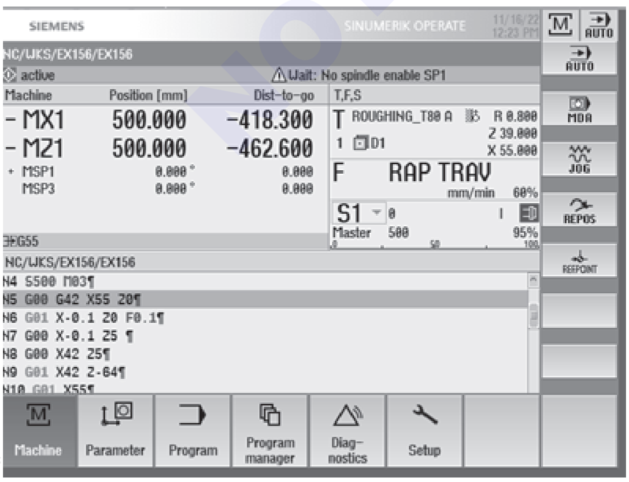
  Example programed value in x 30.00

  z 15.00
- The measured value is diameter 30.05
- The measured value in length 15.04
- The difference in diameter is 0.05
- The different in length is 0.04
- Since the both values are positive
- Enter this values x -0.05 and Z-0.04

Changing the wear offset value in system in siemens control

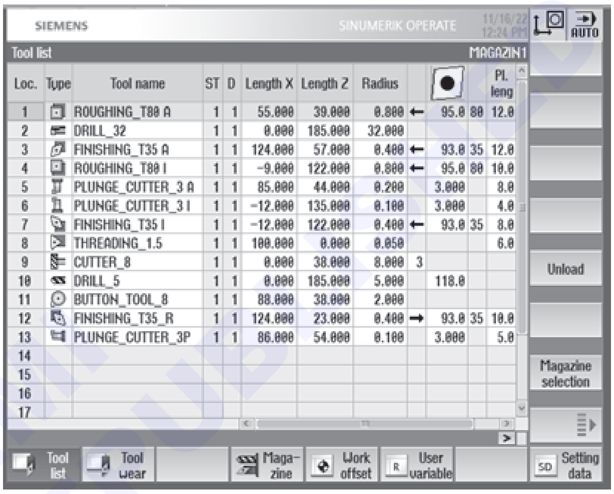
- Press machine key the screen will display as in Fig 1

Fig 1



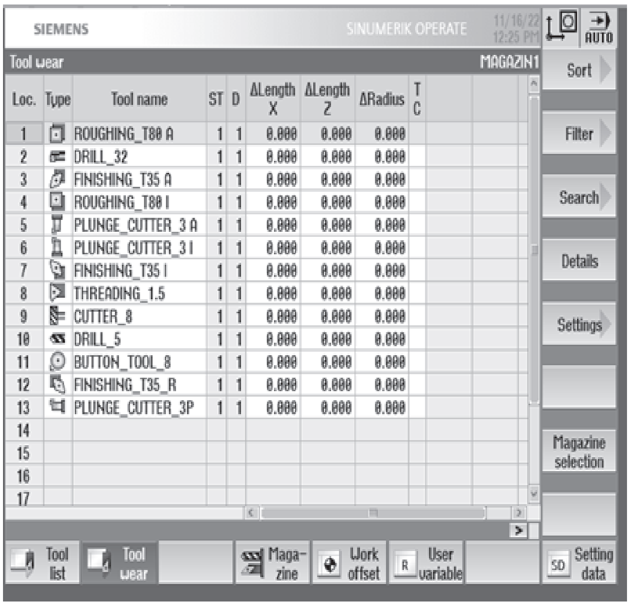
- Press parameter. The screen will display as in Fig 2. will display the tool list,

Fig 2



- Press tool wear key and it leads to wear offset screen as shown in Fig 3.

Fig 3



- Position the curser at tool number 3 and input the determined values in x as negative, and z as negative (x -0.005,z-0.04)

The system will automatically add or subract the from existing wear offset values. It is zero values then x -0.05 and z-0.04 will be active.

Fanuc control

The word offset with two adjectives

1 OFFSET -GEOMENTRY

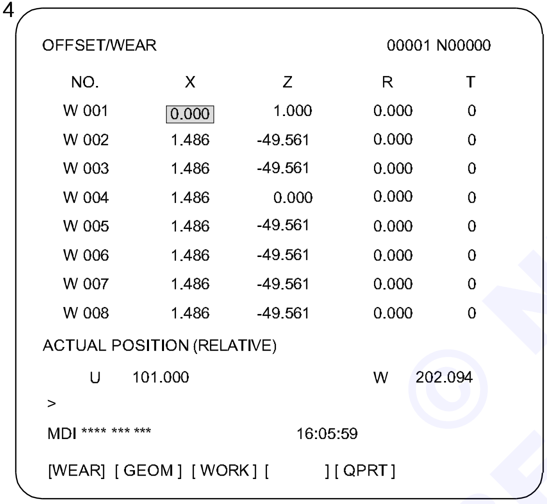
2 OFFSET -WEAR

- Geometry offset is the tool offset matches with turret station number.

To enter the tool offset/geomentry

- Press function key.
- Press chapter selection soft key [OFFSET] or press several times until the tool compensation screen is displayed.
- Pressing soft key [GEOM] displays tool geometry compensation values. (Fig 4)

Fig 4



MA20N23164H4

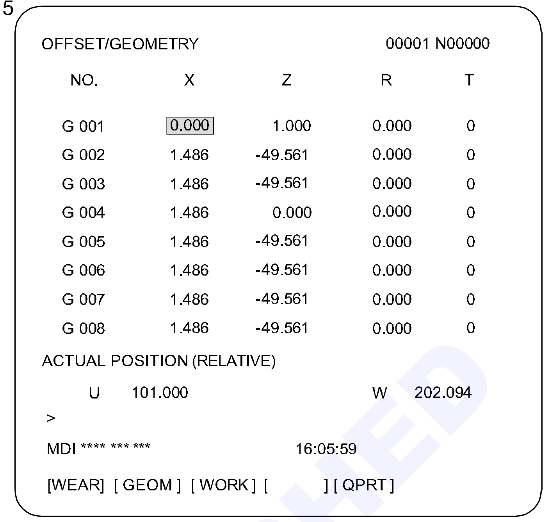
WITH TOOL GEOMETRY OFFSET

- Place the cursor at the required number example tool number 7 means G007.
- Enter the dermined wear value and input the value for X and Z.

For wear input

- Pressing soft key [WEAR] displays tool wear compensation values. (Fig 5)

Fig 5



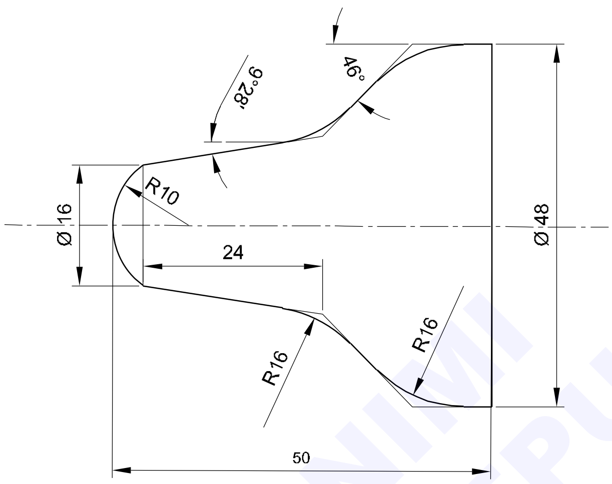
MA20N23164H5

WITH TOOL WEAR OFFSET

- Move the cursor to the compensation value to be set or changed using page keys and cursor keys, or enter the compensation number and press soft key [NO.SRH].
- To set a compensation value, enter a value and press softkey [INPUT].
- To change the compensation value, enter a value to add to the current value (a negative value to reduce the current value) and press soft key [INPUT]. or, enter a new value and press softkey [INPUT].
- TIP is the number of the virtual tool tip (see programming).
- TIP may be specified on the geometry compensation screen or on the wear compensation screen.

Contour program

Objectives : At the end of this exercise you shall be able to
- **write the contour program for a given drawing**
- **verify the program by simulation and dry run method**
- **execute the program in auto mode**
- **check the dimension of the machined part.**



Job Sequence

- Write the CNC program using contour guide lines for preparing contour program
- Prepare a new part program with suitable name first select the workpiece dimension
- Select the tool
- Spindle on command with suitable spindle speed
- Call the cycle 62 (give the suitable name)
- Select cycle 952 (enter the relavant parameter)

Creating contour

- Enter the contour new
- Enter the name as in cycle 62
- Enter the relavant details in countour program
- Check the program in the left menubar
- End the contour program
- Move the tool away from the work piece
- Stop the spindle
- End the main part program
- Verify the program by simulating on the software
- Enter the program in CNC machine
- Verify tool offset, work offset,
- Verify the program by dry run method
- Execute the program in auto mode.
- Check the dimension

-	Ø 50 X 75		ALUMINIUM	-	-	2.3.165
NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.
SCALE 1:1	CONTOUR/BLUE PRINT PROGRAM				DEVIATIONS ± 0.02	TIME :
					CODE NO. MA20N23165E1	

Note: contour programming (siemens) used for machining simple or complex contours with the "contour turning" cycle. A contour comprises separate contour elements comprises of chamfers, radius, under cuts or tangential transitions between the contour element.

Instructor shall demonstrate on how to create contour programming

Contour Program

Enter the values as in Fig 1 to 6.

Fig 1

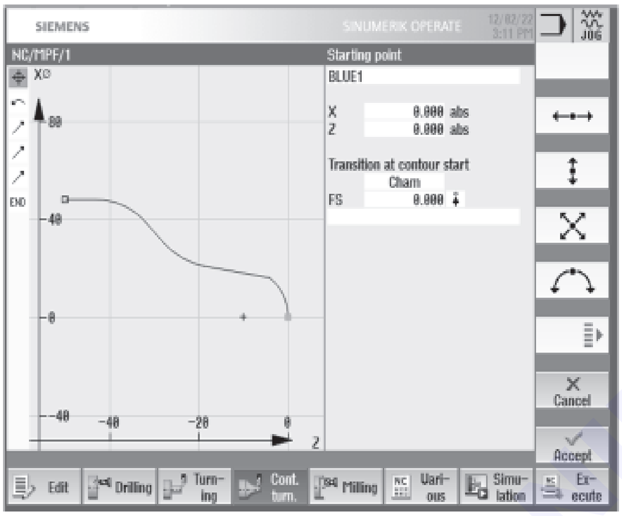


Fig 2

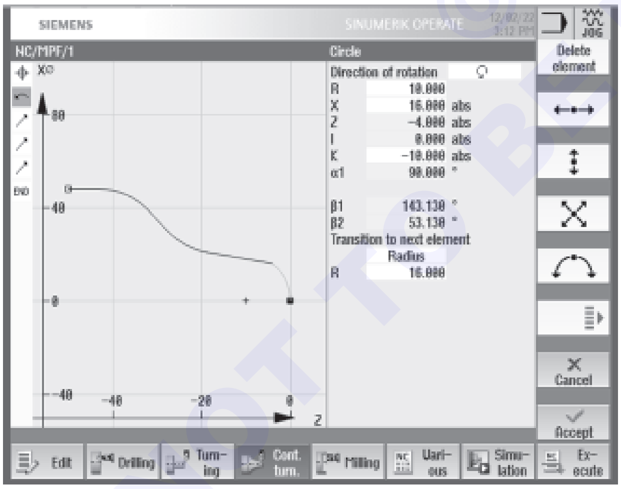


Fig 3

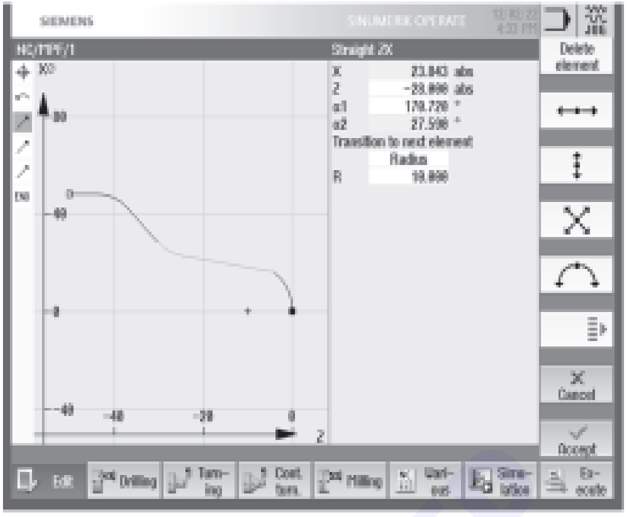


Fig 4

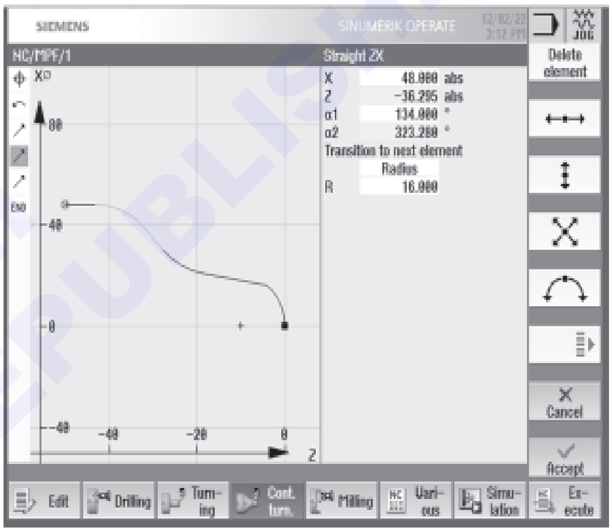


Fig 5

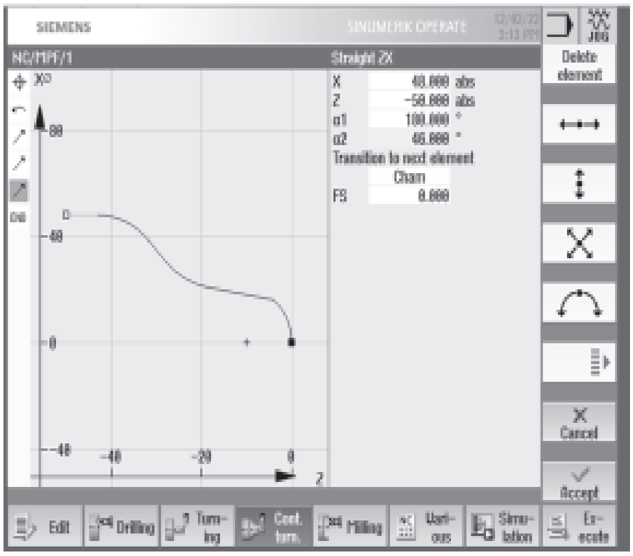
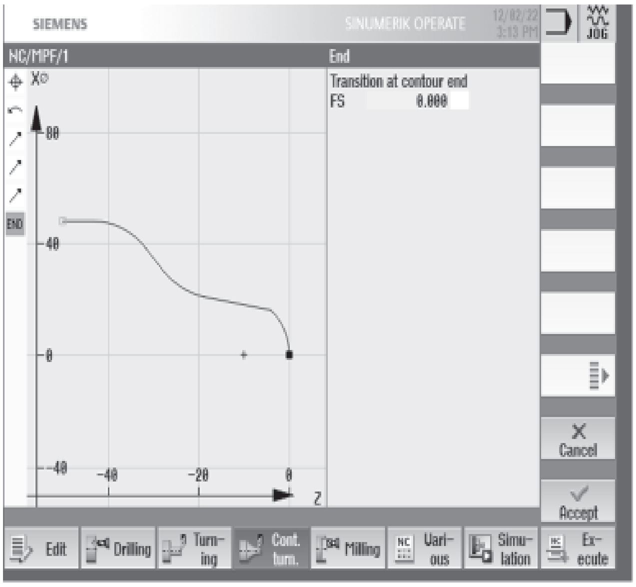
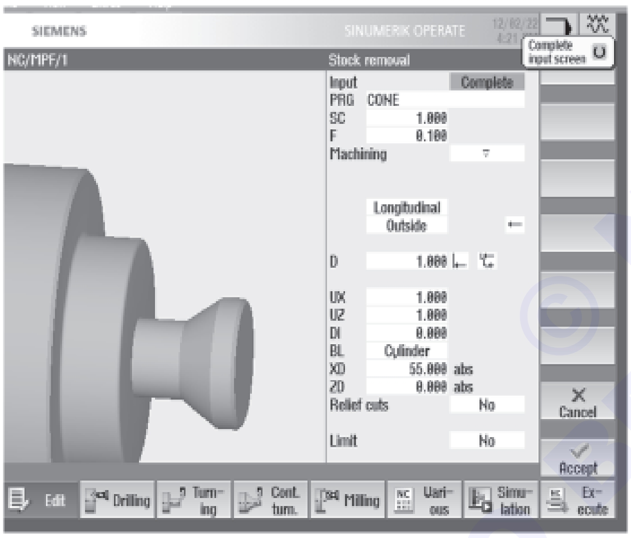


Fig 6



Stock removal details. (Fig 7)

Fig 7



COUNTOURPROGRAM

N1 WORKPIECE(,,,"CYLINDER",192,0,-100,-75,50)

N2 T="FINISHING_T35 A"

N3 S500 M03

N4 G00 X55 Z5

N5 F0.1

N6 CYCLE62("BLUE1",1,,)

CYCLE952("CONE",,"",2201311,0.1,1,2,1,0.1,0.1,1,1,0.1,0,1,55,0,,,,,

2,2,,,0,1,,0,12,10,1,0)

E_LAB_A_BLUE1: ;#SM Z:4

;#7__DlgK contour definition begin - Don't change!;*GP*;*RO*;*HD*

G18 G90 DIAM90;*GP*

G0 Z0 X0 ;*GP*

G3 Z-4 X16 K=AC(-10) I=AC(0) ;*GP*

G1 Z-24 X22.536 RND=10 ;*GP*

Z-36.295 X48 RND=10 ;*GP*

Z-50 ;*GP*

;CON,V64,2,0.0000,4,4,MST:3,2,AX:Z,X,K,I,TRANS:0;

GP;*RO*;*HD*

;S,EX:0,EY:0,ASE:90;*GP*;*RO*;*HD*

;ACCW,DIA:10/0,EY:16,CX:-10,RAD:10;*GP*;*RO*;*HD*

;LA,EX:-24,ASE:170.72;*GP*;*RO*;*HD*

;R,RROUND:10;*GP*;*RO*;*HD*

;LA,EY:48,ASE:134;*GP*;*RO*;*HD*

;R,RROUND:10;*GP*;*RO*;*HD*

;LA,EX:-50,EY:48;*GP*;*RO*;*HD*

;#End contour definition end - Don't change!;*GP*;*RO*;*HD*

E_LAB_E_BLUE1:

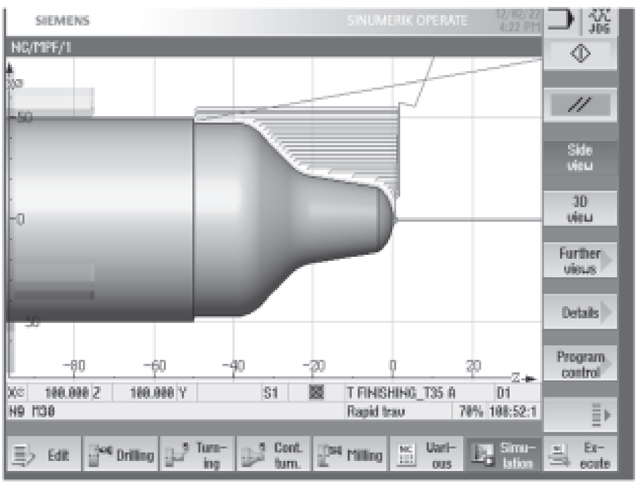
N7 G0 X100 Z100

N8 M05

N9 M30

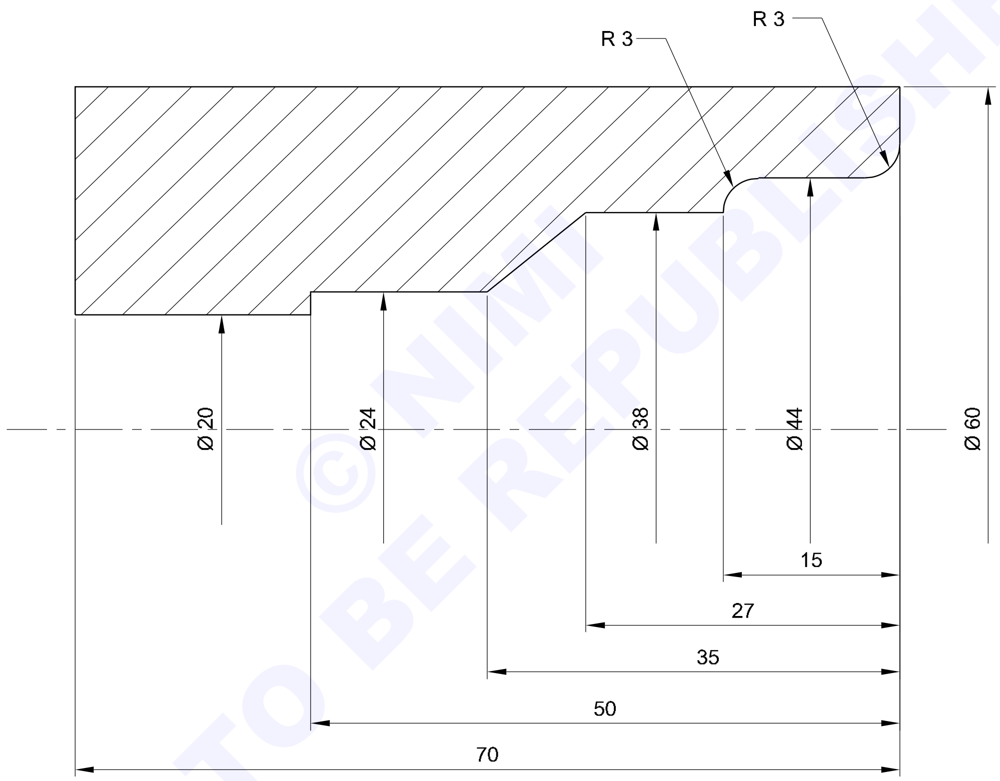
Simulated view (Fig 8)

Fig 8



Drilling and boring operation

Objectives : At the end of this exercise you shall be able to
- **prepare the CNC program using drilling cycle**
- **prepare the program using boring operation with stock removal cycles**
- **verify the program**
- **execute the program in auto mode and check the dimensions.**



1	PRE MACHINED Ø 60 X 70	--	Fe310	--	--	2.3.166
NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.
SCALE 1:1	DRILLING AND BORING			DEVIATIONS	TIME :	
				CODE NO. MA20N23166E1		

Job Sequence

Prepare the CNC program in Fanuc control

- Write program for facing and turning to a diameter 60 mm to a length of 70mm
- Write program for drilling (slot drill/ U drill) dia18mm to a length of 70 mm using G83 cycle
- Write the program for roughing internal stock removal using cycle G70
- Write the program for finishing internal stock removal using cycle G71
- Verify the program in simulator
- Transfer the program to machine/enter the program manually and verify by machine simulation
- Set the tool in turret asper the program
- Set the work piece in chuck projecting 75 mm
- Measure work offset and tool offset enter in relevant area
- Verify the tool offset and work offset
- Verify the program on machine by shifting work offset in single block mode
- Reset the work offset
- Execute the program in auto mode
- Verify the dimensions, if any variation in dimension correct it by wear offset method
- Remove the work piece and clean the machine
- Sample Fanuc program is provided

Note:- Trainees should develop the same part program in simens control and get it verified by your instructor.

Program in Fanuc

01234; (FANUC -B -G CODE)

NI G80 G40;

N2 G18 G90 G21 GS4 G99 G97;

N3 T0101 S1000 M04; (TURNING TOOL)

N4 G00 X70.00 Z0.0;

NS G01 X-0.1 Z0.0 F0.1;

N6 G00 X-1.0 ZS.0;

N7 G00 X60 ZS.0;

N8 G01 X60 Z-70.0;

N9 G01 X65 Z-70.0;

N10 M05

N11 G28 U0.0 W0.0 T0100;

N12 T0303 S800 M03; (SLOT DRILL DIA 18mm)

N13 G00 X0.0 Z50.0;

N14 G90 G98 G83 X0.0 Z-75.0 R5.0 QI0.0 P100 F0. I;

N15 G28 U0.0 W0.0 M05 T0300;

N16 T0606 S800 M04; (BORING TOOL)

N17 G00 X16.0 Z50.0;

N18 G71 U1.0 R1.0;

N19 P20 Q28 U-0.5 W0.5 F0.1;

N20 G01 X51.0 Z0.0;

N21 G02 X44.0 Z-3.0 10.0 K-3.0;

N22 G01 X44.0 Z-12.0;

N23 G03 X38.0 Z-15.0 1-3.0 K0.0;

N24 G01 X38.0 Z-33.0;

N25 G01 X24.0 Z-35.0;

N26 G01 X24.0 Z-S0;

N27 G01 X20.0 Z-S0.0;

N28 G01 X20.0 Z-70.0;

N29 G00 X100 Z100 M05

N29 G28 U0.0 W0.0 T0600;

N30 T0505; (FINISHING TOOL)

N31 S1500 M04;

N31 G70 P20 Q 28 F0.5;

N32 G00 X100 Z100 M05;

N34 G28 U0.0 W0.0 T0500;

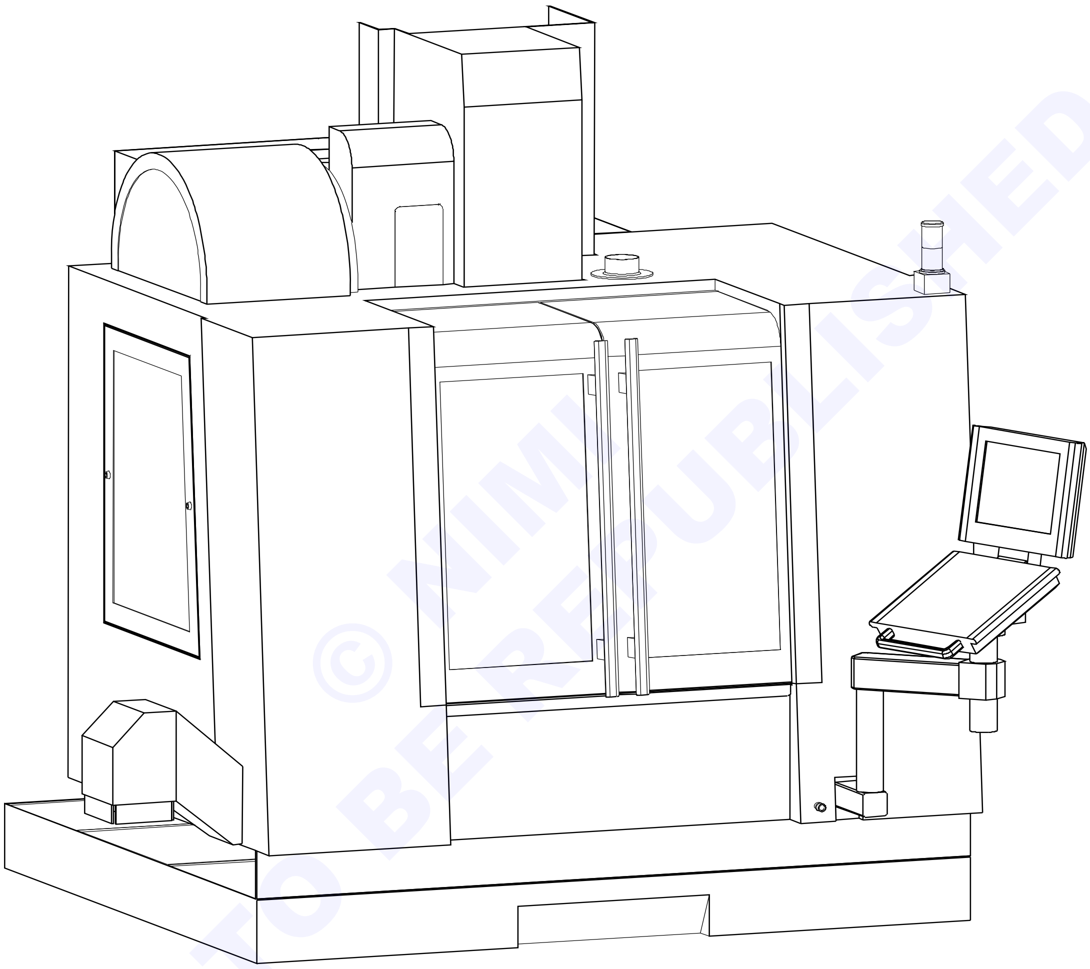
M30;

Identify CNC milling elements and their functions, on the machine

Objectives : At the end of this exercise you shall be able to
- **identify the elements of a VMC**
- **list out the Functions of each Part of a VMC.**

1 Identification of Parts

Fig 1



MA20N24167H1

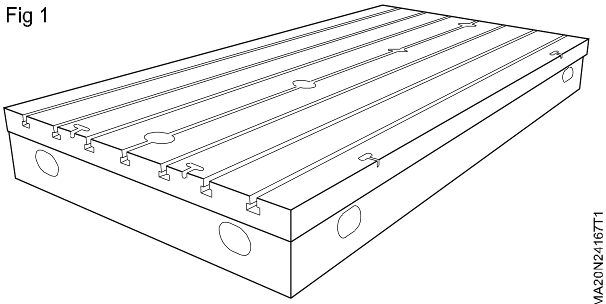
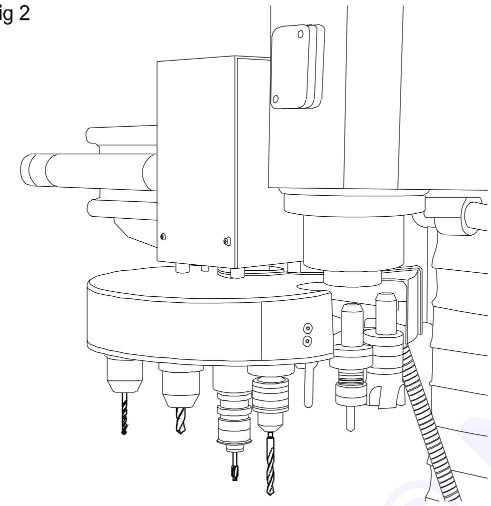
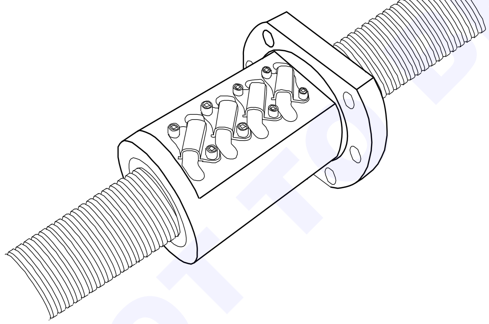
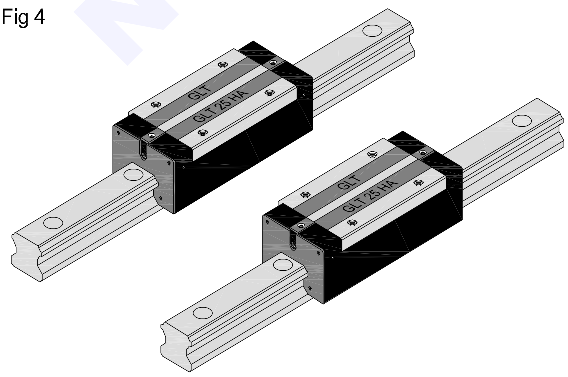
Note : Some parts are internal can be seen in the machine.

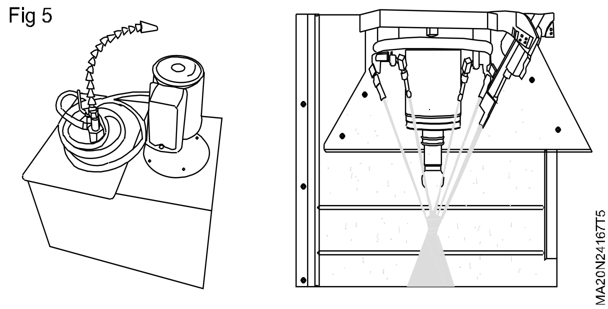
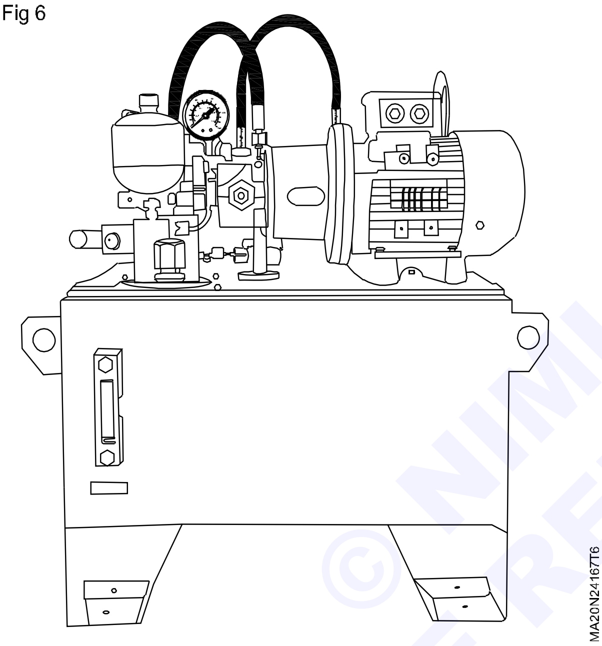
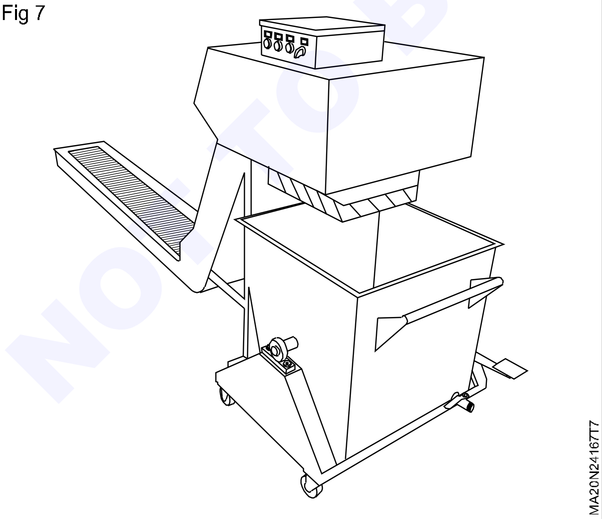
Job Sequence

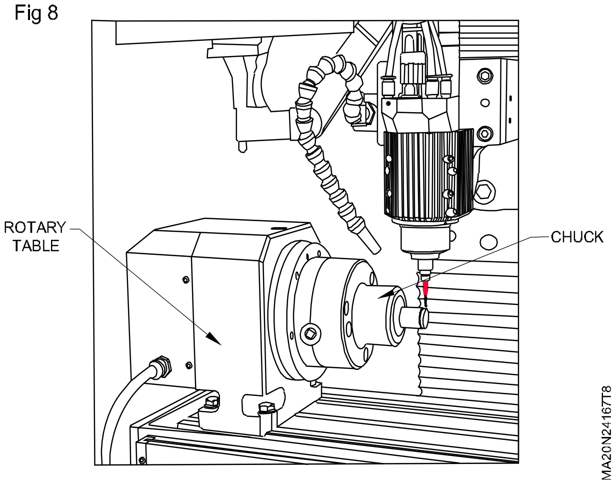
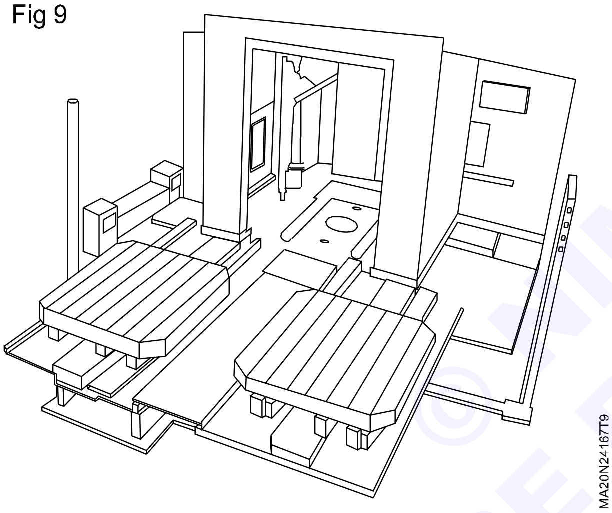
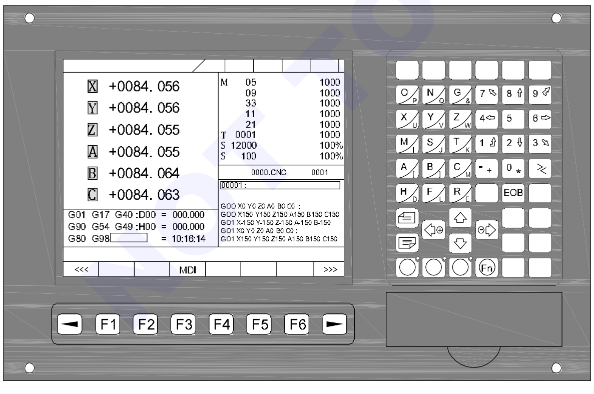
- Name the parts of a CNC milling/VMC machine record it in table 1.
- State the function of each part in the table1, and get it checked by trainees

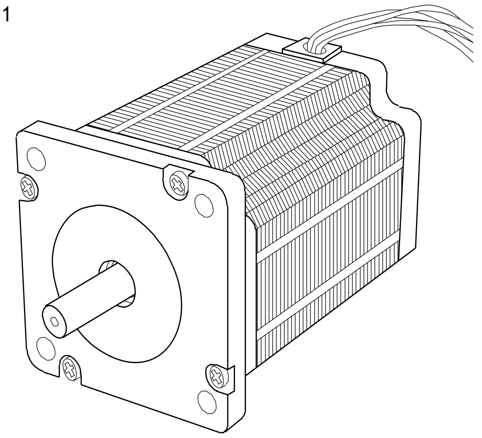
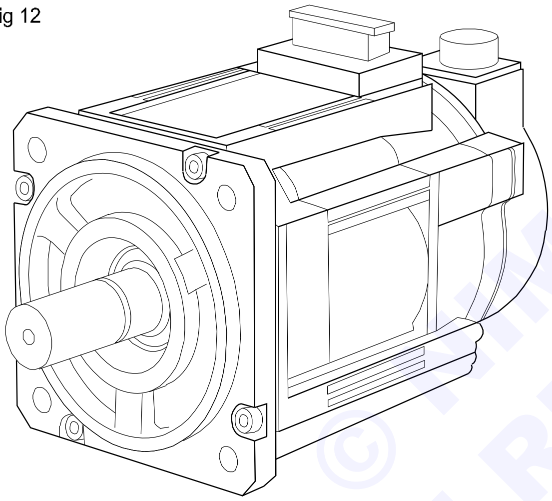
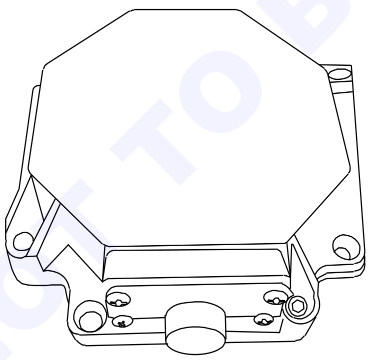
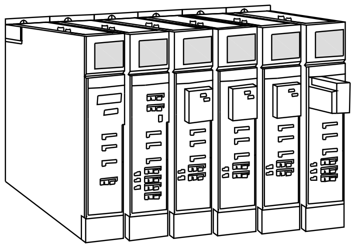
Note: Instructor shall demonstrate all the parts and their function.

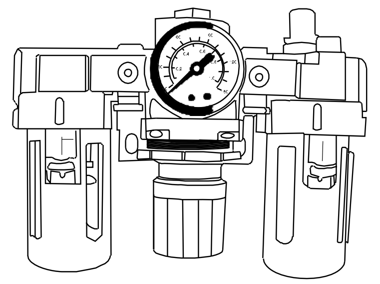
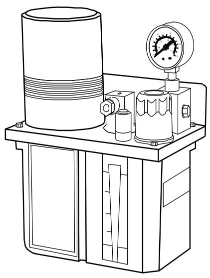
Table 1

Sl.No.	Fig	Name of the Part	Function
1	Fig 1 		
2	Fig 2 		
3	Fig 3 		
4	Fig 4 		

Sl.No.	Fig	Name of the Part	Function
5	Fig 5 		
6	Fig 6 		
7	Fig 7 		

Sl.No.	Fig	Name of the Part	Function
8	Fig 8 		
9	Fig 9 		
10	Fig 10 		

Sl.No.	Fig	Name of the Part	Function
11	Fig 11 		
12	Fig 12 		
13	Fig 13 		
14	Fig 14 		

Sl.No.	Fig	Name of the Part	Function
15	Fig 15 		
16	Fig 16 		

Exercise 2.4.168

Working of CNC VMC parts through didactic/simulation

Objectives : At the end of this exercise you shall be able to
- **familiarize parts of CNC VMC using a didactic/simulation**
- **go through the machine parts and element section in the simulator**
- **make a notes on machine elements and submit.**

Note

- The instructor shall demonstrate the working of parts of CNC VMC using didactic/simulator
- The trainees will follow the instructor instructions and go through the parts and machine element section in - software repeatedly
- Make a notes on the each parts and submit to the instructor

— — — — — — — —

Identify machine over travel limit and emergency stop

Objectives : At the end of this exercise you shall be able to
- **identify machine over travel limit switch on CNC-VMC**
- **identify the emergency switch on VMC and practice put on and off.**

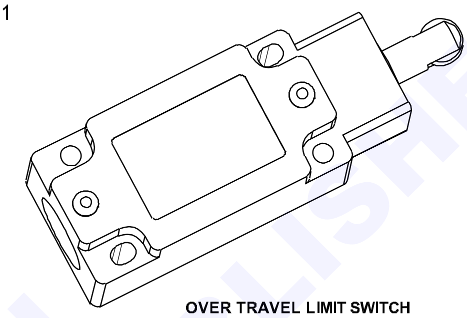
Job Sequence

TASK 1: **Identification of machine over travel limits**

There are two types of over traqvel limit

- Software over travel
- Hardware over travel
- Software over travel can be controlled by the specific parameter
- Hardware over travel limit is controlled by limits switch open the machine axis safe cover
- In axis limit end there will be a fixed limit switch
- One taper dog fixed on movable axis frame
- If the dog pressed the fixed limit switch, over travel alaram appear on the screen
- Fig 1 shows the over travel limit switch

Fig 1



— — — — — — — —

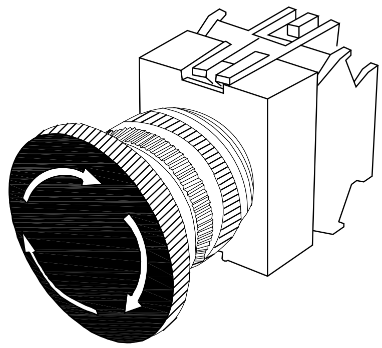
TASK 2: **Identification of Emergency stop bush button**

- Emergency stop buttons are designed in such a manner in which their role is more physical, such as interrupting a power supply to the machine control system. It is a basic big red pushbuttons fixed on Machine control panel.
- Emergency stop pushbutton that has mechanical plastic or metal tabs and grooves internally such that when you push it (interrupting the circuit), it is held in that position until you twist it. They are designed to be large, hard to miss, and easy to push sample is given in Fig 2

  Note: Practice to put off and release the emergency switch.

Caution: Do not try to rotate in anticlock wise direction.

Fig 1



— — — — — — — —

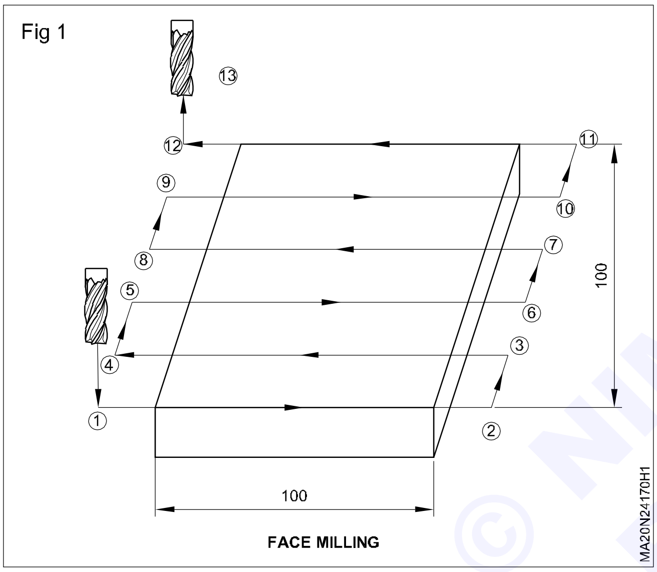
Tool path for milling operation

Objectives : At the end of this exercise you shall be able to
- **prepare the tool path for face milling operation**
- **prepare the tool path for side milling operation**
- **prepare the tool path for pocket milling operations**
- **prepare the tool path for drilling operation.**

Job Sequence

Read and interpret the tool path for the task 1 to 7

TASK 1: **Face milling tool path (Fig 1)**



Read the tool path and interpret the tool movements in face milling

1 Point Tool positioning and Z axis depth 1 mm

2 Point Cutting motion liner X axis

3 Point Cutting motion liner Y axis

4 Point Cutting motion liner X axis

5 Point Cutting motion liner Y axis

6 Point Cutting motion liner X axis

7 Point Cutting motion liner Y axis

8 Point Cutting motion liner X axis

9 Point Cutting motion liner Y axis

10 Point Cutting motion liner X axis

11 Point Cutting motion liner Y axis

12 Point Cutting motion liner X axis

13 Z axis go to safety position

14 Point Spindle stop

15 Point Program rewind

— — — — — — — —

TASK 2: **Tool path for side milling (Fig 1)**

Read and interpret the tool path

1 Point Tool starting positioning and Z axis depth

2 Point Cutting feed X Axis (linear)

3 Point Cutting feed Y Axis (linear)

4 Point Cutting feed CCW X and Y axis with radius

5 Point Cutting feed liner X axis

6 Point Cutting feed FCW X axis and Y axis with radius

7 Point cutting feed linear Y axis

8 Point Cutting feed CCW X and Y axis with radius

9 Point cutting feed linear Y axis

10 Point Cutting feed CCW X and Y axis with radius

11 Point cutting feed linear Y axis

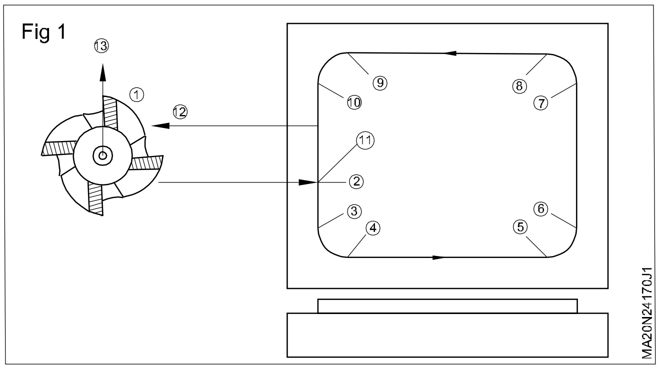
12 Point cutting feed starting position

13 Point Z axis safety position.

Spindle off

Coolant off

Program rewind



— — — — — — — —

TASK 3: **Rectangle pocket tool path (Fig 1)**

Read and interret the tool path

1 Point tool & position and Z axis depth

2 Point Tool cutting feed linear X axis & direction

3 Point Tool cutting feed linear Y axis & direction

4 Point Tool cutting feed linear X axis & direction

5 Point Tool cutting feed linear Y axis & direction

6 Point Tool cutting feed linear X axis - direction

7 Point Cutting feed y axis 0 point

8 Point cutting feed x - direction

9 Point cutting feed Y + direction

10 Point Cutting feed x + direction

11 Point Cutting feed Y - direction

12 Point cutting feed X - direction

13 Point cutting feed Y O position

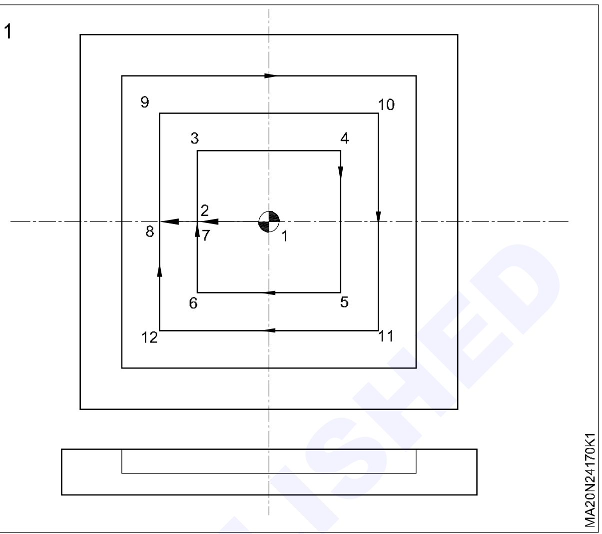
Go to Z axis safe position

Spindle off

Coolant Off

Program rewind

Fig 1



TASK 4: **Circular tool Path (Fig 1)**

Read and interpret the tool path.

1 Point Tool position and Z axis depty

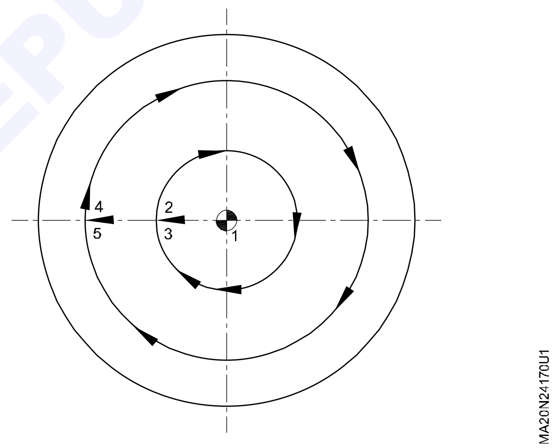
2 Point Tool movement X axis - direction with linear

3 Point Tools movement CW direction with interpretation ending 2 point

4 Point tool movement X axis and direction linear motion

5 Point tool movement CW direction with interpretation ending point 4.

Fig 1

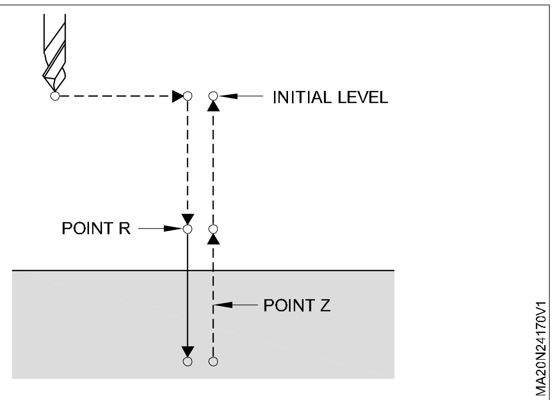


TASK 5: **Tool path for drilling /spot facing/counbter sinking (Fig 1)**

1 Cutting feed is performed to the bottom of the hole. The tool is then retracted from the bottom of the hole in rapid traverse.

Note: for spot facing and counter sinking the tool rotate without 'Z;' movements for a while and retract to R point.

Fig 1



TASK 6: **Tool path for deep hole drilling (Fig 1)**

Read and interpret the tool path

1 Z_ : The distance from point R to the bottom of the hole

2 R_ : The distance from the initial level to point R level

3 Q_ : Depth of cut for each cutting feed

4 Position, drill hole 1, then return to point R

5 Position, drill hole 2, then return to point R

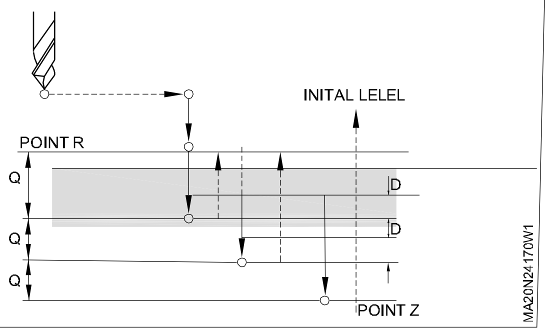
6 Position, drill hole 3, then return to point R

7 Position, drill hole 4, then return to point R

8 Position, drill hole 5, then return to point R

9 Position, drill hole 6, then return to the initial level.

Fig 1



— — — — — — — —

TASK 7: **Tool path for tapping (Fig 1)**

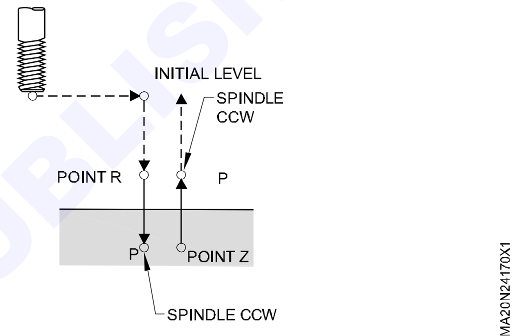
Read and interpret the tool path for tapping

1 Z_ : The distance from point R to the bottom of the hole.

2 R_ : The distance from the initial level to point R level.

3 Q_ : Depth of cut for each cutting feed.

Fig 1



— — — — — — — —

TASK 8: **Tool path for boring and reaming (Fig 1)**

Read and interpret the tool path for boring

1 R - Retract

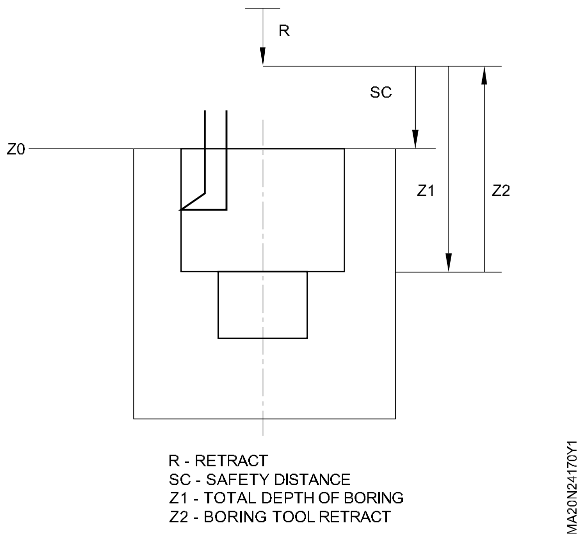
2 SC - Safety distance

3 Z1 - Total depth of boring

4 Z2 - Boring tool retract

Note: The spindle stop rotating while reaching R level

Fig 1



— — — — — — — —

TASK 9: **Tool path for fine boring (Fig 1)**

Read and interpret the tool path.

1 Z_ : The distance from point R to the bottom of the hole

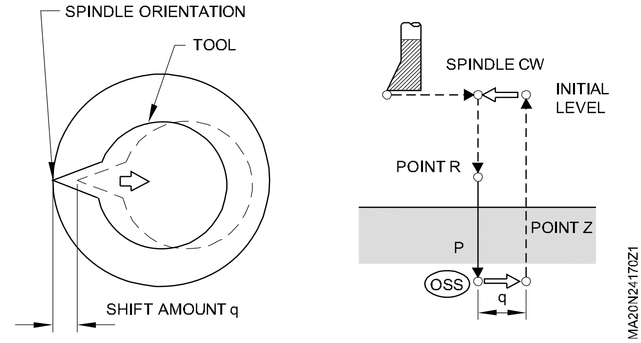
2 R_ : The distance from the initial level to point R level

3 Q_ : Shift amount at the bottom of a hole

4 P_ : Dwell time at the bottom of the hole

5 The fine boring cycle bores a hole precisely. When the bottom of the hole has been reached, the spindle stops, and the tool is moved away from the machined surface of the workpiecce and retracted.

Fig 1



Capital Goods & Manufacturing **Exercise 2.4.171**
Machinist - CNC Milling (VMC- Vertical Milling Center)

Identify common tools, tool holders and inserting

Objectives : At the end of this exercise you shall be able to
- **identify the cutting tool in VMC**
- **identify the tool holders used in VMC**
- **read and interpret the insert size and shapes.**

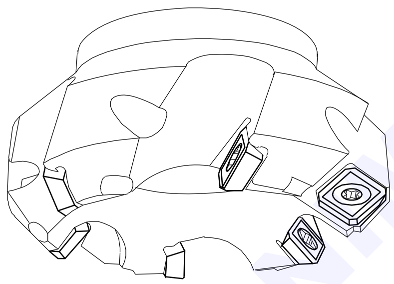
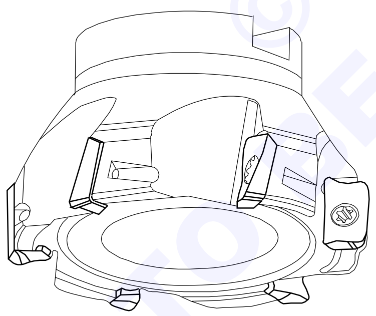
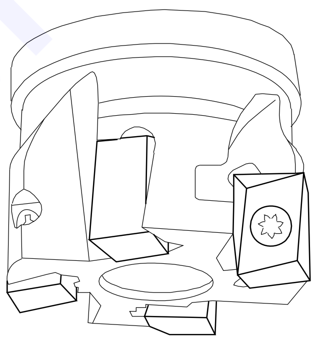
Job Sequence

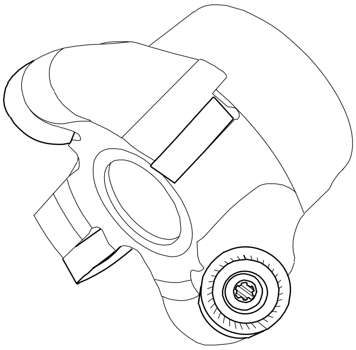
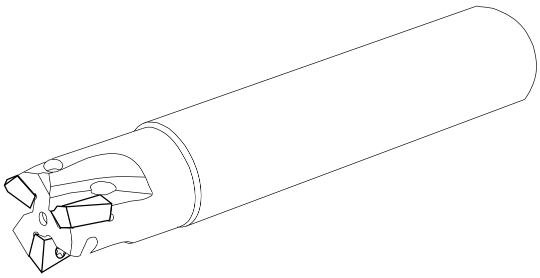
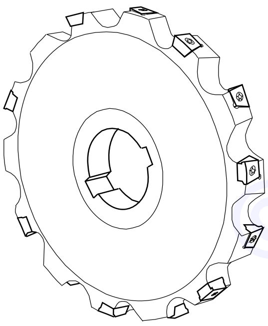
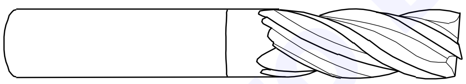
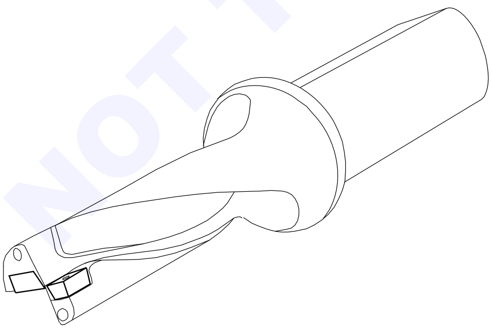
TASK 1: **MIlling cutter**

- Observe the cutting tools listed in table 1 and write the name of the cutting tool in Table 1.

Table 1

Milling cutters

S.No.	Fig	Name of the milling cutter
1		
2		
3		

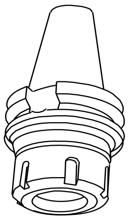
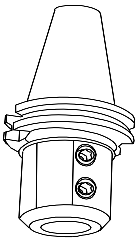
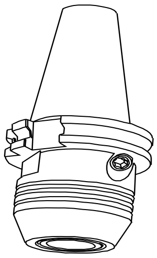
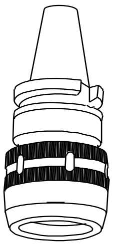
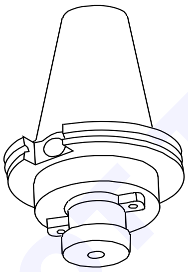
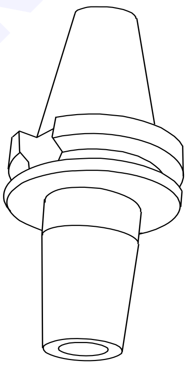
S.No.	Fig	Name of the milling cutter
4		
5		
6		
7		
8		

— — — — — — — —

TASK 2: **Tool holders**

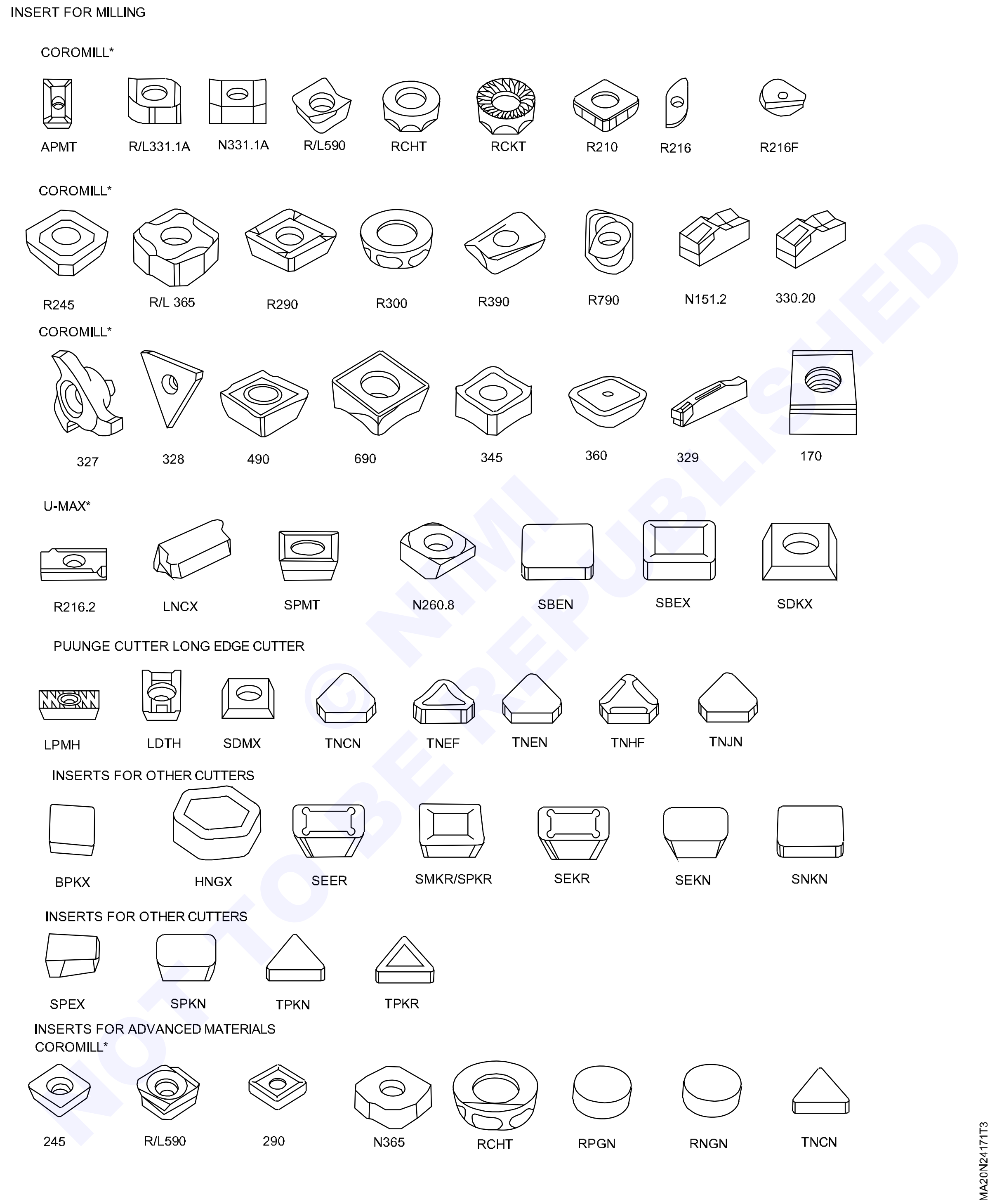
- Observe the tool holders listed in table 2 and write the name of the tool holder and mention its purpose.

Table 2

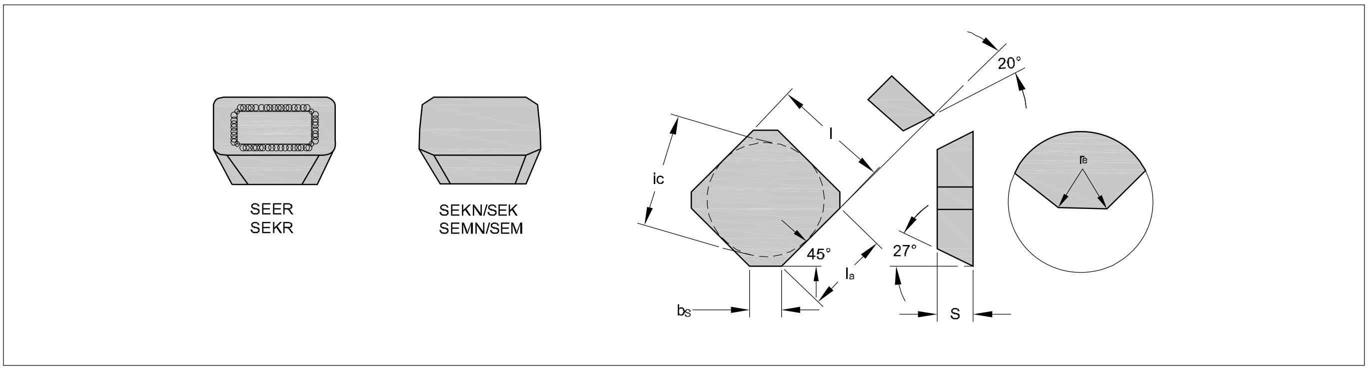
Sl.No.	Fig	Name of the Holder	Purpose
1			
2			
3			
4			
5			
6			

TASK 3: **Milling**

- Read the content provided for insert and interpret the meaning. Write the insert, specification for face mill 45° to mill steel.



Insert specification face milling 45°

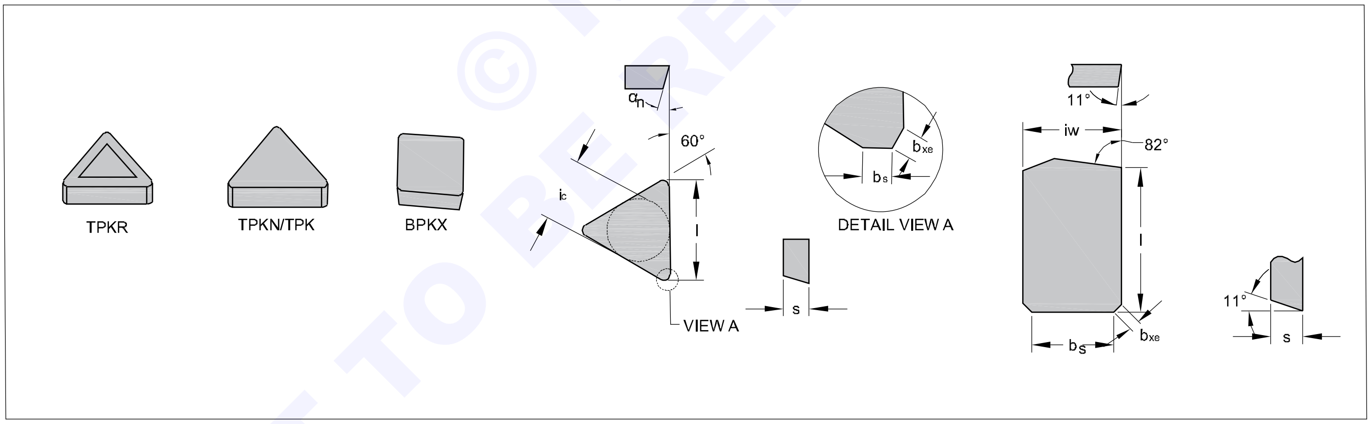


Dimensions, mm (inch)

Size	*iC*	l_a	*l*
12	12.7 (.500)	9.6 (.378)	12.7 (.500)
15	15.9 (.625)	12.7 (.500)	15.9 (.625)

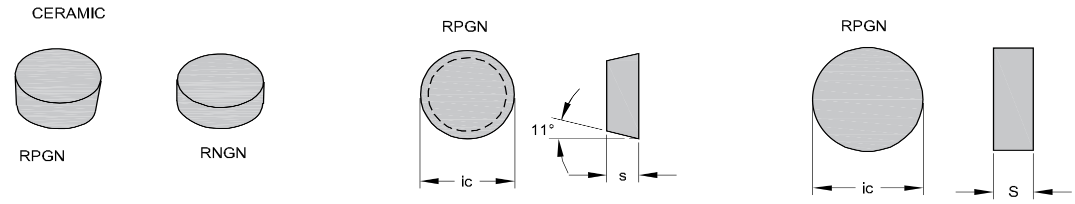
	Insert size	Ordering code	P GC 2030	P GC 2040	P GC 3040	P GC 4230	P GC 4240	P CT 530	M GC 2030	M GC 2040	M GC 4230	M GC 4240	M CT 530	K GC 3020	K GC 3040	K GC 4230	K GC 4240	K - H13A	N CT 530	N - H13A	S GC 2030	S GC 2040	S - H13A	H GC 3040	s mm	s in.	b_s mm	b_s in.	$r_ε$ mm	$r_ε$ in.	ANSI
Medium	12	SEKR 12 03 AZ-WM				☆	☆																		3.18	.125	1.6	.063	1.5	.059	SEKR 12 03 AZ-WM
		SEKR 12 04 AZ-WM				☆	☆	☆						☆					☆						4.76	.188	1.6	.063	1.5	.059	SEKR 12 04 AZ-WM
	15	SEKR 15 04 AZ-WM								☆															4.76	.188	1.6	.063	1.5	.059	SEKR 15 04 AZ-WM
	12	SEKN 12 03 AZ	☆		☆	☆	☆	☆	☆		☆	☆	☆		☆	☆	☆		☆		☆			☆	3.18	.125	1.7	.067	1.2	.047	SEK 42A
		SEKN 12 04 AZ		☆	☆	☆	☆	☆		☆	☆	☆	☆	☆	☆	☆	☆		☆			☆		☆	4.76	.188	1.7	.067	1.2	.047	SEK 43A
		SEMN 12 04 AZ						☆					☆					☆	☆	☆			☆		4.76	.188	2.0	.079			SEM 43A
	15	SEKN 15 04 AZ			☆	☆	☆				☆	☆		☆	☆	☆	☆							☆	4.76	.188	1.6	.063	1.5	.059	SEK 53A
			P25	P40	P20	P25	P40	P20	M25	M30	M15	M40	M20	K20	K30	K30	K35	K25	N15	N15	S25	S30	S20	H25							

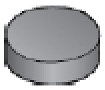
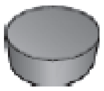
Shoulder milling 90°



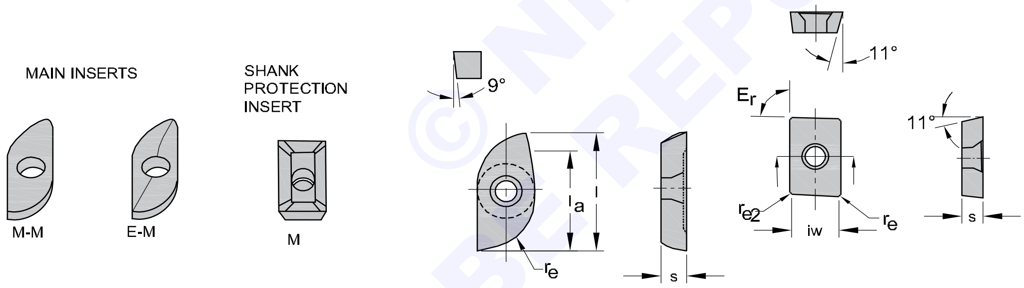
		Ordering code	P GC 2030	P GC 2040	P GC 3040	P GC 4230	P GC 4240	P CT 530	P - S6	P - SM30	M GC 2030	M GC 2040	M CT 530	M - SM30	K GC 3020	K GC 3040	K GC 4230	K GC 4240	K - H13A	K - HM	N CT 530	N - H13A	S GC 2030	S GC 2040	S - H13A	H GC 3040	*iC*	*l*	*s*	b_s	$b_{ε}$	$α_n$°	ANSI
Medium	3/8	TPKN 16 03 PP R			☆	☆	☆	☆		☆			☆	☆	☆	☆	☆	☆		☆	☆					☆	9.5	16.5	3.2	1.2	1.0	11	TPK 32P2 R
																											.375	.650	.125	.047	.039	11	
	3/4	BPKX 19 04 PD R								☆				☆														19.0	4.8	2.6	0.7	11	BPKX 19 04 PD R
																												.750	.188	.102	.028	11	
	1/2	TPKN 22 04 PD R	☆	☆	☆	☆	☆	☆	☆	☆	☆	☆	☆	☆	☆	☆	☆	☆	☆	☆	☆	☆	☆	☆	☆	☆	12.7	22.0	4.8	1.4	0.7	11	TPK 43P2 R
																											.500	.866	.188	.055	.028	11	
		BPKX 15 04 PD R																										15.9	4.8	2.0	0.7	11	BPKX 15 04 PD R
																												.625	.188	.079	.028	11	
Heavy	3/8	TPKR 16 03 PP R-WH			☆	☆	☆									☆	☆	☆								☆	9.5	16.5	3.2	1.2		11	TPKR 32P2 R-WH
																											.375	.650	.125	.047		11	
	1/2	TPKR 22 04 PD R-WH			☆	☆	☆									☆	☆	☆								☆	12.7	22.0	4.8	1.4		11	TPKR 43P2 R-WH
																											.500	.866	.188	.055		11	
			P25	P40	P20	P25	P40	P20	P40	P30	M25	M30	M20	M30	K20	K30	K30	K35	K25	K15	N15	N15	S25	S30	S20	H25							

Round insert for other cutters



	ISO	IC	iC	S CC 6060	ANSI
	RNGN120700E	12	12.7	☆	RNG45A
	RNGN 12 07 00T01020	12	12.7	☆	RNG45T0320
	RPGN060300E	06	1/4	☆	RPG22A
	RPGN090300E	09	3/8	☆	RPG32A
	RPGN120400E	12	1/2	☆	RPG43A
				S10	

Inserts for coromill - Ball nose end mill



Metric version

	For cutter dia.	r_ε	Ordering code	P GC 1025	P GC 1030	P GC 2040	P GC 4220	P GC 4240	M GC 1025	M GC 1030	M GC 2040	M GC 4240	K GC 4220	K GC 4240	K - H13A	N GC 1025	N GC 1030	N - H13A	S GC 1010	S GC 1025	S GC 1030	S GC 2040	S - H13A	S GC S30T	H GC 1010	H GC 1025	H GC 1030	H GC 4220	Dimensions l_a	l	s
Medium	10	5	R216-10 02 E-M	☆	☆	☆			☆	☆	☆					☆	☆		☆	☆	☆	☆			☆	☆	☆		8.6	9.8	1.70
	12	6	R216-12 02 E-M	☆	☆	☆			☆	☆	☆					☆	☆			☆	☆	☆		★		☆	☆		10.8	12	2.38
			R216-12 02 M-M	☆	☆			☆	☆	☆		☆		☆	☆	☆	☆	☆	☆	☆	☆		☆		☆	☆	☆		10.8	12	2.38
	16	8	R216-16 03 E-M	☆	☆	☆			☆	☆	☆					☆	☆			☆	☆	☆		★		☆	☆		14.4	16	3.18
			R216-16 03 M-M	☆	☆	☆	☆	☆	☆	☆	☆	☆	☆	☆	☆	☆	☆	☆	☆	☆	☆	☆	☆		☆	☆	☆	☆	14.4	16	3.18
	20	10	R216-20 T3 E-M	☆	☆	☆			☆	☆	☆					☆	☆			☆	☆	☆		★		☆	☆		17.9	20	3.97
			R216-20 T3 M-M	☆	☆	☆	☆	☆	☆	☆	☆	☆	☆	☆	☆	☆	☆	☆	☆	☆	☆	☆	☆		☆	☆	☆	☆	17.9	20	3.97
	25	12.5	R216-25 04 E-M	☆	☆	☆			☆	☆	☆					☆	☆			☆	☆	☆		★		☆	☆		22.3	24.9	4.76
			R216-25 04 M-M	☆	☆	☆	☆	☆	☆	☆	☆	☆	☆	☆	☆	☆	☆	☆	☆	☆	☆	☆	☆		☆	☆	☆	☆	22.3	24.9	4.76
	30	15	R216-30 06 E-M	☆	☆	☆			☆	☆	☆					☆	☆			☆	☆	☆		★		☆	☆		26.9	29.9	6.35
			R216-30 06 M-M	☆	☆	☆	☆	☆	☆	☆	☆	☆	☆	☆	☆	☆	☆	☆	☆	☆	☆	☆	☆		☆	☆	☆	☆	26.9	29.9	6.35
	32	16	R216-32 06 E-M	☆	☆	☆			☆	☆	☆					☆	☆			☆	☆	☆		★		☆	☆		28.6	31.8	6.35
			R216-32 06 M-M	☆	☆	☆	☆	☆	☆	☆	☆	☆	☆	☆	☆	☆	☆	☆	☆	☆	☆	☆	☆		☆	☆	☆	☆	28.6	31.8	6.35
	40	20	R216-40 07 E-M	☆	☆	☆			☆	☆	☆					☆	☆			☆	☆	☆		★		☆	☆		36.5	39.9	7.94
			R216-40 07 M-M	☆	☆	☆		☆	☆	☆	☆	☆		☆	☆	☆	☆	☆		☆	☆	☆	☆			☆	☆		36.5	39.9	7.94
	50	25	R216-50 07 E-M	☆	☆	☆			☆	☆	☆					☆	☆			☆	☆	☆		★		☆	☆		44.6	49.7	7.94
			R216-50 07 M-M	☆	☆	☆		☆	☆	☆	☆	☆		☆	☆	☆	☆	☆		☆	☆	☆	☆			☆	☆		44.6	49.7	7.94
				P10	P30	P30	P15	P40	M15	M15	M30	M40	K25	K35	K25	N15	N15	N15	S10	S15	S15	S30	S20	S25	H10	H15	H10	H25			

Select cutting tool, insert and holder for each operation

Objectives : At the end of this exercise you shall be able to
- **read and interpret the cutting tool chart**
- **select cutting tool for various milling operations**
- **list the code for coromill cutters.**

Job Sequence

- Selection of milling tool for a particular operation in the given chart/table
- Face milling ∅80 mm for light cut (Fig 1)
- Shoulder milling ∅50 mm for heavy cut (Fig 2)
- Ball nose cutter ∅16 mm (Fig 3)
- Slot milling ∅32 mm (Fig 4)

Steps

1 Define your type of peration

Identify your type of operation

- Face milling
- Shoulder milling
- Profile milling
- Slot milling
- Select your tool
- See page D6

2 Define your material

- Define your material according to ISO
- Steel (P)
- Stainless steel (M)
- Cast iron (K)
- Aluminium (N)
- Heat resistant and titanium alloys (S)
- Hardened material (H)

3 Select your milling cutter

- Choose cutter pitch and mounting
- Use a close pitch cutter as first choice.
- Use a coarse pitch cutter for long overhang and unstable conditions.
- Use an extra close pitch cutter for short chipping materials and super alloys.
- Choose a mounting type.

4 Select your insert

- Choose the insert geometry for your operation:
- Geometry L = Light
- For light cuts when low forces/ power are required
- Geometry M = Medium
- First choice for mixed production
- Geometry H = Heavy
- For rough operations, forging, cast skin and vibrations
- Select insert grade for optimum productivity
- Define your start values
- Cutting speeds and feeds for different materials are given on the insert dispensers and in the tables on page D 312.
- Optimize the values according to the machine and conditions.

Reference sandvic rotating tools manual

Facemilling

General facemilling (Fig 1)

CoroMill 345 Page D81

CoroMill 245 Page D88

CoroMill 490 Page D16

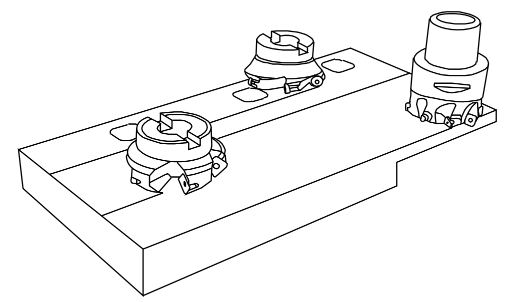
CoroMill 390 Page D24

CoroMill 200 Page D120

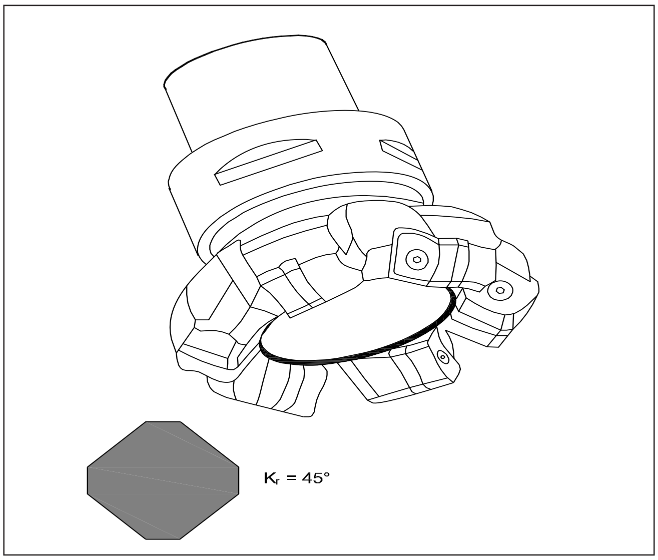
CoroMill 300 Page D108

Facemilling page D6 (Fig 1)

Fig 1



Page D81



Face milling

Example

Diameter = 50 - Coromill 345

Size of insert 13, For light machining

Coroment cap to 345° - 050C5 - 13L Close pitch

Metric version

D_c mm	Ordering code Coarse pitch			Close pitch			Extra close pitch		
	Coromant Capto								
40	345-040C4-13L	-	3	345-040C4-13M	-	4	-	-	-
50	345-050C5-13L	-	3	345-050C5-13M	-	4	345-050C5-13H	5	-
	345-050C6-13L	-	3	345-050C6-13M	-	4	345-050C6-13H	5	-
63	345-063C5-13L	-	4	345-063C5-13M	-	5	345-063C5-13H	6	-
	345-063C6-13L	-	4	345-063C6-13M	-	5	345-063C6-13H	6	-
80	345-080C6-13L	-	4	345-080C6-13M	-	6	345-080C6-13H	8	-
	-	-	-	345-080C8-13M	-	6	345-080C8-13H	8	-
100	-	-	-	345-100C8-13M	-	7	345-100C8-13H	10	-
	Cylindrical shank								
40	345-040A32-13L	-	3	345-040A32-13M	-	4	-	-	-
50	345-050A32-13L	-	3	345-050A32-13M	-	4	-	-	-
	Arbor								
40	345-040Q22-13L	-	3	345-040Q22-13M	-	4	-	-	-
50	345-050Q22-13L	-	3	345-050Q22-13M	-	4	345-050Q22-13H	5	-
63	345-063Q22-13L	-	4	345-063Q22-13M	-	5	345-063Q22-13H	6	-
80	345-080Q27-13L	-	4	345-080Q27-13M	-	6	345-080Q27-13H	8	-
100	345-100Q32-13L	-	5	345-100Q32-13M	-	7	345-100Q32-13H	10	-
125	345-125Q40-13L	-	6	345-125Q40-13M	-	8	345-125Q40-13H	12	-
160	345-160Q40-13L	-	7	345-160Q40-13M	-	10	345-160Q40-13H	-	12
200	345-200Q60-13L	-	8	345-200Q60-13M	-	12	345-200Q60-13H	-	16
250	345-250Q60-13L	-	10	345-250Q60-13M	-	14	345-250Q60-13H	-	18
	CIS Arbor								
80	A345-080J25-13L	-	4	A345-080J25-13M	-	6	A345-080J25-13H	8	-
100	A345-100J31-13L	-	5	A345-100J31-13M	-	7	A345-100J31-13H	10	-
125	A345-125J38-13L	-	6	A345-125J38-13M	-	8	A345-125J38-13H	12	-
160	A345-160J51-13L	-	7	A345-160J51-13M	-	10	A345-160J51-13H	-	12
200	A345-200J47-13L	-	8	A345-200J47-13M	-	12	A345-200J47-13H	-	16
250	A345-250J47-13L	-	10	A345-250J47-13M	-	14	A345-250J47-13H	-	18

[1] 0 = no coolant, 1 = coolant through center

Note!
Mounting dimensions, see Metalcutting Technical guide.

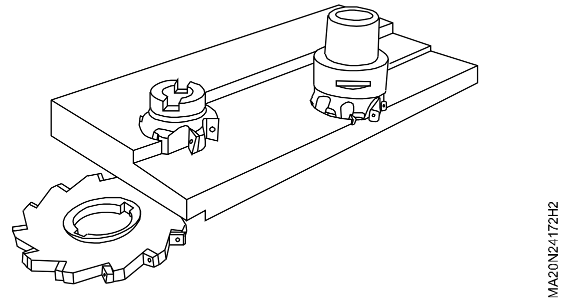
= Even pitch

= Differential pitch

Face and shoulder milling (Fig 2)

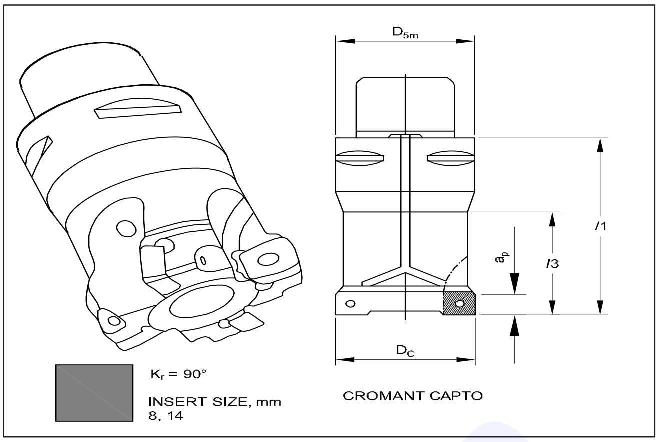
CoroMill 490 Page D16, CoroMill 390 Page D24, CoroMill 331 Page D138, CoroMil Centuryl Page D50

Fig 2



Endmill and square shoulder face mill (Page D16)

Diameter 20-84mm



Metric version

	D_c mm	Ordering code Coarse pitch			Close pitch			Extra close pitch			Coolant[1]	Dimensions	D_{5m}	l_1	l_3	Max a_p	n_{max}
		Coromant Capto															
08	20	490-020C3-08L	2	-	-	-	-	-	-	-	1	0.2	32	80	40	5.5	48500
		490-020C4-08L	2	-	-	-	-	-	-	-	1	0.4	40	70	40	5.5	39000
		490-020C5-08L	2	-	-	-	-	-	-	-	1	0.6	50	75	40	5.5	28000
		490-020C6-08L	2	-	-	-	-	-	-	-	1	1.0	63	80	40	5.5	20000
	25	-	-	-	490-025C3-08M	3	-	-	-	-	1	0.3	32	80	60	5.5	40400
		-	-	-	490-025C4-08M	3	-	-	-	-	1	0.4	40	70	45	5.5	39000
		-	-	-	490-025C5-08M	3	-	-	-	-	1	0.6	50	75	50	5.5	28000
		-	-	-	490-025C6-08M	3	-	-	-	-	1	1.0	63	80	53	5.5	20000
	32	-	-	-	490-032C3-08M	4	-	-	-	-	1	0.4	32	80	60	5.5	33900
		-	-	-	490-032C4-08M	4	-	-	-	-	1	0.5	40	70	45	5.5	33900
		-	-	-	490-032C5-08M	4	-	-	-	-	1	0.7	50	75	50	5.5	28000
		-	-	-	490-032C6-08M	4	-	-	-	-	1	1.0	63	80	53	5.5	20000
		-	-	-	490-032C8-08M	4	-	-	-	-	1	2.0	80	80	45	5.5	14000
	36	-	-	-	490-036C3-08M	-	4	-	-	-	1	0.3	32	50	30	5.5	31300
	40	-	-	-	490-040C4-08M	-	4	490-040C4-08H	6	-	1	0.6	40	70	45	5.5	29300
		-	-	-	490-040C5-08M	-	4	490-040C5-08H	6	-	1	0.8	50	75	50	5.5	28000
		-	-	-	-	-	-	490-040C6-08H	6	-	1	1.2	63	80	53	5.5	20000
		-	-	-	-	-	-	490-040C8-08H	6	-	1	2.2	80	80	45	5.5	14000
	44	-	-	-	490-044C4-08M	-	5	490-044C4-08H	6	-	1	0.6	40	60	40	5.5	27600
	50	-	-	-	490-050C5-08M	-	5	490-050C5-08H	7	-	1	1.0	50	75	50	5.5	25500
		-	-	-	490-050C6-08M	-	5	490-050C6-08H	7	-	1	1.4	63	80	53	5.5	20000
		-	-	-	-	-	-	490-050C8-08H	7	-	1	2.4	80	80	45	5.5	14000
	54	-	-	-	490-054C5-08M	-	5	490-054C5-08H	7	-	1	0.9	50	60	40	5.5	24300
	63	-	-	-	490-063C6-08M	-	6	490-063C6-08H	8	-	1	1.2	63	50	23	5.5	20000
		-	-	-	-	-	-	490-063C8-08H	8	-	0	2.8	80	80	45	5.5	14000
	66	-	-	-	490-066C6-08M	-	6	490-066C6-08H	-	8	1	1.3	63	50	28	5.5	20000
	80	-	-	-	490-080C8-08M	-	8	490-080C8-08H	-	10	0	3.4	80	80	45	5.5	14000
	84	-	-	-	490-084C8-08M	-	8	490-084C8-08H	-	10	0	2.7	80	60	30	5.5	14000
14	40	-	-	-	490-040C4-14M [2]	-	3	490-040C4-14H [2]	4	-	1	0.5	40	70	45	10.0	26400
		-	-	-	490-040C5-14M [2]	-	3	490-040C5-14H [2]	4	-	1	0.8	50	75	50	10.0	26400
		-	-	-	490-040C6-14M [2]	-	3	490-040C6-14H [2]	4	-	1	1.2	63	80	53	10.0	20000
		-	-	-	-	-	-	490-040C8-14H [2]	4	-	1	2.1	80	80	45	10.0	14000
	44	-	-	-	490-044C4-14M [2]	-	3	490-044C4-14H [2]	4	-	1	0.6	40	70	70	10.0	24600
	50	-	-	-	-	-	-	490-050C5-14H [2]	5	-	1	1.0	50	75	50	10.0	22400
		-	-	-	490-050C5-14M	-	4	-	-	-	1	1.0	50	75	50	10.0	13700
		-	-	-	-	-	-	490-050C6-14H [2]	5	-	1	1.4	63	80	53	10.0	20000
		-	-	-	490-050C6-14M	-	4	-	-	-	1	1.4	63	80	53	10.0	13700
		-	-	-	-	-	-	490-050C8-14H [2]	5	-	1	2.3	80	80	45	10.0	14000
	54	-	-	-	-	-	-	490-054C5-14H	5	-	1	0.9	50	60	60	10.0	21300
		-	-	-	490-054C5-14M	-	4	-	-	-	1	0.9	50	60	60	10.0	13000
	63	-	-	-	490-063C6-14M	-	5	490-063C6-14H	6	-	1	1.8	63	80	53	10.0	11700
		-	-	-	-	-	-	490-063C8-14H	6	-	1	2.6	80	80	45	10.0	11700
	66	-	-	-	490-066C6-14M	-	5	490-066C6-14H	6	-	1	1.5	63	65	65	10.0	11400
	80	-	-	-	490-080C6-14M	-	6	490-080C6-14H	8	-	1	1.9	63	65	65	10.0	10100
		-	-	-	490-080C8-14M	-	6	490-080C8-14H	8	-	1	3.2	80	80	45	10.0	10100
	84	-	-	-	490-084C8-14M	-	6	490-084C8-14H	8	-	1	3.0	80	70	70	10.0	9800

1) 0 = no coolant, 1 = coolant through center

2) No shims used

= Even pitch

Profile milling (Fig 3)

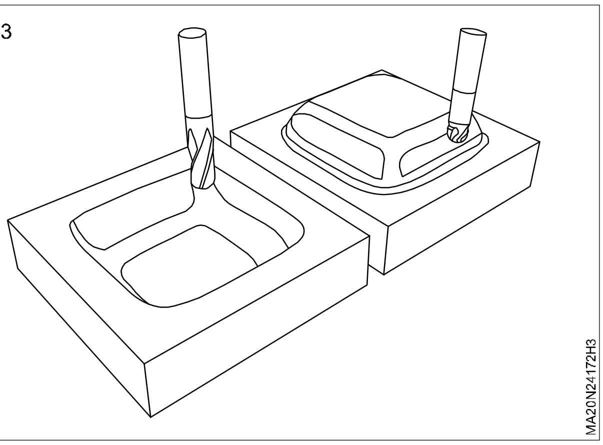
Finishing

CoroMill Ball Nose Page D133

CoroMill 316 Page D202

CoroMill Plura Page D214

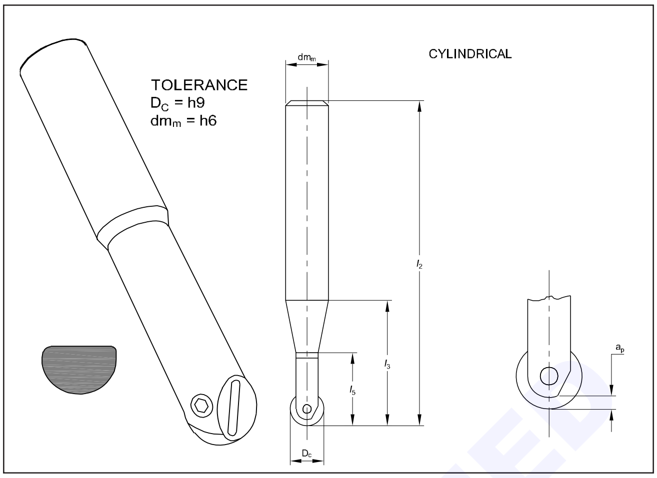
Fig 3



CoroMill Ball Nose Finishing endmill (Page D133)

Steel and Carbide shank

Diameter 8 - 32 mm



Metric version

Insert size	D_c mm	Ordering code	Dimensions (kg)	l_2	l_3	l_5	dm_m	Max a_p	n_{max}
		Cylindrical shank							
8	8	R216F-08A12C-035	0.2	92	35	19	12	1.2	40000
		R216F-08A12C-053	0.2	110	53	19	12	1.2	40000
		R216F-08A12C-075	0.2	132	75	19	12	1.2	23400
		R216F-08A12S-035	0.2	92	35	19	12	1.2	40000
		R216F-08A12S-053	0.2	110	53	33.5	12	1.2	33600
		R216F-08A12S-075	0.2	132	75	19	12	1.2	16800
10	10	R216F-10A12C-053	0.2	110	53	22	12	1.5	40000
		R216F-10A12C-075	0.3	132	75	22	12	1.5	23400
		R216F-10A12S-038	0.2	95	38	22.4	12	1.5	40000
		R216F-10A12S-053	0.2	110	53	38.7	12	1.5	40000
		R216F-10A12S-075	0.2	132	75	21.8	12	1.5	20300
12	12	R216F-12A12C-053	0.3	110	53		12	1.8	40000
		R216F-12A12S-026	0.2	83	26		12	1.8	40000
		R216F-12A12S-053	0.2	110	53		12	1.8	40000
		R216F-12A16C-085	0.4	145	85	21.5	16	1.8	21000
		R216F-12A16S-085	0.3	145	85	22.5	16	1.8	19800
16	16	R216F-16A16C-063	0.4	123	63		16	2.4	43000
		R216F-16A16S-032	0.3	92	32		16	2.4	36000
		R216F-16A16S-063	0.3	123	63		16	2.4	36000
		R216F-16A20C-100	0.7	166	100	29.5	20	2.4	25500
		R216F-16A20S-100	0.4	166	100	29.5	20	2.4	20000
20	20	R216F-20A20S-038	0.4	104	38		20	3.0	40000
		R216F-20A20S-075	0.4	141	75		20	3.0	40000
		R216F-20A25C-115	1.1	191	115	35	25	3.0	18500
		R216F-20A25S-115	0.7	191	115	35	25	3.0	18400
25	25	R216F-25A25S-045	0.5	121	45		25	3.8	40000
		R216F-25A25S-090	0.7	166	90		25	3.8	37100
		R216F-25A32S-135	1.1	215	135	42.5	32	3.8	16500
30/32	30/32	R216F-32A32S-054	0.8	134	54		32	4.8	35500
		R216F-32A32S-107	1.7	187	107		32	4.8	32500
		R216F-32A32S-160	1.6	240	160	54	32	4.8	14500

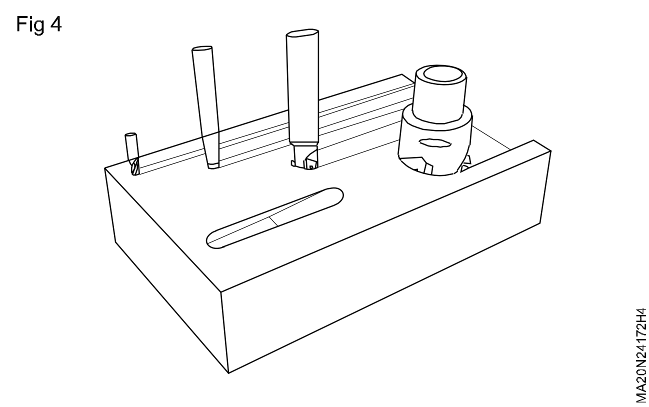
R216F-08A12S-035

S = Steel shank

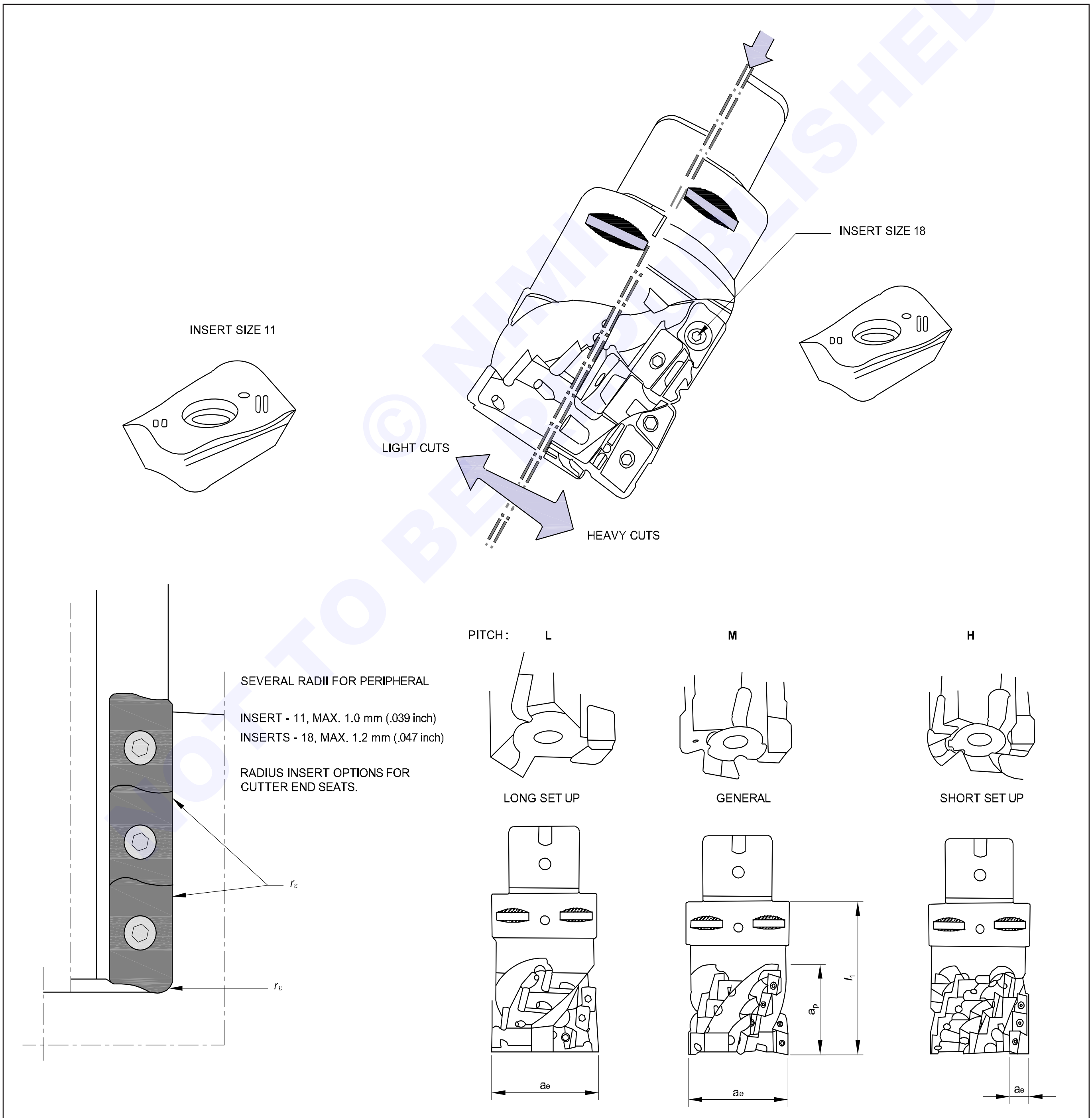
C = Carbide shank

Slot milling (Fig 4)

Fig 4

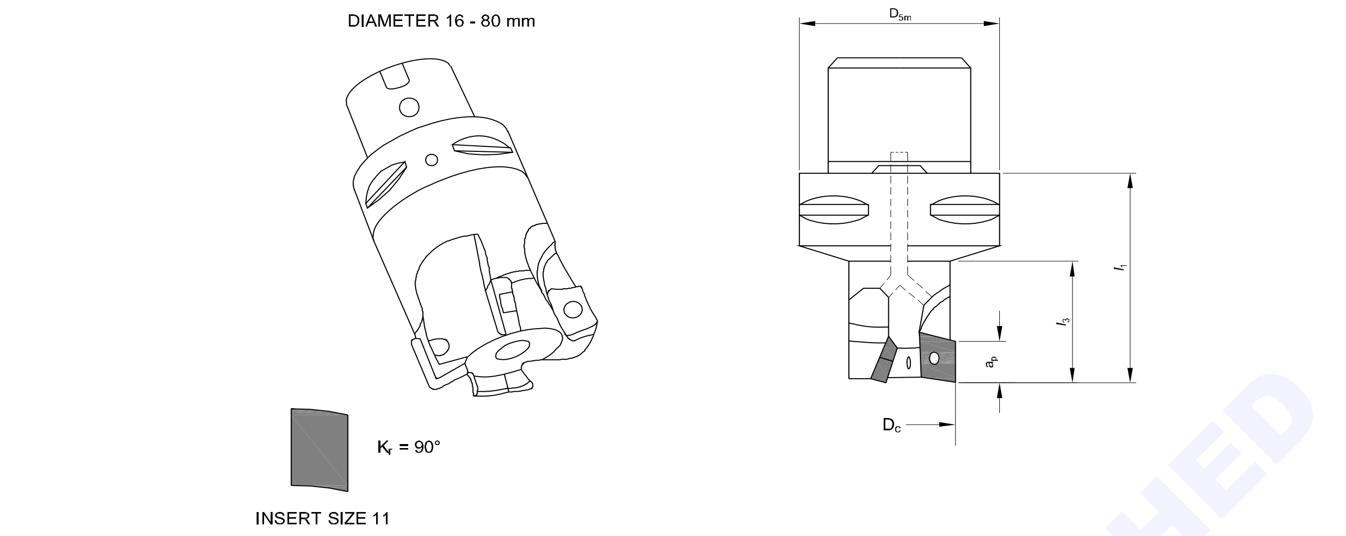


Diameter 32-200mm/1.250-8.000 inch (Page D26)



Endmill and square shoulder facemill (Page D26)

Diameter 16-80 mm



Metric version

(insert size)	D_c mm	Ordering code: Coarse pitch	Even pitch	Differential pitch	Close pitch	Even pitch	Differential pitch	Extra close pitch	Even pitch	Differential pitch	Coolant[1]	Dimensions, mm: (weight)	D_{5m}	l_1	l_3	Max a_p	n_{max}[2]
		Coromant Capto															
11	16	R390-016C3-11L050	2	-	-	-	-	-	-	-	1	0.2	C3	50	25	10.0	39000
		R390-016C4-11L	2	-	-	-	-	-	-	-	1	0.5	C4	50	25	10.0	39000
	20	R390-020C3-11L050	2	-	R390-020C3-11M050	3	-	-	-	-	1	0.2	C3	50	25	10.0	34600
		R390-020C4-11L	2	-	-	-	-	-	-	-	1	0.5	C4	50	25	10.0	34600
		-	-	-	R390-020C5-11M095	3	-	-	-	-	1	1.0	C5	95	40	10.0	34600
		-	-	-	R390-020C6-11M110	3	-	-	-	-	1	1.6	C6	110	40	10.0	34600
	25	R390-025C3-11L050	2	-	R390-025C3-11M050	3	-	-	-	-	1	0.2	C3	50	32	10.0	36500
		R390-025C4-11L	2	-	R390-025C4-11M	3	-	-	-	-	1	0.5	C4	55	32	10.0	36500
		-	-	-	R390-025C5-11M095	3	-	-	-	-	1	1.1	C5	95	45	10.0	36500
		-	-	-	R390-025C6-11M110	3	-	-	-	-	1	1.6	C6	110	45	10.0	36500
	32	R390-032C3-11L050	-	2	R390-032C3-11M050	-	3	-	-	-	1	0.3	C3	50	35	10.0	31000
		R390-032C4-11L	2	-	R390-032C4-11M	-	3	-	-	-	1	0.6	C4	65	40	10.0	31000
		R390-032C5-11L	2	-	R390-032C5-11M	-	3	-	-	-	1	0.8	C5	65	40	10.0	31000
		-	-	-	R390-032C5-11M095	-	3	-	-	-	1	1.1	C5	95	50	10.0	31000
		-	-	-	R390-032C6-11M080	3	-	-	-	-	1	1.5	C6	80	40	10.0	31000
		-	-	-	R390-032C6-11M110	-	3	-	-	-	1	1.7	C6	110	50	10.0	31000
	36	-	-	-	R390-036C3-11M050	3	-	-	-	-	1	0.4	C3	50	50	10.0	29000
		-	-	-	R390-036C3-11M075	3	-	-	-	-	1	0.5	C3	75	75	10.0	29000
	40	-	-	-	R390-040C4-11M	-	4	R390-040C4-11H	-	6	1	0.8	C4	70	50	10.0	27000
		-	-	-	R390-040C5-11M	-	4	R390-040C5-11H	-	6	1	1.1	C5	75	50	10.0	27000
		-	-	-	R390-040C6-11M080	4	-	-	-	-	1	1.6	C6	80	40	10.0	27000
	44	-	-	-	R390-044C4-11M060	4	-	-	-	-	1	0.8	C4	60	60	10.0	25600
		-	-	-	R390-044C4-11M075	4	-	-	-	-	1	0.9	C4	75	75	10.0	25600
	50	-	-	-	R390-050C5-11M060	5	-	-	-	-	1	1.0	C5	60	60	10.0	23700
		-	-	-	R390-050C6-11M080	5	-	-	-	-	1	1.8	C6	80	40	10.0	23700
	54	-	-	-	R390-054C5-11M060	5	-	-	-	-	1	1.2	C5	60	60	10.0	22700
		-	-	-	R390-054C5-11M080	5	-	-	-	-	1	1.4	C5	80	80	10.0	22700
	63	-	-	-	R390-063C5-11M060	5	-	-	-	-	1	1.4	C5	60	60	10.0	20700
		-	-	-	R390-063C6-11M080	6	-	-	-	-	1	2.2	C6	80	40	10.0	20700
	66	-	-	-	R390-066C6-11M060	6	-	-	-	-	1	1.9	C6	60	60	10.0	20200
		-	-	-	R390-066C6-11M080	6	-	-	-	-	1	2.3	C6	80	80	10.0	20200
	80	-	-	-	R390-080C6-11M060	7	-	-	-	-	1	2.2	C6	60	60	10.0	18200
		-	-	-	R390-080C6-11M080	7	-	-	-	-	1	2.7	C6	80	80	10.0	18200

1) 0 = no coolant, 1 = coolant through center

2) n_{max} (max. rev/min) for holders must also be considered.

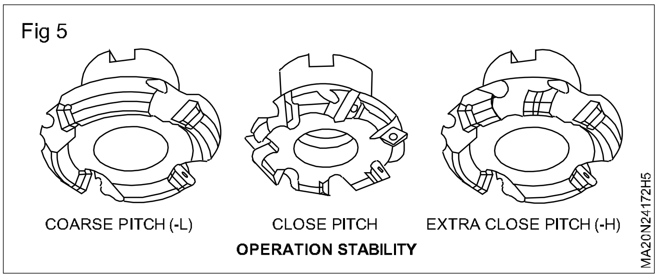
(Even pitch symbol) = Even pitch

(Differential pitch symbol) = Differential pitch

Note:

When using inserts with radius >1.6 mm, standard cutter bodies have to be modified accordingly: r = r_ε - 0.5 mm.

Operation stability (Fig 5)



Coarse pitch (-L)

Reduced number of inserts, with differential pitch, for best productivity when stability and power are limited. Extended tooling, small machines.

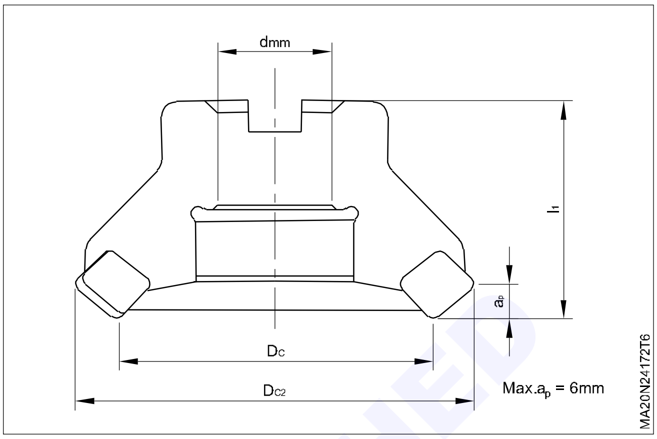
Close pitch

General purpose milling and mixed production.

Extra close pitch (-H)

Maximum number of inserts for best productivity under stable conditions, short chipping materials. Heat resistant materials.

Arbor mounting



	Ordering code	Choose L, M or H to complete the ordering code							Insert
D_c mm		x=L	x=M	x=H	D_{c2}	l1	dmm	Max rpm	
50	R245-050Q22-12x	3	4	5	62,5	40	22	16250	12
63	-063Q22-12x	4	5	6	75,5	40	22	14400	
80	-080Q27-12x	4	6	8	92,5	50	27	12700	
100	-100Q32-12x	5	7	10	112,5	50	32	11300	
125	-125Q40-12x	6	8	12	137,5	63	40	10100	
160	-160Q40-12x	7	10	16	172,5	63	40	8900	

General code key for coromill cutters

R	A	390	-	063	Q	22	L	-	11	M	050
1	2	3		4	5	6	7		8	9	10

1 Style
R= Right hand rotating

2 Performance
A= Inch

3Main code
E.g: 390 = Coromill 390

4 Cutting diameter
E.g. 063= 63mm

5Type coupling	
A = Cylindrical, mm B = Weldon mm C = Coromant Capto D = Cylindrical inch J = CIS Tenon drive M = Weldon, inch N = Whistle Notch inch Q = Arbor mounting mm O = Cylindrical inch	R = Arbor mounting inch T = Threaded coupling W = Whistle Notch mm HA= HSK form A

6 Coupling size
22 = 22 mm

7 Extra long
L = Extra long

8 Insert size
11 = 11 mm (l_a)

9 Pitch
L = Coarse pitch M = Close pitch H = Extra close pitch

10 Length, l1
E.g.: 050 = 50 mm

Fix inserts and tools in tool holders

Objectives : At the end of this exercise you shall be able to
- **fix the insert in the cutter**
- **fix the cutter in cutter holder.**

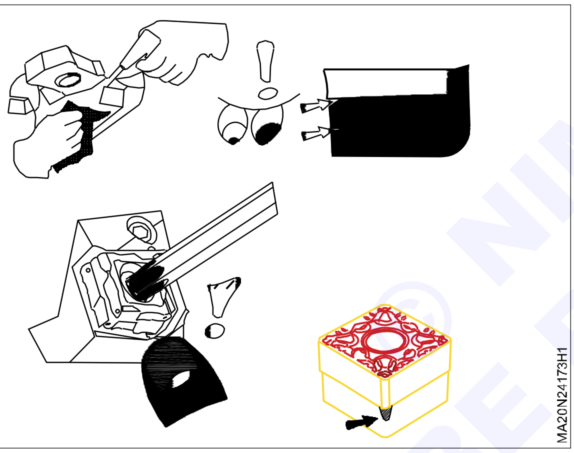
Job Sequence

Shim and insert fixing

The shim and insert seat (Fig 1)

- Check shim damage.
- Clean insert seat and damaged location and support for cutting edge.
- If necessary index or replace shim.
- Proper insert location against support points.
- IMPORTANT that shim corners have not been knocked off during machining or handling.

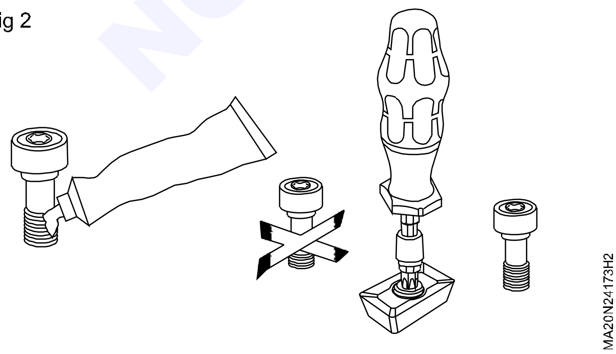
Fig 1



Clamping screws (Fig 2)

- Correct screw-tightening torque.
- Careful screw lubrication to prevent seizure.
- Lubricant should be applied to the screw thread as well as the screw head face.
- Replace worn or exhausted screws
- Use correct keys

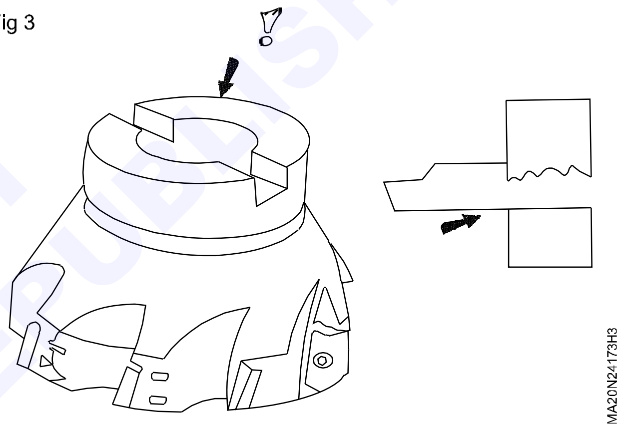
Fig 2



Contact faces (Fig 3)

- Always check supporting and contact faces of tool holders, milling cutters and drills, making sure there is no damage or dirt.
- In boring operations it is especially important to have the best possible clamping. If the bar is not supported all the way out to where the holder ends, this will add to the overhang of the tool and help to induce vibration

Fig 3



Production security (Fig 4)

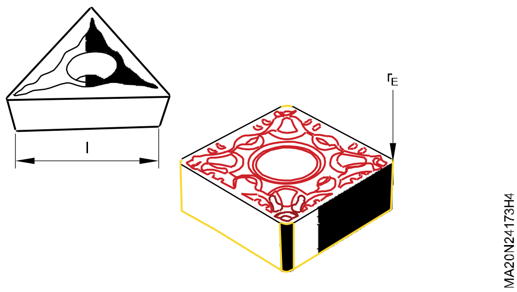
- Important to choose the right insert size, insert shape and geometry and insert nose radius to achieve good chip flow.
  - Select largest possible point angle on the insert for strength and economy
  - Select largest possible nose radius for insert strength.
  - Select a smaller nose radius if there is a tendency to vibrate.

l = cutting edge length (insert size)

la = effective cutting edge length

r_E = nose radius

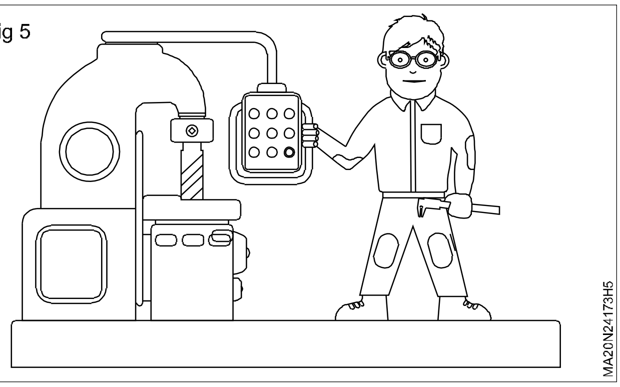
Fig 4



Stability (Fig 5)

- Stability is the key factor for successful metal cutting, affecting machining costs and productivity.
- Make sure that any unnecessary play, overhand, weakness, etc. has been eliminated and that correct types and sizes of tools are employed for the job

Fig 5



Maintenance tips

- Check tool wear and shims for damage
- Make sure insert seat is clean.
- Make sure of correct insert location.
- Make sure correct keys and drivers are used.
- Insert screws should be correctly tightened.
- Lubricate screws before tool assembly.
- Make sure contact faces are clean and undamaged on tools, holding tools and machine spindles.
- Make sure boring bars are clamped well and that holder is undamaged at the end.
- A well organized, maintained and documented tool inventory is a production cost saver.
- Stability is always a critical factor in any metal cutting operation.

Decide cutting tools material for various application

Objectives : At the end of this exercise you shall be able to
- **interpret the meaning of PMKSH,**
- **state the corresponding work material for PMKSH**
- **select the appropriate tool material to machine work material like, carbon steel, cast iron and Hard steel etc.**

Job Sequence

- Read the content of guide lines for selection of cutting tool materials for cemented carbide.
- Make a note of P,M,K,S,H grades make a note of relative work material against PMKSH
- Select the cutting tool materials in terms of P,M,K,S,H grade to machine carbon steel, Stainless steel, cast iron, and hardened steel and list in the tabular column.

Guide lines for selectin of cutting tool materials cemented carbides

Work Material	Tool Life		Work Material	Hardness
	Life	Performance		
P	90min	15min	Carbon steel, alloy steel	180HB
M	90min	15min	Stainles steel	180HB
K	90min	15min	Cast iron	180HB
S	25min	5 min	Titanium alloy	320HB
			Ni, Co-Based alloy	400HB
H	80min	10min	Hardened steel	HRC60

Classification and designation rules of cemented carbide for cutting tools

- Cemented carbide grades for cutting tools are divided into six categories: P,M,K,S and H according to the different fields of use, as listed in Table 1.
- Each category is divided into several groups in order to meet different usage requirements and according to the different wear resistance and toughness of the cemented carbide materials for cutting tools, which are represented by double-digit such as 01,10 and 20 etc.
- When necessary, a supplementary group number can be inserted between the two group numbers, represented by 05,15 and 25 etc.

Table 1 Cemented carbides types for cutting tools

Types	Fields of use
P	Processing of long-cut materials such as steel, cast steel, Long cut malleable cast iron, etc.
M	General alloy for processing stainless steel, cast steel, manganese steel, malleable cast iron, alloy steel, alloy cast Iron etc.,
K	Machining of short-cut materials such as cast iron, chilled cast iron, short-cut malleable iron, grey cast iron etc.
N	Processing of non -ferrous metals and non-metallic materials, such as aluminium, magnesium, plastics, wood etc.
S	Processing of heat resistant and high quality alloys such as heat resistant steel, alloys containing nickel, cobalt, titanium etc.
H	Machining of hard cutting materials, such as hardened steel, chilled cast iron etc.

Groups			Mechanical properties		
Types	**Group number**	**Main components**	**Rockwell hardness HRA,**	**Vickers hardness HV,**	**Flexural strength /MPa Ru,**
P	01	Alloys/coating alloys based on TNC and WC with Co (N+M0, Ni+Co) as binder	92.3	1750	700
	10		91.7	1680	1200
	20		91	1600	1400
	30		90.2	1500	1550
	40		89.5	1400	1750
M	01	Take WC as the base, Co as the binder, and add a small amount of TiC (TaC, NbC) alloy/ coating alloy	92.3	1730	1200
	10		91	1600	1350
	20		90.2	1500	1500
	30		89.9	1450	1650
	40		88.9	1300	1800
K	01	Take WC as the base, Co as the binder, or add a small amount of TaC, NbC alloy/coating alloy.	92.3	1750	1350
	10		91.7	1680	1460
	20		91	1600	1550
	30		89.5	1400	1650
	40		88.5	1250	1800
N	01	Take WC as the base, Co as the binder, or add a small amount of TaC, NbC or CrC alloy/coating alloy.	92.3	1750	1450
	10		91.7	1680	1580
	20		91	1600	1650
	30		90	1450	1700
S	01	Take WC as the base, Co as the binder, or add a small amount of TaC, NbC or TiC alloy/coating alloy.	92.3	1730	1500
	10		91.5	1650	1580
	20		91	1600	1650
	30		90.5	1550	1750
H	01	Take WC as the base, Co as the binder, or add a small amount of TaC, NbC or TiC alloy/coating alloy.	92.3	1730	1000
	10		91.7	1680	1300
	20		91	1600	1650
	30		90.5	1520	1500

Note:

1 **Choose one of Rockwell hardness and vickers hardness;**

2 **The above data are requrements for non-coated cemented carbide, and the coated products can be reduced by 30-50 according to the corresponding vickers hardness.**

Select cutting parameter from tool manufacturer catalog

Objectives : At the end of this exercise you shall be able to
- **read and interpret the meaning of cutting parameter, and its equation to calculate cutting parameter**
- **select the cutter holder for a particular operation from the catalog**
- **select the insert for the selected the holder to machine required material**
- **select the cutting speed and depth of cut from the catalog.**

Job Sequence

The cutting parameters for milling

The milling parameter evaluated are spindle speed, feed rate, and depth of cut.

Technological parameters

The technological parameters of milling include:

- Rotational speed n [RPM];
- Diameter of the tool Dc [mm]:
- Cutting speed Vc [m/min] - Equation - 1;
- Feed speed Vf [mm/min] - equation 2;
- Feed per revolution ff [mm/rev];
- Feed per blade/tooth fz [mm/tooth - equation 3;
- Width a_e and depth of cut a_p [mm]

Table 1 presents the basic computational relationships

Number	Technological parameter	Equation	Component description
1	cutting speed V_c	$V_c = \frac{\pi D_c . n}{1000}$	D_c - diameter of the tool n - rotational speed
2	feed speed V_f	$Vf = F_f.n$	f_f - feed per revolution n - rotational speed
3	feed per tooth f_z	$f_z = \frac{f_f}{z} = \frac{V_f}{z.n}$	f_f - feed per revolution n - rotational speed V_f - feed speed

Work piece material

In order to facilitate the selection of parameters, the ISO material groups are used. The same division applies total machining methods. There are 6 materials groups. (Table 2)

Table 2

Work piece material	ISO marking	Sample material:
Steel	P	Low-alloy steel
Stainless steel	M	Austenitic stainless steel
Cast iron	K	Grey cast iron, ductile cast iron
Aluminium alloys	N	Cast
Heat-resistant alloys	S	high -alloys based on iron, nickel, cobalt and titanium
Hardened steel	H	Hasrdened and tempered steel

Catalogs whether in paper or electronic versions (eg. Sandvik Coromant or ISCAR or SECO) constitute a significant user support in making the first choice and mathing the tools and parameters to the individual technological tast.

Power and torque

When milling, the main drive power requirements depend on factors such as:

The amount of material to be removed;

Average chip thickness;

Tool (insert) geometry (cutter);

Cutting speeds

Task

From the given details select to cutting parameter to face mill on mild steel with 100 mm cutter diameter CoroMill 245-approach angle 45 degree

Cutter holder

Insert

Feed rate

Cutting speed

Refer Page	D2 for guidance for milling tool selection
	D4 for operation
	D88 - D90 for cutter holder selection for face milling
	D93 for insert selection
	D14 for insert code key
	D308 for feed per tooth
	D312-313 for cutting speed

How to choose your milling tool

1 Define your type of operation

Identify your type of operation

- Facemilling
- Shoulder milling
- Profile milling
- Slot milling
- Select your tool
- See page D6

2 Define your material

Define your material according to ISO:

Steel (P)

Stainless steel (M)

Cast iron (K)

Aluminium (N)

Heat resistant and titanium alloys (S)

Hardened material (H)

See material cross reference list in Chapter 1

3 Select your milling cutter

Choose cutter pitch and mounting

Use a close pitch cutter as first choice.

Use a coarse pitch cutter for long overhang and unstable conditions.

Use an extra close pitch cutter for short chipping materials and super alloys.

Choose a mounting type.

4 Select your insert

Choose the insert geometry for your operation:

Geometry L = Light

For light cuts when low forces/ power are required

Geometry M = Medium

First choice for mixed production

Geometry H = Heavy

For rough operations, forging, cast skin and vibrations

Select insert grade for optimum productivity

5 Define your start values

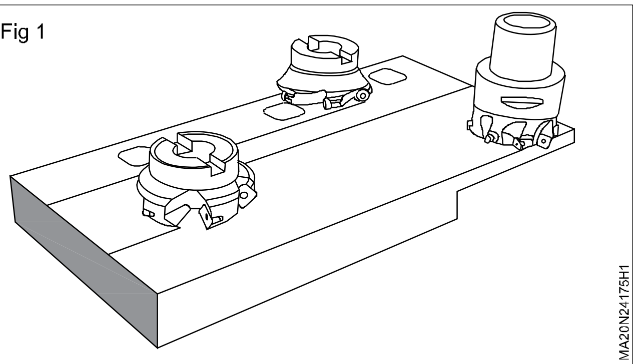
Cutting speeds and feeds for different materials are given on the Insert dispensers and in the tables on page D 312.

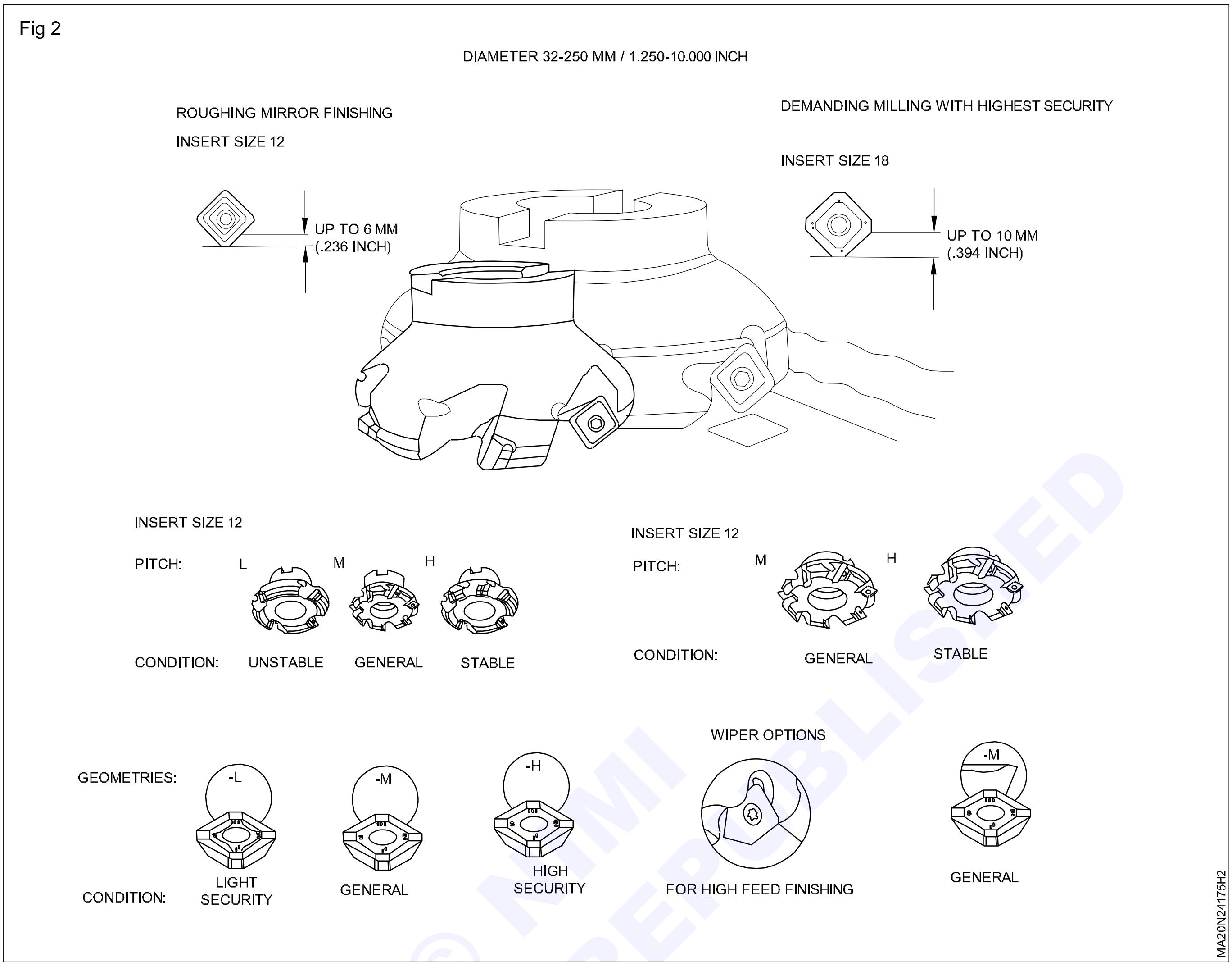
Optimize the values according to the machine and conditions

General facemilling (Fig 1&2)

CoroMill 345	Pate D81
CoroMill 245	Pate D88
CoroMill 490	Pate D16
CoroMill 390	Pate D24
CoroMill 200	Pate D120
CoroMill 300	Pate D108

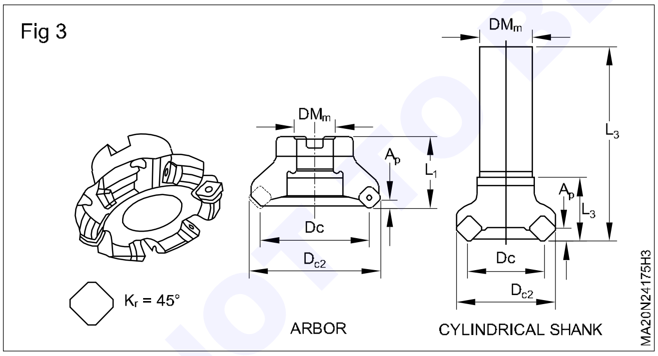
Fig 1





Face mill (Fig 3)

Diameter 32 – 250 mm



Metric version

□ [1]	D_c mm	Ordering code: Coarse pitch	Even pitch	Close pitch	Even pitch	Differential pitch	Extra close pitch	Even pitch	Differential pitch	Dimensions: Weight (kg)	dm_m	D_{c2}	l_1	l_2	l_3	Max a_p	n_{max} [2]
		Cylindrical shank															
12	32	-	-	R245-032A32-12M[3]	3	-	-	-	-	1.0	32.0	44.5		120	39	6	18250
	40	R245-040A32-12L	3	-	-	-	-	-	-	1.1	32.0	52.5		120	39	6	18250
	50	R245-050A32-12L	3	R245-050A32-12M	4	-	-	-	-	1.4	32.0	62.5		120	39	6	16250
	63	R245-063A32-12L	4	R245-063A32-12M	5	-	-	-	-	1.1	32.0	75.5		120	39	6	14400
	80	R245-080A32-12L	4	R245-080A32-12M	6	-	-	-	-	2.1	32.0	92.5		120	39	6	12700
		Arbor															
12	50	R245-050Q22-12L	3	R245-050Q22-12M	4	-	R245-050Q22-12H	5	-	0.5	22.0	62.5	40			6	16250
	63	R245-063Q22-12L	4	R245-063Q22-12M	5	-	R245-063Q22-12H	6	-	0.6	22.0	75.5	40			6	14400
	80	R245-080Q27-12L	4	R245-080Q27-12M	6	-	R245-080Q27-12H	8	-	1.0	27.0	92.5	50			6	12700
	100	R245-100Q32-12L	5	R245-100Q32-12M	7	-	R245-100Q32-12H	10	-	1.4	32.0	112.5	50			6	11300
	125	R245-125Q40-12L	6	R245-125Q40-12M	8	-	R245-125Q40-12H	12	-	2.7	40.0	137.5	63			6	10100
	160	R245-160Q40-12L	7	R245-160Q40-12M	10	-	R245-160Q40-12H	16	-	5.0	40.0	172.5	63			6	8900
	200	R245-200Q60-12L	8	R245-200Q60-12M	12	-	R245-200Q60-12H	20	-	6.7	60.0	212.5	63			6	7950
	250	R245-250Q60-12L	10	R245-250Q60-12M	14	-	R245-250Q60-12H	24	-	8.5	60.0	262.5	63			6	7100
18	80	-	-	R245-080Q32-18M[3]	-	4	R245-080Q32-18H	-	5	1.6	32.0	98.8	50			10	6100
	100	-	-	R245-100Q32-18M[3]	-	4	R245-100Q32-18H	-	6	1.9	32.0	118.8	50			10	5400
	125	-	-	R245-125Q40-18M	-	5	R245-125Q40-18H	-	7	3.6	40.0	138.8	63			10	4900
	160	-	-	R245-160Q40-18M	-	6	R245-160Q40-18H	-	9	8.7	40.0	178.8	63			10	4300
	200	-	-	R245-200Q60-18M	-	8	R245-200Q60-18H	-	12	12.0	60.0	218.8	63			10	3800
	250	-	-	R245-250Q60-18M	-	10	R245-250Q60-18H	-	14	8.9	60.0	268.8	63			10	3400
		CIS Arbor															
12	80	RA245-080J25.4-12L	4	RA245-080J25.4-12M	6	-	RA245-080J25.4-12H	8	-	1.2	25.4	92.5	50			6	12700
	100	RA245-100J31.75-12L	5	RA245-100J31.75-12M	7	-	RA245-100J31.75-12H	10	-	2.2	31.8	112.5	63			6	11300
	125	RA245-125J38.1-12L	6	RA245-125J38.1-12M	8	-	RA245-125J38.1-12H	12	-	3.4	38.1	137.5	63			6	10100
	160	RA245-160J50.8-12L	7	RA245-160J50.8-12M	10	-	RA245-160J50.8-12H	16	-	5.0	50.8	172.5	63			6	8900
	200	RA245-200J47.625-12L	8	RA245-200J47.625-12M	12	-	RA245-200J47.625-12H	20	-	6.7	47.625	212.5	63			6	7950
	250	RA245-250J47.625-12L	10	RA245-250J47.625-12M	14	-	-	-	-	8.5	47.625	262.5	63			6	7100
18	80	-	-	RA245-080J25-18M[3]	-	4	RA245-080J25-18H	-	5	1.5	25.4	101.3	50			10	6100
	100	-	-	RA245-100J31-18M[3]	-	4	RA245-100J31-18H	-	6	2.7	31.8	118.8	50			10	5400
	125	-	-	RA245-125J38-18M	-	5	RA245-125J38-18H	-	7	3.7	38.1	143.8	63			10	4900
	160	-	-	RA245-160J51-18M	-	6	RA245-160J51-18H	-	9	9.3	50.8	178.8	63			10	4300
	200	-	-	RA245-200J47-18M	-	8	RA245-200J47-18H	-	12	12.0	47.625	218.8	63			10	3800
	250	-	-	RA245-250J47-18M	-	10	RA245-250J47-18H	-	14	8.9	47.625	268.8	63			10	3400

1) Inserts are ordered separately.

2) n_{max} (max. rev/min) for holders must also be considered.

3) Without shim

Note!

Mounting dimensions, see Metalcutting Technical guide.

Bolt circle for cutters 200 and 250mm = 4"

(Even pitch symbol) = Even pitch

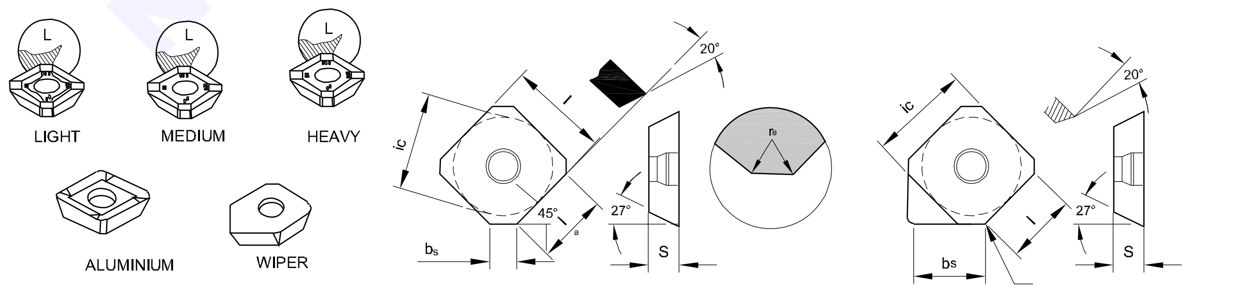
(Differential pitch symbol) = Differential pitch

Inserts for CoroMill® 245 (Fig 4)

Cemented carbide / Cermet

The Wiper inserts can also be used for turn milling.

Fig 4



Dimensions, mm (inch)

Size	iC	l_a	s	Max a_p	Max a_p, wiper
12	13.4 (.528)	10 (.394)	3.97 (.156)	6.5 (.256)	2.5 (.098)
18	18 (.709)	13.9 (.547)	6.10 (.240)	9.75 (.384)	9.75 (.384)

			P						M						K									N				S							H					Dimensions, millimeter, inch (mm, in.)			
			GC	GC	GC	GC	GC	CT	GC	GC	GC	GC	GC	CT	GC	GC	GC	GC	GC	GC	-	GC	GC	GC	-	-	GC	GC	GC	GC	GC	-	GC	GC	GC	GC	GC	GC	CT				
	⊡	Ordering code	1030	2040	4220	4230	4240	530	1030	1040	2030	2040	4240	530	1020	3040	3220	4220	4230	4240	H13A	K15W	K20W	1030	H10	H13A	K15W	1010	1030	2030	2040	H13A	S30T	S40T	1010	1030	3040	4220	530	b_s mm	b_s in.	$r_ε$ mm	$r_ε$ in.
Light	12	R245-12 T3 E-AL																							☆															2.3	.091	2.5	.098
		R245-12 T3 E-KL													★		☆				☆	☆				☆	☆					☆								2.0	.079	1.5	.059
		R245-12 T3 E-ML	☆							☆	★	☆												☆					☆	☆	☆					☆				2.1	.083	1.5	.059
		R245-12 T3 E-PL	★			☆		☆						☆										☆				☆	☆				★	★	★	☆			☆	2.1	.083	1.5	.059
		R245-12 T3 M-KL													★	☆	☆						☆														☆			1.47	.083	1.7	.059
		R245-12 T3 M-PL	★		☆	☆	☆		☆				☆					☆	☆	☆								☆	☆				★	★	★			☆		2.05	.081	1.4	.055
Medium	12	R245-12 T3 K-MM								☆	★	☆											☆							☆	☆		★	★						2.0	.079	1.5	.059
		R245-12 T3 M-KM													★	☆	☆				☆		☆			☆						☆					☆			1.47	.079	1.7	.059
		R245-12 T3 M-PM	☆		☆	★	☆	☆	☆				☆	☆				☆	☆	☆								☆	☆						★			☆	☆	2.0	.079	1.5	.059
	18	R245-18 T6 M-KM													★	☆	☆						☆														☆			1.5	.059	1.0	.039
		R245-18 T6 M-MM		☆							☆	★																		☆	☆									1.5	.059	1.0	.039
		R245-18 T6 M-PM	☆		☆	★	☆						☆					☆	☆	☆				☆				☆	☆						★	☆		☆		1.5	.059	1.0	.039
Heavy	12	R245-12 T3 M-KH													☆	☆	★						☆														☆			1.47	.079	1.7	.059
		R245-12 T3 M-PH			☆	★	☆						☆					☆	☆	☆																		☆		1.53	.060	1.6	.063
Wiper	12	R245-12 T3 E-W	★					☆						☆	★		☆				☆	☆		☆		☆	☆	☆	☆			☆			★	☆			☆	8.2	.323	1.5	.059
	18	R245-18 T6 E-W	★								★				★									☆					☆	☆						☆				10.8	.425	1.0	.039
			P30	P40	P15	P25	P40	P20	M15	M35	M25	M30	M40	M20	K20	K30	K20	K25	K30	K35	K25	K15	K25	N15	N10	N15		S10	S15	S25	S30	S20	S25	S35	H10	H10	H25	H25	H15				

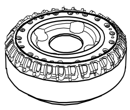
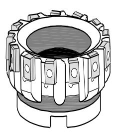
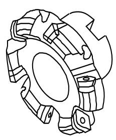
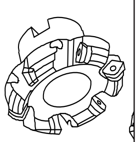
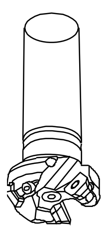
R245-12 T3 E-PL

E = Highest edge sharpness and precision

K = Highest edge sharpness

M = Highest edge security

Shoulder milling

			Feed per tooth, f_z (mm/tooth)	Max. chip thickness h_{ex}, mm	Feed per tooth, f_z (inch/tooth)	Max. chip thickmess h_{ex}, (inch)
k_r 90° (0°)	Insert geomtry	Insert size	Starting value (min.- max.)	Starting vaule (min.- max.)	Starting value (min.- max.)	Starting value (min.- max.)
AUTO-FS	SBEN		0.17 (0.1-0.3)	0.17 (0.1-0.3)	.007 (.004-.012)	.007 (.004-.012)
Finishing	SBEX					
	SBEX -11					
R/LA262.4						
R/L262.4						
R/L262.42						
T-Line						
Roughing	CDE		0.17 (0.1-0.3)	0.17 (0.1-0.3)	.007 (.004-.012)	.007 (.004-.012)
						
R260.90						
Facemilling k_r 75° 10°	Insert geomtry	Insert size	Starting value (min.-max.)	Starting value (min.- max.)	Starting value (min. - max.) value	Starting (min. - max.)
Coromill 345	E-PL					
	E-ML					
	E-KL	13	0.15 (0.07-0.20)	0.10 (0.07-0.14)	.006 (.003-.008)	.004 (.003-.006)
	M-PL					
	M-KL					
	M-PM					
	M-MM		0.30 (0.15-0.45)	0.21 (0.10-0.32)	.012 (.006-.018)	.008 (.004-.013)
	M-KM					
	M-PH		0.45 (0.35-0.55)	0.32 (0.25-0.39)	.012 (.014-.022)	.013 (.010-.015)
	M-KH		0.40 (0.30-0.50)	0.28 (0.21-0.35)	.016 (.012-.020)	.011 (.008-.014)
CoroMill® 245	E-PL					
	E-ML		0.14 (0.08-0.21)	0.10 (0.06-0.15)	.006 (.003-.008)	.004 (.002-.006)
	E-KL					
	CT530					
	H13A		0.11 (0.07-0.17)	0.08 (0.06-0.12)	.004 (.003-.007)	.003 (.008-.014)
	H10					
	M-PL		0.17 (0.07-0.21)	0.12 (0.06-0.15)	.007 (.003-.008)	.005 (.002-.006)
	M-KL					
	M-PM		0.24 (0.10-0.28)	0.17 (0.07-0.20)	.009 (.004-.011)	.007 (.003-.008)
	M-KM					
	CT530		0.12 (0.08-0.18)	0.09 (0.06-0.13)	.005 (.003-.007)	.004 (.002-.005)
	H13A					
	K-MM		0.23 (0.10-0.28)	0.16 (0.07-0.20)	.009 (.004-.011)	.006 (.003-.008)
	M-PH		0.35 (0.10-0.42)	0.25 (0.07-0.30)	.014 (.004-.017)	.010 (.003-.030)
	M-KH					
	E-AL		0.24 (0.10-0.28)	0.17 (0.07-0.28)	.009 (.004-.011)	.010 (.003-.08)

	E Ceramic					
	CC6190		0.21 (0.10-0.30)	0.15 (0.07-0.20)	.008 (.004-.012)	.006 (.003-.008)
	E CBN					
	CB50		0.14 (0.07-0.21)	0.10 (0.06-0.15)	.006 (.003-.008)	.004 (.002-.006)
	E PCD					
	CD10		0.14 (0.07-0.21)	0.10 (0.06-0.15)	.006 (.003-.008)	.004 (.002-.006)

Milling with large enlargement, metric values

ISO P			Specific cutting force K_c 0.4	Hardness brinell		CT530	GC1010
						Max chip thickness h_{ex} mm	
						0.1 - 0.15 - 0.2	0.05-0.1-0.2
MC NO.	CMC No.	Material	N/mm²	HB	mc	cutting speed V^{c1}/m/min	
		Steel unalloyed					
P1.1.Z.AN	01.1	C = 0.1–0.25%	1500	125	0.25	430–390–350	-
P1.2.Z.AN	01.2	C = 0.25–0.55%	1600	150	0.25	385–350–315	-
P1.3.Z.AN	01.3	C = 0.55–0.80%	1700	170	0.25	365–330–300	-
P1.3.Z.AN	01.4		1800	210	0.25	315–290–260	-
P1.3.Z.HT	01.5		2000	300	0.25	235–210–195	-
		Low alloyed (alloying elements . 5%)					
P2.1.Z.AN	02.1	Non-hardened	1700	175	0.25	300.275.245	-
P2.5.Z.HT	02.2	Hardened and tempered	1900	300	0.25	195.180.160	-
		High alloyed (alloying elements > 5%)					
P3.0.Z.AN	03.11	Annealed	1950	200	0.25	230.205.185	180-165-135
P3.1.Z.AN	03.13	Hardened tool steel	2150	200	0.25	190.170.155	150-135-110
P3.0.Z.HT	03.21		2900	300	0.25	165.150.135	130-120-100
P3.0.Z.HT	03.22		3100	380	0.25	105.95.85	80-75-60
		Castings					
P1.5.C.UT	06.1	Unalloyed	1400	150	0.25	305.280.250	245-220-180
P2.6.C.UT	06.2	Low alloyed (alloying elements . 5%)	1600	200	0.25	245.220.200	195-175-145
P3.0.C.UT	06.3	High alloyed (alloying elements > 5%)	1950	200	0.25	180.160.145	140-130-105
ISO M			**Specific cutting force K_c 0.4**	**Hardness brinell**		**CT530**	**GC1025**
						Max chip thickness h_{ex} mm	
						0.1 - 0.15 - 0.2	**0.05-0.1-0.2**
MC NO.	**CMC No.**	**Material**	**N/mm²**	**HB**	**mc**	**cutting speed V_{c1}/m/min**	
		Stainless steel Ferritic/martensitic					
P5.0.Z.AN	05.11	Non-hardened	1800	200	0.21	285 .255 .230	255 .225 .180
P5.0.Z.PH	05.12	PH-hardened	2850	330	0.21	205 .185 .165	180 .160 .130
P5.0.Z.HT	05.13	Hardened	2350	330	0.21	215 .190 .170	185 .165 .135

		Austenitic					
M1.0.Z.AQ	05.21	Non-hardened	1950	200	0.21	265 .240	215 250 .225 .180
M1.0.Z.PH	05.22	PH-hardened	2850	330	0.21	200 .175	160 170 .155 .125
M2.0.Z.AQ	05.23	Super austenitic	2250	200		-	-
		Austenitic-ferritic (Duplex)					
M3.1.Z.AQ	05.51	Non-weldable . 0.05%C	2000	230	0.21	260 .235 .210	205 .185 .145
M3.2.Z.AQ	05.52	Weldable < 0.05%C	2450	260	0.21	230 .205 .185	175 .155 .125
		Stainless steel . Cast Ferritic/martensitic					
P5.0.C.UT	15.11	Non-hardened	1700	200	0.25	255 .230 .205	225 .200 .160
P5.0.C.PH	15.12	PH-hardened	2450	330	0.25	180 .160 .145	155 .140 .115
P5.0.C.HT	15.13	Hardened	2150	330	0.25	195 .175 .155	170 .155 .120
M1.0.C.UT	15.21	Non hardened	1800	200	0.25	255 .225 .205	235 .210 .170
M1.0C.PH	15.22	PH-hardened	2450	330	0.25	180 .160 .145	160 .140 .115
M2.0.C.AQ	15.23	Super austenitic	2150	200		-	-
		Austenitic-ferritic (Duplex)					
M3.1.C.AQ	15.51	Non-weldable . 0.05%C	1800	230	0.25	245 .220 .195	195 .175 .140
M3.2.C.AQ	15.52	Weldable < 0.05%C	2250	260	0.25	215 .190 .170	160 .145 .115
ISO K			**Specific cutting force K_c 0.4**	**Hardness brinell**		**CB50**	**GC6190**
						Max chip thickness h_{ex} mm	
						0.1 - 0.15 - 0.2	**0.05-0.1-0.2**
MC NO.	**CMC No.**	**Material**	**N/mm²**	**HB**	**mc**	**cutting speed V_{c1}/m/min**	
		Malleable cast iron					
	07.1	Ferritic (short chipping)	790	130	0.28	-	1300 –1050 –880
K1.1.C.NS	07.2	Pearlitic (long chipping)	900	230	0.28	-	1100 –890 –730
		Gray cast iron					
K2.1.C.UT	08.1	Low tensile strength	890	180	0.28	850 –720 –620	1600–1300–1050
K2.2.C.UT	08.2	High tensile strength	1100	245	0.28	910 –780 –670	1200–990–810
		Nodular cast iron					
K3.1.C.UT	09.1	Ferritic	900	160	0.28	-	1000–830–680
K3.3.C.UT	09.2	Pearlitic	1350	250	0.28	495 –420 –360	840 –690 –570

CNC programme using G00, G01, G02, & G03

Objectives : At the end of this exercise you shall be able to
- write the co-ordinates points in absolute and incremental co-ordinate systems
- write the CNC programme using linear and circular interpolation
- enter the CNC programme in CNC simulator and verify the programme.

Fig 1

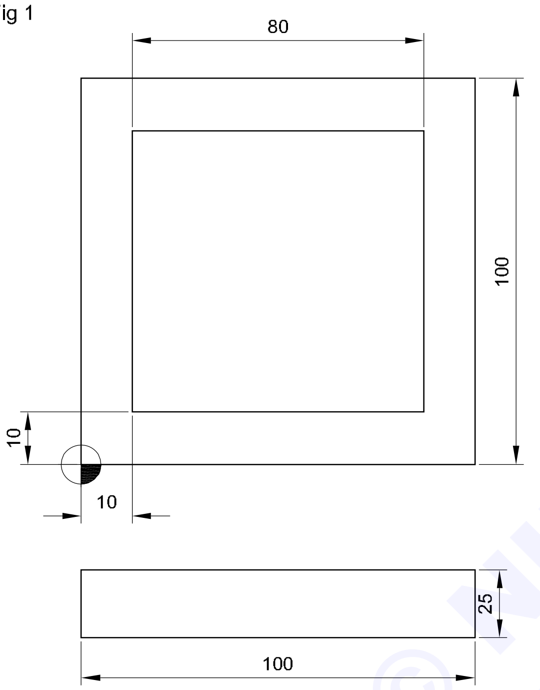


Fig 2

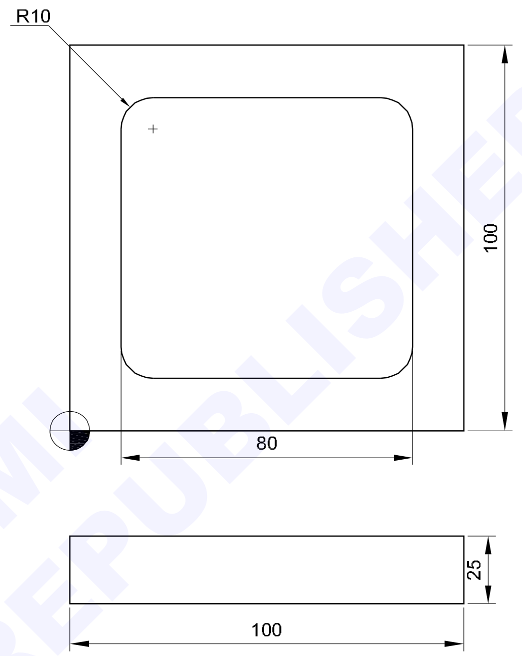
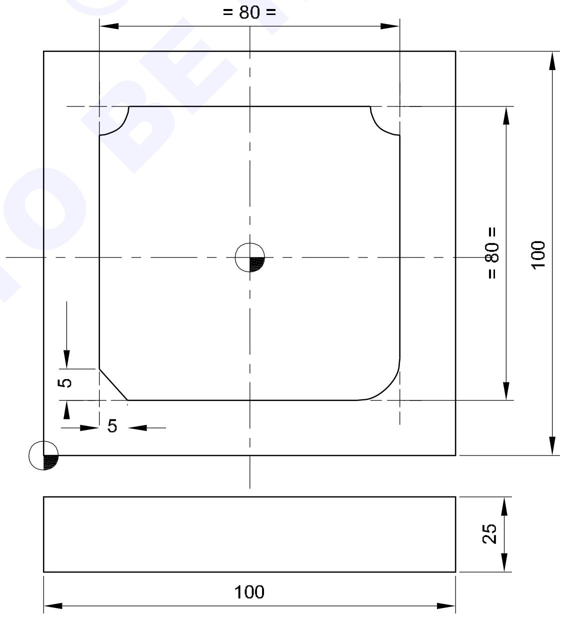


Fig 3



–	PRE MACHINED 100 X 100 X 25 mm	–	ALUMINIUM-LM25	–	1	2.4.176
NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.
SCALE 1 : 1	CNC PROGRAM				TOLERANCE ±0.02	TIME
					CODE NO. MA20N24176E1	

1 **Task 1 -** Write the CNC programme for Fig 1 in absolute co-ordinate system (only to mark the profile).

2 **Task 2 -** Write the CNC programme for Fig 1 in incremental co-ordinate system.

3 **Task 3 -** Write the CNC programme for Fig 2 in absolute co-ordinate system.

4 **Task 4 -** Write the CNC programme for Fig 2 in incremental mode.

5 **Task 5 -** Write the CNC programme for Fig 3 in absolute co-ordinate system.

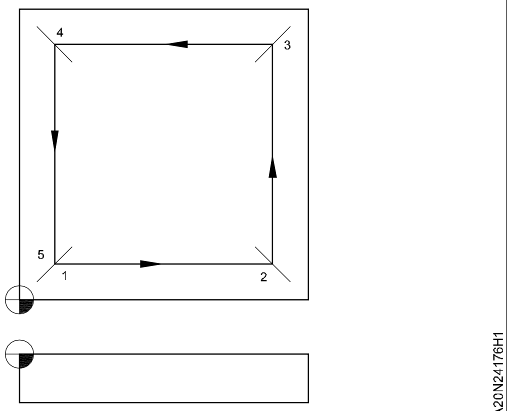
6 **Task 6 -** Write the CNC programme for Fig 3 in incremental co-ordinate system.

Job Sequence

TASK 1: **Write the co-ordinates points in absolute and incremental co-ordinate systems**

Write the co-ordinates ponts for Fig 1 in G90

Fig 1



Point	G Code	X	Y	Z
1	G00	10	10	5
1	G01	10	10	-0.5
2	G01	90	10	
3	G01	90	90	
4	G01	10	90	
5	G01	10	10	
6	G00	10	10	50

Write the CNC Programme

N1 G71 G94 (Metric & feed rate in mm/minutes)

N2 G75 Z0 (Go to machine reference Z axis)

N3 G75 X0 Y0 (go to machine reference X and Y axis)

N4 M06 T01 D1 (Tool call number and tool height D1)

N5 M03 S1500 (spindle on CW with RPM M 1500)

N6 M08 (coolant on)

N7 G00 G93 G54 X10 Y0 Z5 (Rapid movement to starting, position X10 Y0 Z5)

N8 G01 X10 Y10 Z-0.5 F100 (Depth of cut in work piece 0.5mm)

N9 G01 X90 Y10 (linear interpolation on from point 1 to Point 2)

N10 G01 X90 Y10 (linear interpolation on from point 2 to 3)

N 11 G0 X90 Y10 Linear movement from point 3 to 4)

N12 G01 X10 Y10 (Linear movement from point 4 to point 5)

M13 G00 Z50 (tool moves upto by 50 mm above work piece rapidly)

M14 M09 (coolant off)

M15 M05 (spindle off)

M16 M30 (program stop)

Enter the programme in CNC simulator

Verify the programme by simulating

If there is any error in the tool path correct it accordingly

— — — — — — — —

TASK 2: **Write programme for Fig 1 in incremental mode**

Write the incremental coordinates for Fig 1 (G91)

	G Code	X	Y	Z
N1	G01	0	0	-5.50
N2	G01	80	0	
N3	G01	0	80	
N4	G01	80	0	
N5	G01	0	-80	
N6	G00			50

Write the CNC Programme

N1 G71 G94 (Metric & feed rate in mm/minutes)

N2 G75 Z0 (Go to machine reference Z axis)

N3 G75 X0 Y0 (go to machine reference X and Y axis)

N4 M06 T01 D1 (Tool call number and tool height D1)

N5 M03 S1500 (spindle on CW with RPM M 1500)

N6 M08 (coolant on)

N7 G00 G93 G54 X10 Y0 Z5 (Rapid movement to starting position X10 Y0 Z5)

N8 G01 X0 Y0 Z-5.50;

N9 G01 X80 Y0

N10 G01 X0 Y80

N11 G01 X80 Y0

N12 G01 X0 Y-80

N13 G00 Z50 (tool moves upto by 50 mm above work piece rapidly)

N14 M09 (coolant off)

N15 M05 (spindle off)

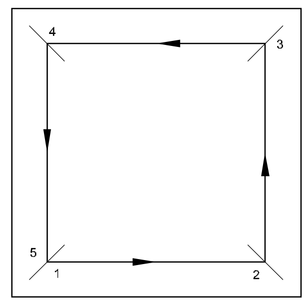
N16 M30 (program stop)

Enter the programme in CNC simulator

Verify the programme by simulating

If there is any error in the tool path correct it accordingly

Fig 1



— — — — — — — —

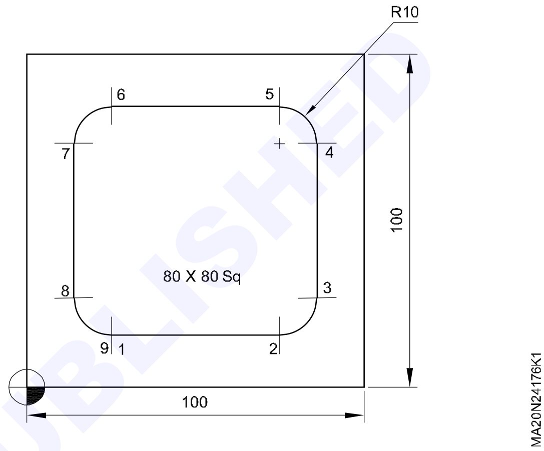
TASK 3: **Write the CNC programme for Fig 1 in absolute co-ordinate system**

Write the co-ordinate for Fig 1

G90

	G	X	Y	Z	I	J	F
N1	G00	20	10	5			
	G01	20	10	-0.5			100
N2	G01	80	10				
N3	G03	90	20		0	10	
N4	G01	80	80				
N5	G03	70	90		-10	0	
N6	G01	20	90				
N7	G03	10	80		0	-10	
N8	G01	10	20				
N9	G03	20	10		10	0	
N10	G00	20	10	50			

Fig 1



Write the CNC programme

N1 G71 G94

N2 G75 Z0

N3 G75 X0 Y0

N4 M06 T01 D1

N5 M03 S1500

N6 M08

N7 G00 G90 G54 X20 U10 Z5

N8 G00 X20 Y10 Z5

G01 X20 Y10 Z-0.5

N9 G01 X80 Y10

N10 G03 X90 Y20 I0 J10

N11 G01 X80 Y80

N12 G03 X70 Y90 I-10 J0

N13 G01 X20 Y90

N14 G03 X10 Y80 I0 J-10

N15 G01 X10 Y20

N16 G03 X20 Y10 I10 J0

N17 G00 X20 Y10 Z50

N18 M09

N19 M05

N20 M30

Enter the CNC programme in the CNC simulator

Verify the programme by simulating

— — — — — — — —

TASK 4: **Write the CNC programme for the Fig 1 in incremental co-ordinates**

Write the incremental co-ordinates for Fig 1

G91

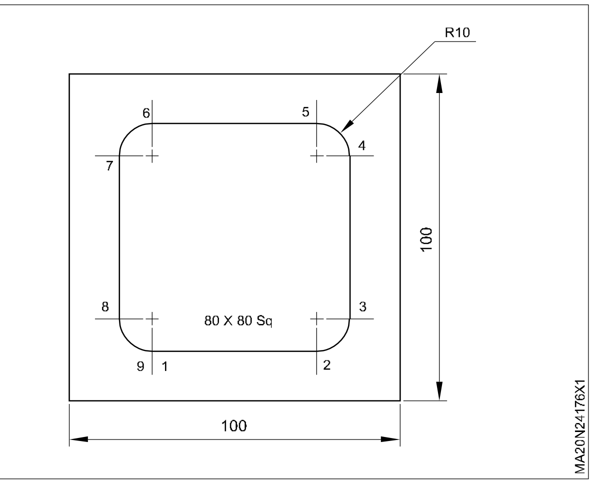
Sl.No	G code	X	Y	Z	I	J
N1	G01	0	0	-5.5	-	-
N2	G01	60	0			
N3	G03	10	10		0	10
N4	G01	0	60			
N5	G03	-10	10		-10	0
N6	G01	- 60	0			
N7	G03	-10	- 10		-10	0
N8	G01	0	- 60			
N9	G03	10	- 10		10	

Write the CNC Programme

N1 G71 G94

N2 G75 Z0

Fig 1



N3 G75 X0 Y0

N4 M06 T01 D1

N5 M03 S1500

N6 M08

N7 G00 G90 G54 X20 Y10 Z5

N8 G01 G91 X0 Y0 Z-5.5 F100

N9 G01 X0 Y0 Z-5.5

N10 G01 X60 Y0

N11 G03 X10 Y10 I0 J10

N12 G01 X0 Y60

N13 G03 X-10 Y10 I-10 J0

N14 G01 X-60 Y0

N15 G03 X-10 Y-10 I0 J-10

N16 G01 X0 Y-60

N17 G03 X10 Y-10 I10 J0

N18 G00 G90 Z50

N19 M05 M09

M30

Enter the CNC programme in CNC simulator

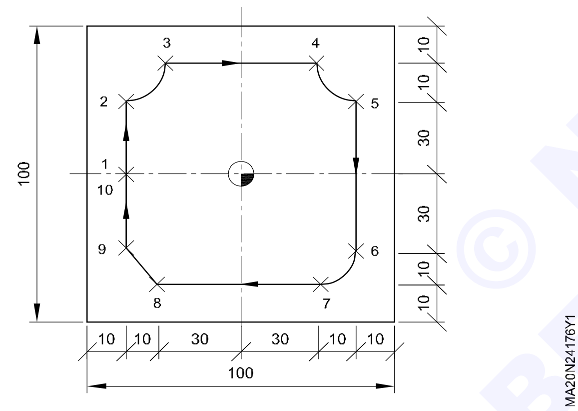
Verify the programme by simulating

If there is any error correct the programmes accordingly

— — — — — — — —

TASK 5: **Write the CNC programme in absolute co-ordinate system for Fig 1**

Fig 1



S.No	Tool path	X	Y	Z	R
1	G00	-40	0		
2	G01	-40	30		
3	G03	-30	40		CR=10
4	G01	30	40		
5	G03	40	30		CR=10
6	G01	40	-30		
7	G02	30	-40		CR=10
8	G01	-30	-40		
9	G01	-40	-30		
10	G01	-40	0		

Write the CNC programme

N1 G71 G94 (Metric & feed)

N2 G75 Z0 (go to machine ref Z axis)

N3 G75 X0 Y0 (go to machine ref X and Y axis)

N4 M06 T01 D1 (Tool calling no.1 and tool height calling D1)

N5 M03 S1500 (Spindle on CW. RPM 1500)

N6 M08 (coolant on)

N7 G00 G90 G54 X0 Y0 (Rapid mode tool work offset position X,Y)

N8 G00 Z5 (Rapid mode Z axis position safety distance)

N9 G00 X-40 Y0 (Tool starting position in program mode)

N10 G01 Z0 F300 (Linear mode with feed rate job surface position 0)

N11 G01 Z-1 F50 (depth of cut in Job Z axis is feeded)

N12 G01 Y30 F 300 (Linear mode cutting feed Y axis with feed rate)

N13 G03 X-30 Y 40 CR=10 (tool path X and Y anticlockwise with radius) CR Center radius

N14 G01 X30 Y40 (Tool path linear X axis)

N15 G03 X40 Y30 CR = 10 (Tool path X and Y anti clock wise with radius)

N16 G01 X40 Y-30 (Tool path linear mode Y axis)

N17 G02 X30 Y-40 CR=10 (Tool path X and Y clock wise with radius)

N18 G01 X-30 Y -40 (Tool linear mode X axis)

N19 G01 X-40 Y -30 (Tool path linear mode X and Y axis)

N20 G01 X-40 Y0 (Tool path starting point linear mode)

N21 G00 G90 Z20 (tool safe position rapid mode)

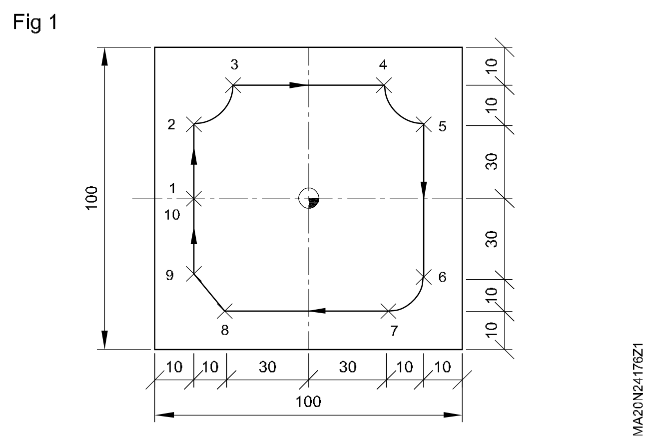
N22 M09 (Coolant off)

N23 M05 (Spindle off)

N24 M30 (Program rewind)

TASK 6: **Write the co-ordinates in incremental mode for Fig 1**

Fig 1



S.No	Tool path	X	Y	Z	R
1	G00	-40	0		
2	G01	-0	30		
3	G03	10	10		CR=10
4	G01	60	0		
5	G03	10	-10		CR=10
6	G01	0	-60		
7	G02	-60	-10		CR=10
8	G01	-10	0		
9	G01	-10	10		
10	G01	0	30		

Write the program in Incremental mode

N1 G91 incremental

WORK PIECE (, " " , , "RECTANGLE", 64,0,-20,-80,100,100)

N2 G71 G94 (Metric feed)

N3 G75 Z0 (Go to machine Ref. Z axis)

N4 G75 X0 Y0 (Go to machine Ref. X and Y axis)

T= 'CUTTER4' M6 D1 (Tool calling, Tool length)

N5 M03 S1500 (Spindle on CW)

N06 M08 (coolant on)

N07 G00 G90 G54 X0 Y0 (Tool work offset position with Rapid mode)

N08 G00 Z5 (Z axis safety position)

N09 G00 X-40 Y0 (Tool path starting point)

N10 G01 Z-1 F50 (depth of cut Z axis)

N11 G01 Y 30 F300 (Tool path linear Y axis)

N12 G03 X10 Y10 CR=10 (Tool path Y axis Anti clock wise with radius)

N13 G01 X60 Y0 (Tool path linear X axis)

N14 G03 X10 Y-10 CR=10 (Tool path X, Y axis anti clock wise with radius)

N15 G01 X0 Y-60 (Tool path linear Y axis)

N16 G01 X-10 Y-10 CR=10 (Tool path X, Y axis clock wise with radius)

N17 G01 X-60 Y0 (Tool path linear X axis)

N18 G01 X-10 Y10 (Tool path linear X Y axis)

N19 G01 X0 Y30 (Tool path linear starting)

N20 G00 G90 Z20 (Tool safe distance)

N21 M09 (Coolant off)

N22 M05 (Spindle off)

N23 M30 (Program rewind)

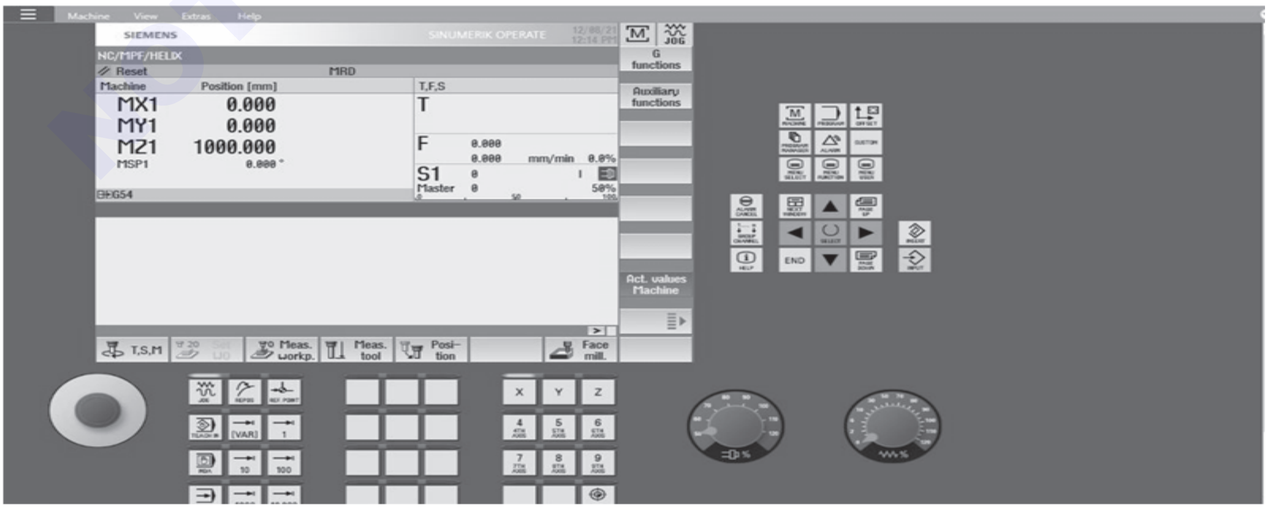
Skill Sequence

Edit or enter a new program in Siemens 828 D control

Objective: This shall help you to
- **enter a new program in Siemens 828 D control.**

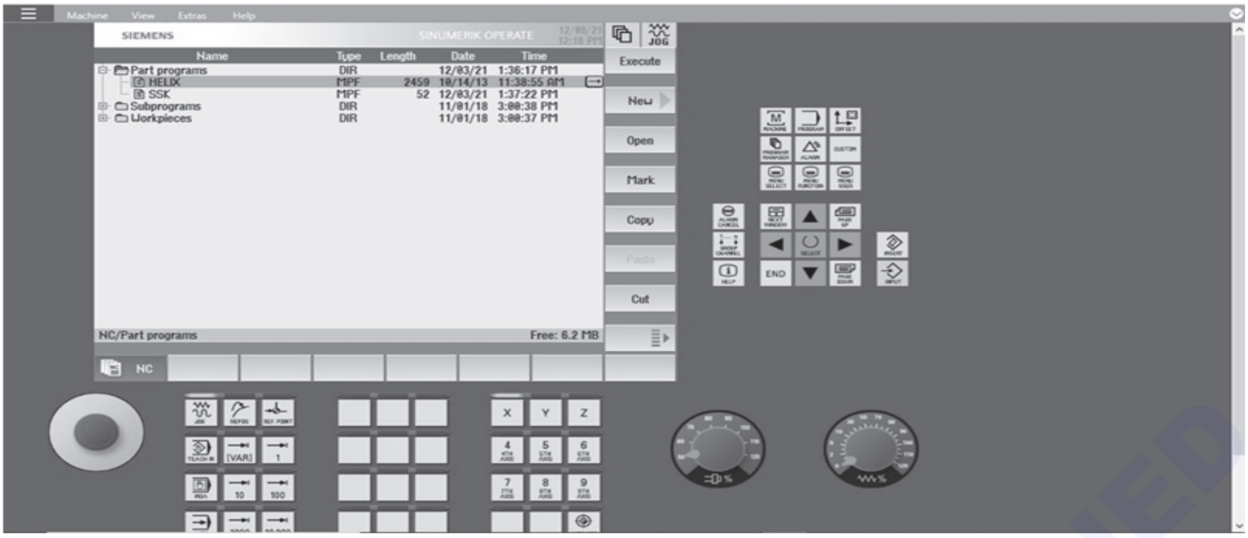
Siemens control 828 panel (Fig 1)

Fig 1



1 Select program manager key press in HMI panel (Fig 2)

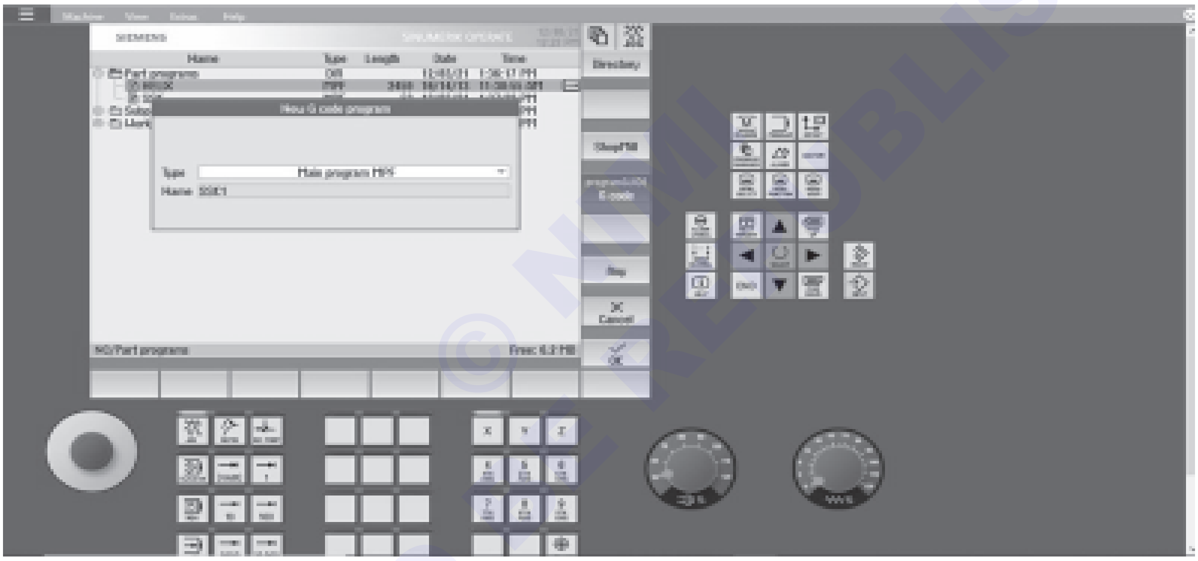
Fig 2



2 Select part program key in HMI panel

3 Select new press in soft key (Fig 3)

Fig 3



4 Enter the file name press ok

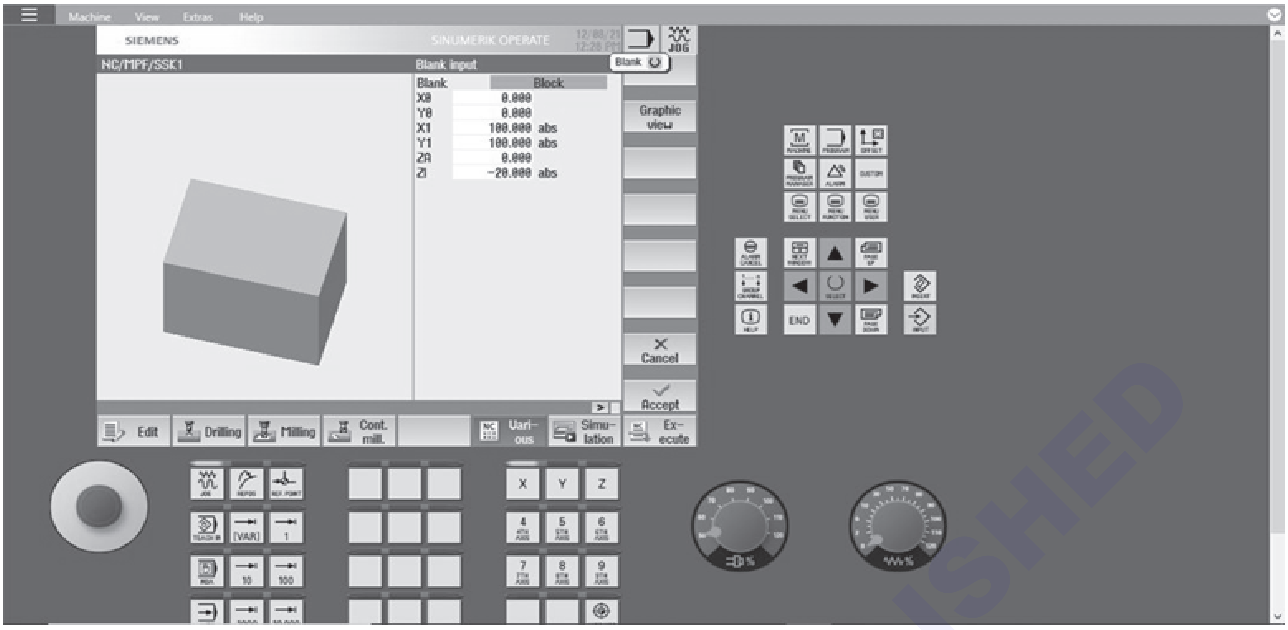
5 File screen open (Fig 4)

Fig 4

6 Select block and enter the size (use various soft key)

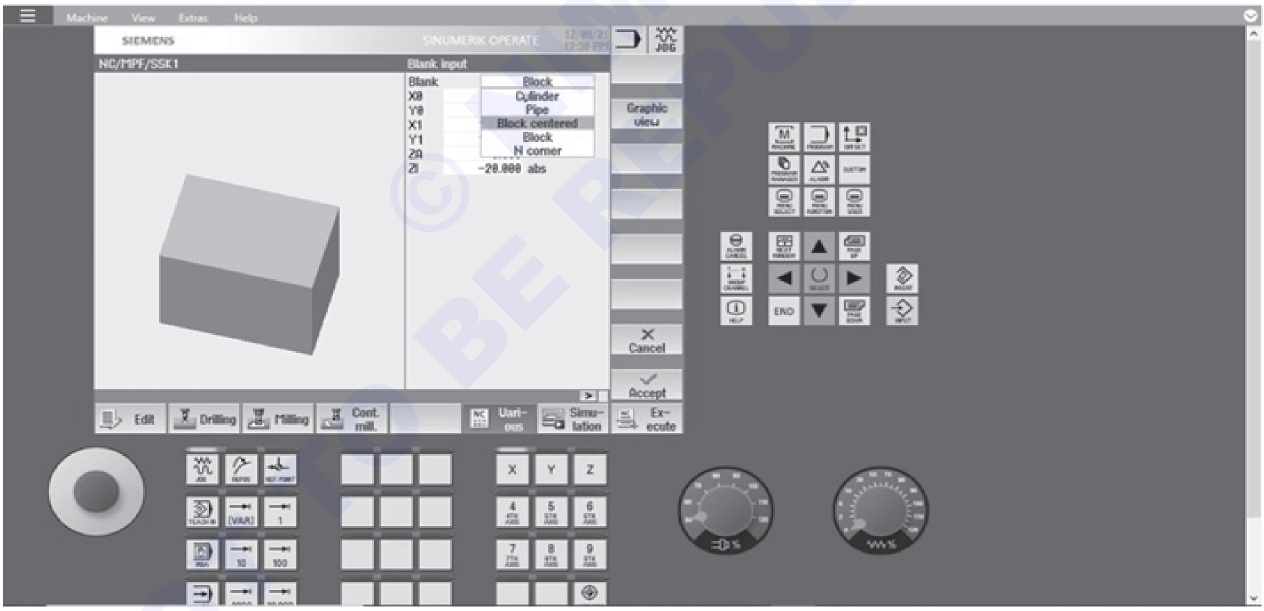
7 Press blank in soft key (Fig 5)

Fig 5



8 Select block centred (Fig 6)

Fig 6



9 Block entered select input key (Fig 7)

10 Enter the material size accept press isoft key (Fig 8)

11 Work piece (, " " ,, " RECTANGLE", 64, 0 -20, 80,100,100)___

12 Type the program using alpha numerical key pad (Fig 9)

Fig 7

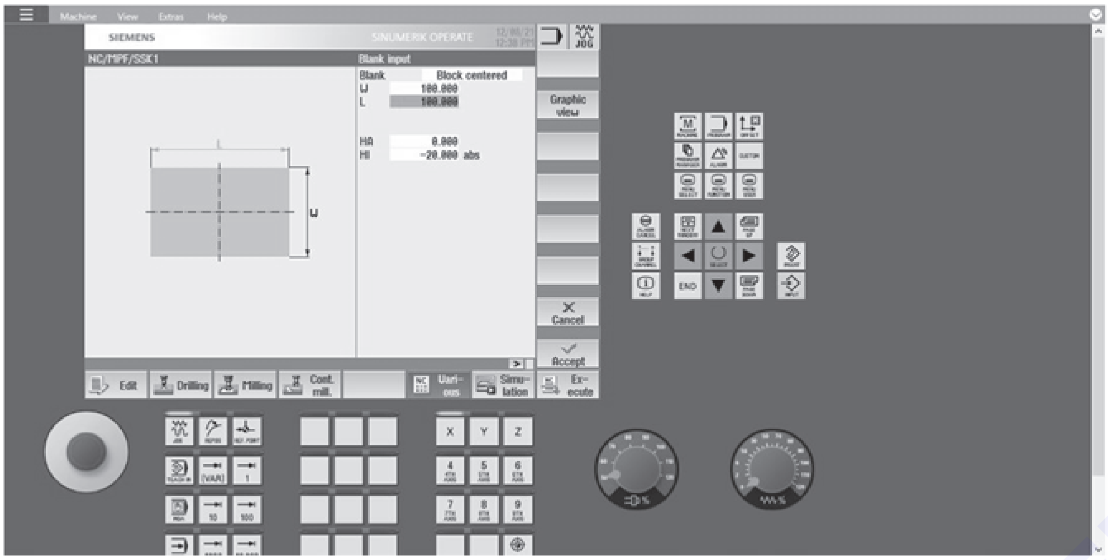


Fig 8

Fig 9

Simulation checking method in Siemens control

Objective: This shall help you to
- **simulate the part program.**

1 Open main program (Fig 1)

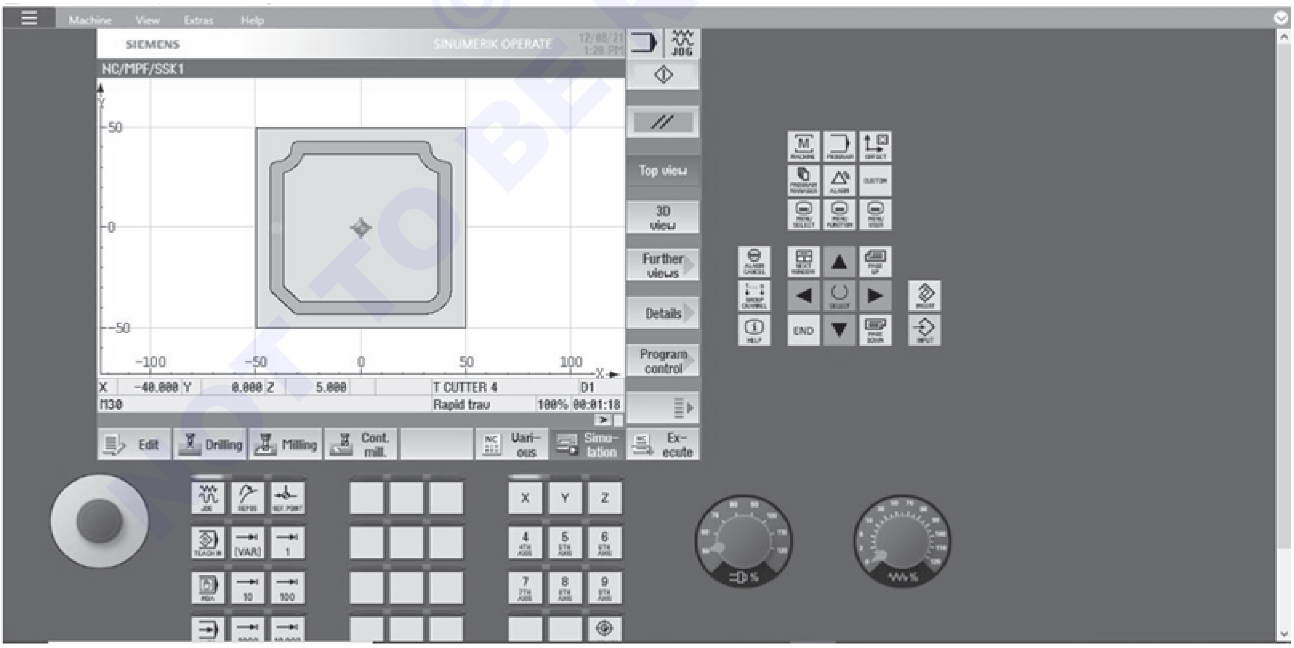
Fig 1

2 Press simulation soft key

3 Simulation will start

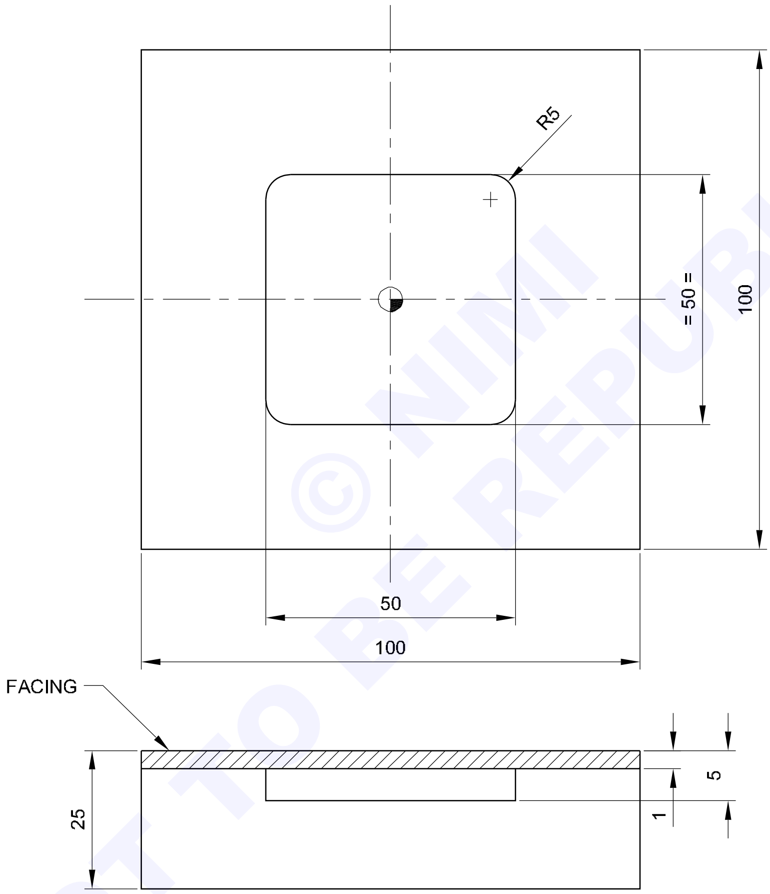
4 Simulation show the finished product (Fig 2)

Fig 2



Facing and slot milling with sub program technique

Objectives : At the end of this exercise you shall be able to
- **write the program for face milling**
- **write the program for slot milling**
- **enter the program in simulator and verify the program.**



NOTE:- 1. FACE MILL CUTTER DAIMETER 50
2. SIOT MILL CUTTER DIAMETER 10MM

1	–	–	ALUMINIUM	–	–	2.4.177
NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.

SCALE 1 : 1	**FACING AND SLOT MILLING**	TOLERANCE	TIME
		CODE NO. MA20N24177E1	

Job Sequence

Note

1 Face mill cutter diameter 50 mm

2 Slot mill cutter diameter 10 mm

Fanuc Control programme

O0005; (Face milling and pocket milling)

Program Header

1 G40 G15 G69 G 80
2 G 21 G94
3 G28 G91 Z0
4 G28 G90 G55 X0 Y0
5 M06 T05 M03 S1800;

Body of the Programme

6 G00 G43 H5 Z5
7 G00 X-75 Y-35
8 G01 Z0 F500
9 M98 P6 L2
10 G00 G90 I 20
11 M05
12 M09
13 M06 T01
14 M03 S1500
15 G00 G90 G55 X0 Y0
16 G00 G43 H1 Z5
17 G01 Z0 F200
18 M98 P7 L10

Programme footer

19 G00 G90 Z20
20 M05;
21 M09;
22 M30;

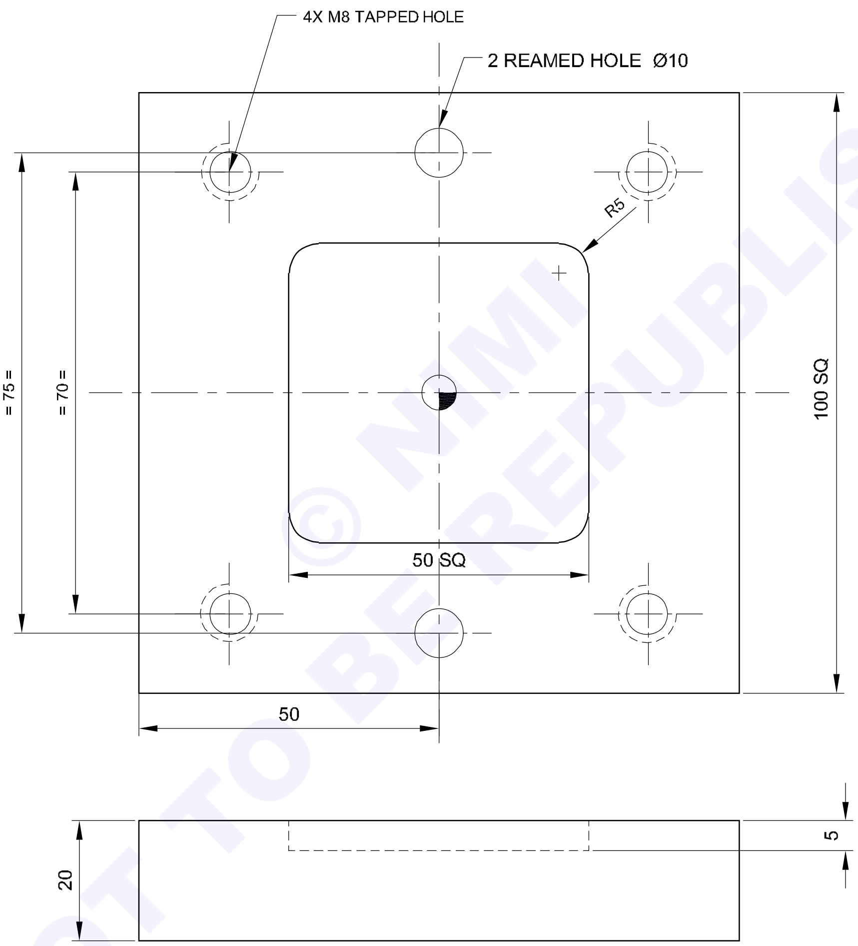
Note: The body of the programme will remain same for most of the control, only header and footer will vary

Sub Programme

O0006; (facing)
N1 G1 G91 Z-0.5 F200
N2 G1 G90 X75 F300
N3 G1 Y0;
N4 G1 X-75;
N5 G1 Y 35;
N6 G1 X75;
N7 G0 G91 Z1;
N8 G0 G90 X-75 Y-35;
N9 G00 G091 Z-2;
N10 M99;
O0007: (pocket)
G01 G91 Z-0.5 F50;
G01 G90 X-8 F300;
G01 Y8;
G01 X8;
G01 Y-8;
G01 X-8;
G01 Y0;
G01 X-16;
G01 Y 16;
G01 X 16;
G01 Y -16;
G01 X- 16;
G01 Y0;
G01 X-20;
G01 Y20;
G01 X20;
G01 Y-20;
G01 X-20;
G01 Y0;
G0 X0 Y0;
N99;

CNC part programme, pocket milling and - canned cycles

Objectives : At the end of this exercise you shall be able to
- **write the part programme for pocket milling**
- **write the part programme for centre drilling**
- **write drilling and tapping programme using canned cycles**
- **write the part programme for reaming using canned - cycle.**



–	–	–	–	–	–	2.4.178
NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.
SCALE 1 : 1	CNC PART PROGRAMS - POCKET MILLING				TOLERANCE	TIME
					CODE NO. MA20N24178E1	

Job Sequence

- Write the CNC programme for pocket milling
- Write the CNC programme for centre drilling
- Write the CNC programme for drilling
- Write the CNC programme for counter sinking
- Write the CNC programme for taping
- Write the CNC programme for reaming
- Enter all te programme in CNC simulator and get it verified by your instructor

Program for pocket and canned cycles

Task 1 Pocket milling

O0010 (Pocket milling)

N1	G40	G15	G69	G80;
N2	G21	G94;		
N3	G28	G91	X0	Y0 Z0;
N4	M06	T05	(slot drill ∅10mm)	
N5	M03	S1000;		
N6	G90	G55	G00	X0 Y0;
N7	G43	H5	Z5;	
N8	G01	Z0	F200;	
N9	M98	P20	L10;	
N10	G00	G90	Z20;	
N11	M05;			
N12	M09;			
N13	M30;			

Sub program

O00020 (packet)

N1	G02	G91	Z-0.5 F50;
N2	G01	G90	X-8 F300;
N3	G01	Y8;	
N4	G01	X8;	
N5	G01	Y-8;	
N6	G01	X-8;	
N7	G01	Y0;	
N8	G01	X-16;	
N9	G01	Y 16;	
N10	G01	X16;	
N11	G01	Y-16;	
N12	G01	X-16;	
N13	G01	Y0;	
N14	G01	X-20;	
N15	G01	Y20;	
N16	G01	X20;	
N17	G01	Y-20;	
N18	G01	X-20;	
N19	G01	Y0;	
N20	G0	X0 Y0;	
N21	M99;		

Task 2 Centre drilling

O0011

N1	G40	G15	G69	G80;
N2	G21	G94;		
N3	G28	G91 Z0;		
N4	M07	T06 (center drill bit);		
N6	G90	G00	G55	X0 Y0;
N7	G43	H6	Z5;	
N8	G00	X-35	Y-35;	
N9	G81	X-35	Y-35	Z-6R2 F50;
N10	X-35	Y35;		
N11	X35	Y35;		
N12	X35	Y-35;		
N13	G00	G80	Z100;	
N14	M05;			
N15	G28	G91	G00	X0 Y0 Z0;
N16	M30;			

Task 3 Drilling

O0012

N1	G40	G15	G69	G80;
N2	G21	G94;		
N3	G28	G91	Z0;	
N4	M07	T07 (drill ∅6.8);		
N5	M03	S1000		
N6	G90	G00	G55	X0 Y0;
N7	G43	H7	Z5;	
N8	G00	Z-35	Y-35;	
N9	G83	X-35	Y-35 Z-25 Q10 R2 F50;	
N10	X-35	Y35;		
N11	X35	Y35;		
N12	X35	Y-35;		
N13	G00	G80	Z100;	
N14	M05;			
N15	G28	G91	G00	X0 Y0 Z0;
N16	M30;			

Task 4 (Counter sinking)

O00013

N1	G40	G15	G69	G80;
N2	G21	G4	G90;	
N3	G28	G91	Z0;	
N4	M06	T15 (counter sink 90°∅15);		
N5	M03	S250;		
N6	G55	X-35	Y-35 Z100 H15;	
N7	G81	X-35	Y-35	Z-5 R2 F50;
N8	X-35	Y35;		
N9	X35	Y35;		
N10	X35	Y-35;		
N11	G00	G80	Z100;	
N12	M05;			
N13	G28	G91	G00	X0 Y0 Z0;
N14	M30;			

Task 5 (Tapping)

O0014

N1	G40	G15	G69	G80;
N2	G21	G94;		
N3	G28	G91	Z0;	
N4	M07	T08 (Tap M8);		
N5	M03	S100;		
N6	G90	G00	G55	X0 Y0;
N7	G43	H8 Z5;		
N8	G00	X-35	Y-35;	
N9	G95	G84	X-35 Y-35 Z-25 R2 G1.25;	
N10	X-35	Y35;		
N11	X35	Y35;		
N12	X35	Y-35;		
N13	G00	G80	Z100;	
N14	M05;			
N15	G28	G91	G00	X0 Y0 Z0;
N16	M30;			

Task 6 (Reaming)

O0015

N1	G40	G15	G69	G80;	
N2	G21	G94;			
N3	G28	G91	Z0;		
N4	M07	T06 (center drill bit)			
N5	M03	S250;			
N6	G90	G00	G55 X0 Y0;		
N7	G43	H6 Z5;			
N8	G00	X-35	Y-35;		
N9	G81	X-35	Y-35	Z-6 R2 F50;	
N10	X-35	Y35;			
N11	X35	Y35;			
N12	X35	Y-35;			
N13	G00	G80	Z100;		
N14	M05;				
N15	G28	G91	G00	X0 Y0 Z0;	
N16	M06	T08; (drill .. 9.5);			
N17	M03 S750;				
N18	G90	G55	G00	H8	0 Y 37.5;
N19	G98	G81	X0 Y37.5 Z-25 R2 F100;		
N20	X0	Y-37.5;			
N21	G00	G93	G80	Z100;	
N22	G91	G28	X0 Y0 Z0;		
N23	G90;				
N24	M06	T09 (Reamer …10);			
N25	M03	S250;			
N26	G43	H9 Z100;			
N27	G90	G00	G55	X0 Y37.5;	
N28	G85	X0	Y37.5 Z-25 F50;		
N29	G00	G90	G80	Z100;	
N30	G91	G28	G00	X0 Y0 Z0;	
N31	M05;				
N32	M30;				

Avoiding collision caused by program error

Objectives : At the end of this exercise you shall be able to
- **check the program for its correctness**
- **make deliberate errors in program to-have collisions on simulator**
- **verify the causes and effects of errors in the program.**

Note: This exercise shall be carried out on virtual shop floor simulator

Job Sequence

- Load any simple program to execute in automode operation
- Read the program and understand the tool movement with respect to the work
- Plat tool path with respect to the program
- Simulate the program and verify the correctness
- Now run the same program in single block mode
- Observe the distance to move from the present position to the targeted position
- Physically observe, if there is any obstruction for the movement
- Release feed hold button and the tool will move the targeted position
- Similarly check all the blocks in the program
- Now make changes in Z axis movement that is the 'Z' axis value should touch the table surface
- Reset program
- Execute the program in single block be careful in the Z axis movement particularly 'Z' axis value changed block
- Observe the distance to in 'Z' axis of your release the feed hold button the tool will collide with the table
- Release the feed hold button and observe the collision in simulator
- Similarly change the values in X and Y axis and observe the over travels.

Example: Correct program

Exp No.1: FACE MILLING

Draw the tool path for the program O00001

Cutter dia 50 work piece 100 x 100

O0001:

N5 G40 G49 G50 G80 G69;

N10 G90 G51 G94;

N15 T06 M19;

N20 S600 M03;

N25 G00 G55 G43 H06 X20 Y-30 Z50;

N30 G00 Z0 F100 M07;

N35 G01 X20 Y130;

N40 G00 Z5;

N45 G00 X65 Y130 Z0;

N50 G01 X65 Y-30;

N55 G00 Z5;

N60 G00 X110 Y130;

N65 G00 Z0;

N70 G01 X110 Y130;

N75 G00 Z50;

N80 G91 28 X0 Y0 Z0 M09;

N85 M30;

- Change 'Z' value G00 Z0 in to G00 z-50 in block number 65, and run the program in single block (only in simulator) and observe the effect
- Similarly change the X20 y130 value into X-100 in block number 35 and observe the over travel in 'X' axis

Wrong Program

O0001:

N5 G40 G49 G50 G80 G69;

N10 G90 G21 G94;

N15 T06 M19;

N20 S600 M03;

N25 G00 G55 G43 H06 X20 Y-30 Z50;

N30 G00 Z0 F100 M07;

N35 G01 X20 Y130;

N40 G00 Z5;

```
N45 G00 X65 Y130 Z0;
N50 G01 X65 Y-30;
N55 G00 Z5;
N60 G00 X110 Y130;
N65 G00 Z-50;
N70 G01 X110 Y130;
N75 G00 Z50;
N80 G91 28 X0 Y0 Z0 M09;
N85 M30;
```

Note:

- **The SINUMERIK collision avoidance option provides optimum protection against unwanted collision of moving machine components with static machine components. The collision monitoring is also possible for complex machining, such as 5-axis simultaneous milling and turning with the B axis**
- **Settings Collision avoidance**
- **The collision monitoring can be activated in the Machine operating area for the JOG, MDA and automatic operating modes.**

Conduct a preliminary check on the CNC VMC machine

Objectives : At the end of this exercise you shall be able to
- **check the cleanliness of the machine**
- **check the voltage stabilizer for current and voltage**
- **check the power back for the condition**
- **check the coolant tank and the cutting fluids.**

Job Sequence

Cleaning

- Check the cleanliness of the machine outside and its surrounding
- Check the cleanliness of the machine inside table slides column etc.
- Check the condition of the tool in the tool magazine and cleanliness check the cleanliness of the spindle

Voltage stabilizer

- Check the voltage stabilizer and its functions
- Check the income voltage and current
- Check the output voltage and current
- Observe for any voltage fluctuation

Power back

- Check the hydraulic power back
- Check the oil level in the tank
- Check the condition of the oil
- Check for any leakage of oils

Centralised lubrication system

- Check the function of centralised lubricating system
- Check the main pressure of the hydraulic system

Coolant

- Check the condition of coolant tank
- Check the condition of the cutting fluid in the coolant tank
- Check the level of coolant in the coolant tank

> **Note:**
> - **If you found anything unusual or any loose parts immediately inform to your instructor**

Machine starting, and homing

Objectives: At the end of this exercise you shall be able to
- **start the CNC machining centre**
- **reference the machine axes**
- **operate in jog mode**
- **operate in incremental and MDI modes.**

Job Sequence

TASK 1**: Starting the CNC machining centre**

- Switch on the stabilizer main switch
- Make sure the stabilizer is in servo mode
- Check the stabilizer output voltage is in between 400-430v
- Switch on machine main switch
- Switch on control panel switch, machine computer screen will start working
- The axis display and other details appears and emergency indication flickering on the screen
- Release the emergency push button and reset the machine
- Now the machine is ready for referencing.

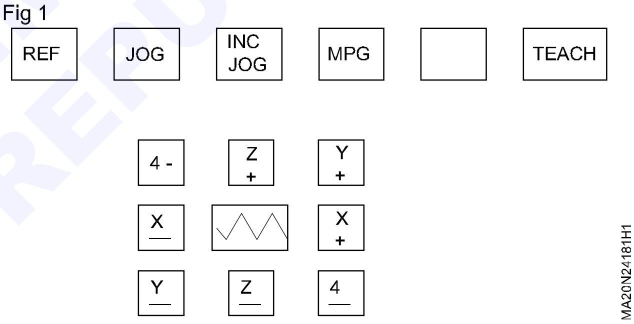
— — — — — — — —

TASK 2: **Reference position return**

- Start the machine
- Go to jog mode by pressing jog switch
- Move all the axis towards the centre of the machine table by selecting appropriate axis switches
- Go to reference point return by pressing the "ref switch. (Fig 1)
- Press the "X +" "Y+" "Z+ and "C+" switches. All the axes are referred to reference point level and the reference position return completion LED will glow
- Now the display shows the following position X=0.000 Y=450.00 Z = 420.00

Display and steps may varey machine to machine

Fig 1



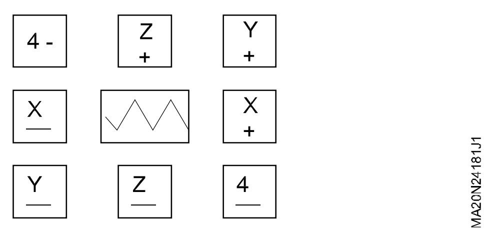
- The reference position may be reached by giving the following command by selecting the MDI mode in between the operation after first time reaching the reference point position G0 G91 G28 X0 Y0 Z0 B0.

— — — — — — — —

TASK 3: **Jog mode operation**

- Press the "Jog" switch in the keyboard
- Keep the feed over ride switch near to 50% position
- Press the appropriate axis with direction switch "continuously" until the desired movement is achieved. (Fig 1)

Fig 1



- The movement may be made rapid by simultaneously pressing the axis and rapid switch

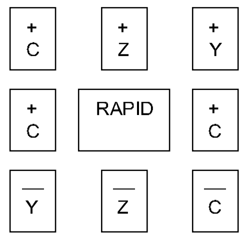
• **If the finger is released from the switch the movement is stopped immediately** • **The feed rate may be increased or decreased as desired by changing the feed override switch position** **The axis may be stopped at '0' position.**

TASK 4: **Incremental JOG / MPG mode**

- Press "Inc Jog" switch
- Press any one of the inc x1, inc 10, inc 100, inc 1000, inc 10000
- Press the axis switch (+) or (-) to move for the particular incremental feed (Fig 1).

  (or)
- Activate MPG (manual pulse generator) switch
- Press any one of the inc x1………x10.000 switch as desired
- Press the axis with direction + or
- Rotate the MPG knob. The movement Per division is equal to the selection of inc switch.

Fig 1



MA20N24181Y1

TASK 5: **MDI mode - operation**

- Press the MDI key.
- Press the 'program' key.

  The following screen appear (may varey machine to machine) (Fig 1)

Fig 1

PROGRAM (MDI)
O0000
%

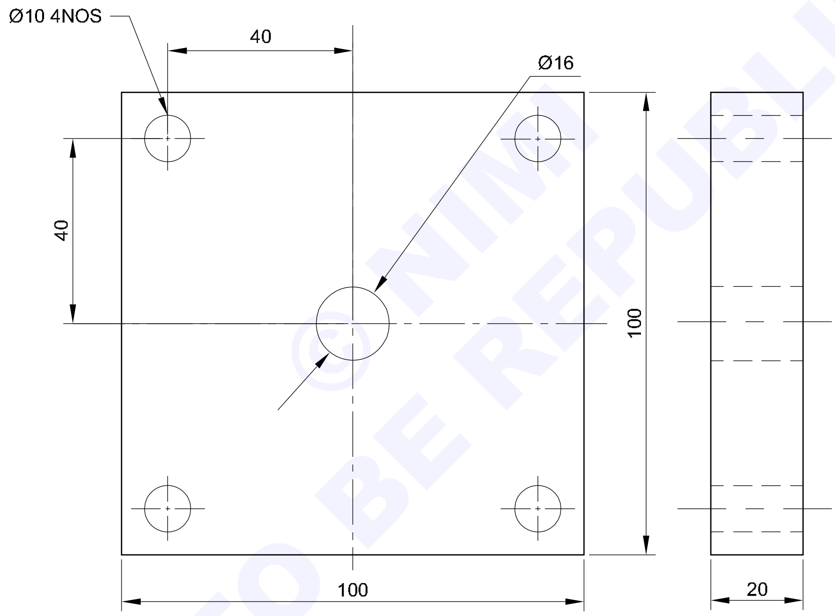
10:26:11 S OT
MDI
PROGRAMMING NUMBER O0000 IS GENERATED AUTOMATICALLY

MA20N24181X1

- Prepare the program blocks with a block M 99 at the end to return to beginning the block.
- To erase the program created in MDI either press "Reset" key or enter address O0000 then press the "Delete" key in the MDI panel.
- Place curser in the first block and push cycle start key for executing position.
- To stop the operation press "Reset" key (or) rotate the "Feed holed" key to "0" position (or) press the emergency switch.

Entering the CNC program in edit mode

Objectives : At the end of this exercise you shall be able to
- **prepare program for face milling**
- **prepare program for drilling without canned cycle**
- **enter the program on CNC simulator and verify it.**



–	–	–	–	–	–	2.4.182
NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.

SCALE 1 : 1	ENTRING THE PROGRAM IN EDIT MODE	TOLERANCE	TIME
		CODE NO. MA20N24182E1	

Job Sequence

TASK 1: **Write the CNC program for face milling**

- Face milling (program)

  Cutter dia 50 work piece 100 x 100

  00001: (face milling)

  N5 G 40 G49 G50 G80 G69

  N10 G90 G21 G94

  N 15 T06 M19 (Tool change command)

  N 20 S600 M03

  N25 G00 G55 G43 H06 X 20 Y-30 Z50 (Tool length compensation in '+ ' direction)

  N 30 G00 Z0 F100 M07;

  N35 G01 X20Y 130;

  N 40 G00X65Y 130 Z0;

  N45 G01 X65Y-30;

  N50 G00 X110 Y -30;

  N55 G01 X 110Y 130;

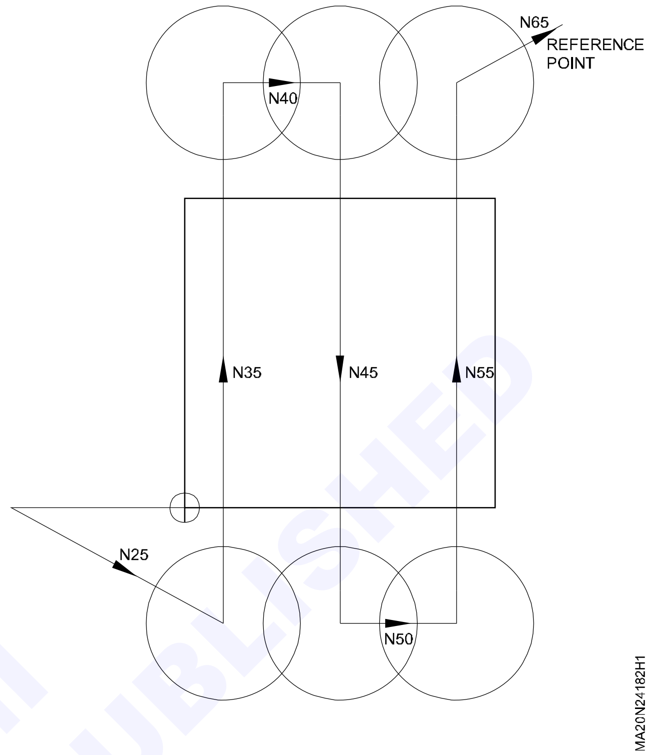
  N 60 G00 Z 50;

  N 65 G91 G28Z0 Y0 Z0 M09: (Return to reference point)

  N 70 M 30;

  Tool path for face milling is shown in Fig 1

Fig 1



N = Indicate line number in programme

Dotted line indicates the movement with rapid (G00)

Thick line indicate movement with feed rate (G01)

TASK 2: **Write CNC program for drilling without canned cycles**

00002: (Drilling)

N5 G 40 G49 G50 G80 G69;

N10 G 90 G21 G94:

N15 T02 M19: (centre drill);

N 20 S 600 M 03;

N25NG 00 G55 G43 H 02 X0 Y0 Z5;

N 30 G 01 Z-7 F 50;

N35 G 04 X3;

N40 G 00 Z5;

N45 G00 X 40 Y 40;

N50 G01 Z-5 F 50;

N55 G 04 X3;

N60 G 00 Z5 ;

N65 G00 X - 40 Y 40;

N70 G 01 Z - 5 X 3;

N75 G 04 X 3;

N80 G 00 Z 5;

N85 G 00 X - 40 Y - 40;

N90 G 01 Z - 5 F 50;

N100 G 04 X 3;

N105 G 00 Z5;

N115 G 00 X 40 Y - 40;

N120 G 01 Z - 5 F 50;

N125 G 04 X 3;

N130 G00 Z5;

N135 M 05;

N140 G28 G91 G 00 X0 Y0 Z0;

N145 T 03 M19;

N150 S 600 M 03;

N155 G00 G 90 G 55 G 43 H 03 X0 Y0 Z5;

N160 G 01 Z-25 F 50;

N165 G 04 X 3;

N170 G 00 Z 5;

N175 G 00 X 40 Y 40

N180 G 01 Z-25 F 50;

N182 G 04 X 3;

N190 G 00 Z 5;

N195 G 00 X - 40 Y 40;

N200 G 01 Z-25 F 50;

N205 G 04 X 3;

N210 G 00 Z 5;

N215 G 00 X-40 Y - 40;

N220 G 01 Z 25 F 50;

N225 G 04 X 3;

N230 G 00 Z 5;

N235 G 00 X 40 Y - 40;

N240 G 01 Z-25 F 50;

N245 G 04 X 3;

N250 G 00 Z 5;

N255 M 05;

N260 G 49;

N265 G 28 G 91 G 00 Y0 Y0 Z0;

N360 T 04 M 19 (Drill o 15)

N365 M 03 S 500;

N370 G 55 G 90 G 00 G 43 H 04 X 0 Y0 Z5;

N375 G 01 Z-25 F 50;

N380 G 00 Z 100;

N385 G 49;

N390 M 05;

N395 G 28 G 91 G 00 X0 Y0 Z0;

N400 M 30;

— — — — — — — —

TASK 3: **Enter the program in CNC simulator and verify it (Task 1 & Task 2)**

Programs can be created in the edit mode using the program editing fuctions

Creating programs using the MDI panel

Procedure creating programs using the MDI paned

Procedure

1. Enter the EDIT mode
2. Press the PRGRM key
3. Press address key O and enter the program number
4. Press the insert key

Explanations

Comments in a program

For the full key type MDI panel, comments can be written in program using the control in/out codes

Example O 0001 (Fanuc series 0)

M08 (Coolant ON);

- When the INSERT key is pressed after the control-out code ('comments, and control-in code') have been typed, the typed comments are registered.
- When the INSERT key is pressed midway through comments, to enter the rest of comments later, the data typed before the INSERT kye is pressed may not be correctly registered (not entered, modified or lost) because the date is subject to an entry check which is performed in normal editing

Note the following to enter a comment:

- Control-in code cannot) be registered by itself
- Comments entered after the key is pressed must not begin with a number, space, or address 0
- If an abbreviation for a macro is entered, the abbreviation is converted into a macro word and registered
- Address 'O' and subsequent number, or a space can be entered but are omitted when registered.

— — — — — — — —

Skill Sequence

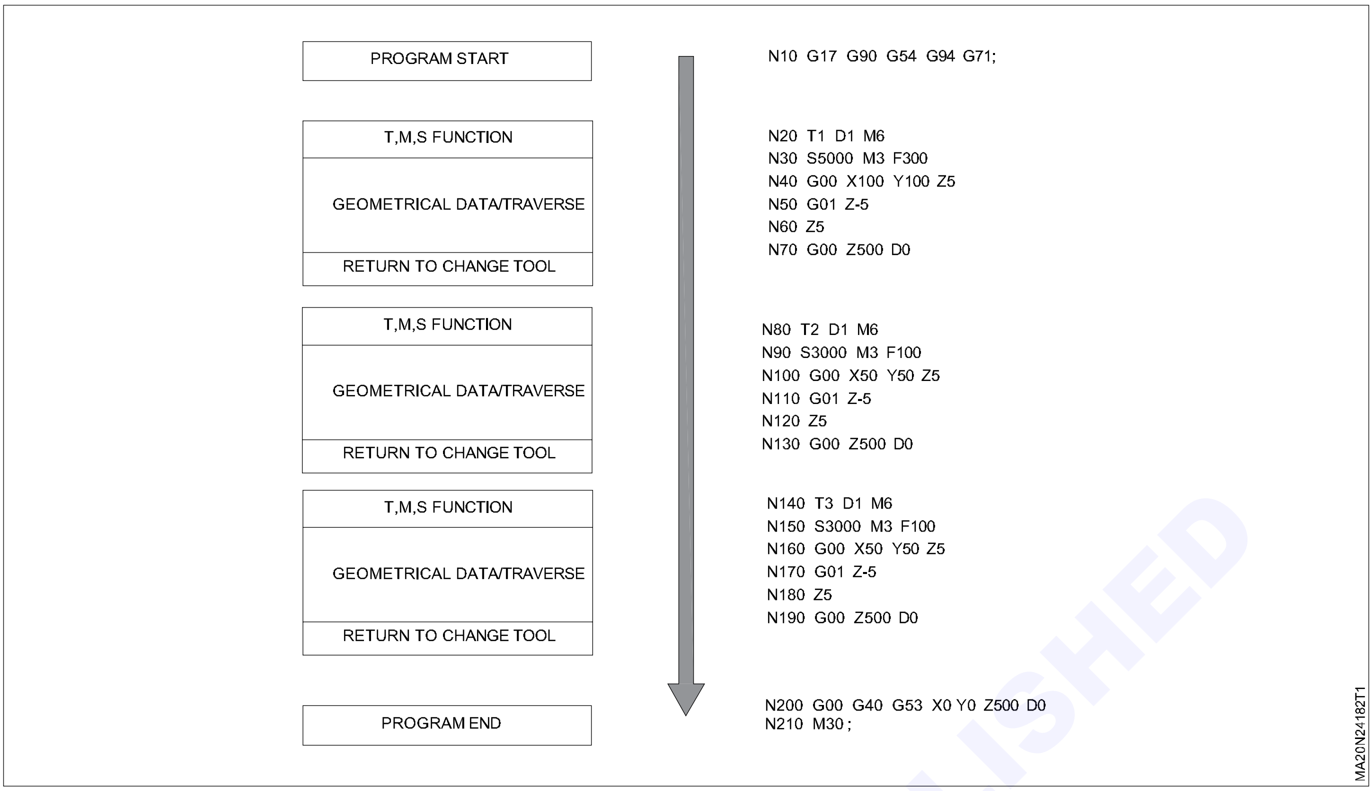
Editing part program (Siemens)

Objectives: This shall help you to
- **organise the standard part program**
- **edit the part program**
- **search, copy, paste and remember in part program.**

Editing the part program

Standard program structure

Standard program structure provides easy way of part programming and a clear view of the machining sequences. Siemens recommends that you use the following program structure.



Editing a part program

Methods for editing part programs

You can edit a part program with one of the following methods:

- Editing on a computer and transferring it to the PPU (Power Processing Unit) via USB interface
- Editing on a computer and transferring it to the PPU via Ethernet interface
- Editing it directly on the PPU

Editing a part program on the PPU

You can edit a part program only when it is not being executed

Note that any modification to the prat program in the program editior window is stored immediately

Note:

Steps 1 to 4: Search for a program file

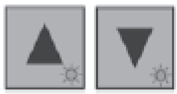
Steps 5 to 9: Edit the selected program in the open program editor window

1 Select the program management operating area.

2 Press this softkey to enter the system directory for storing part programs

3 Select the desired program file/directory in one of the following methods.

- Navigate to the program/directory with the cursor keys

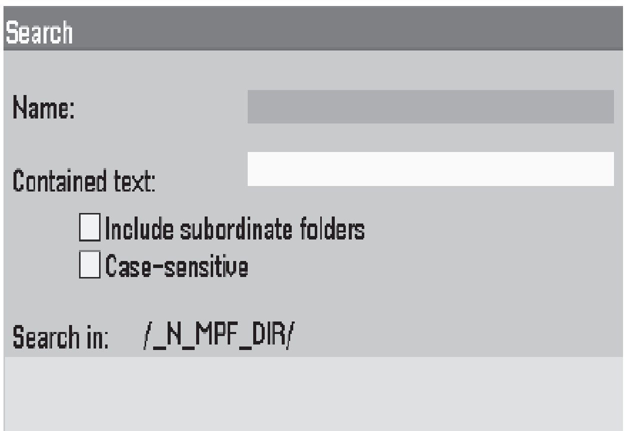


- Open the search dialog box and enter the desired search item

Note:

If you search for a program, the file name extension must be entered in the first input field of the dialog box (Fig 1)

Fig 1



- On the PPU, press the alphabetic or numeric key that contains the first character of the desired program/directory name. The control system automatically highlights the first program/directory whose name starts with that character. If necessary, press the key continuously until you find the desired program/directory

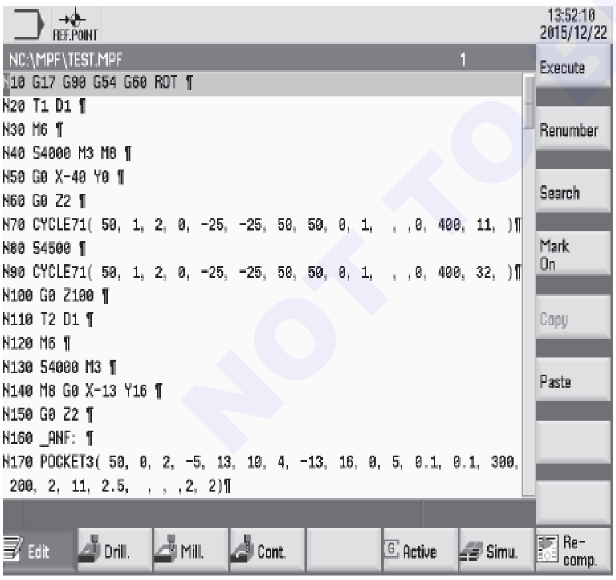
4 Press this key to open either the selected program in the program editor or the selected directory. In the latter case, perform Step 3 and then Step 4 until the selected program is opened in the program editor.

5 Edit the program text in the program editor window using the following keys on the PPU (Fig 2)

Fig 2

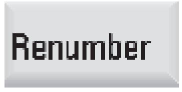
6 When necessary, select the following vertical softkeys to complete more program editing operations (Fig 3)

Fig 3



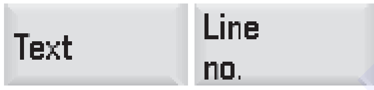
Renumbering program blocks

With this softkey, you can modify the block numbering (Nxx) of a program opened in the program editor window. After you press this softkey, the block number is inserted at the beginning of the program block in ascending order and is increased by an increment of 10 (for example, N10, N20, N30)



Searching in programs

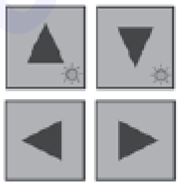
Pressing this softkey opens the search dialog box. You can use the search function to quickly arrive at points where you would like to make changes, for example, in very large programs. You can search with specified text or line number by selecting the corresponding softkey.



Copying/deleting/pasting program blocks

A Press this softkey in opened program editor window to insert a mark.

B Select the desired program blocks with the cursor



C Press this softkey to copy the selection to the buffer memory

- OR -

Press this key to delete the selected program blocks and to copy them into the buffer memory.

D Place the cursor on the desired insertion point in the program and press this softkey. The content of the buffer memory is pasted.

7 If you want to program cycles, press the corresponding softkey to open the desired cycle programming window.

8 If you want to program contours, press this softkey to open the contour programming window.

9 Press program manager key to return to the program management operating area after finishing editing.

Note

When you are in the main screen of the program management operating area, you can select to make the following operations between programs:

- Searching for programs
- Copying/cutting/pasting programs
- Deleting/restoring programs
- Renaming programs

Frequently used programming instructions

Inch or metric dimensions (G70/G71)

Description	Programming example
G71:	N10 G17 G90 G54 G 71
With G71 at the program start both the geometrical data and the feedrates are evaluated as metric units	N20 T1 D1 M6 N30 S5000 M3 G94 F300 N40 G00 X100 Y100 Z5 N50 G01 Z-5 N60 Z5 N70 G00 Z500 D0
G70:	
With G70 program start, the geometrical data is evaluated as inches, bu the feedrates are not affected and remain as metric units.	N10 G17 G90 G54 G 70 N20 T1 D1 M6 N30 S5000 M3 G94 F300 N40 G00 X3.93 Y3.93 Z5 N50 G01 Z-0.787 N60 Z0.196 N70 G00 Z19.68 D0

Maintaining tools on the ATC

Objectives : At the end of this exercise you shall be able to
- **identify the pockets where the appropriate tools are to be mounted**
- **fix the tool manually in the magazine**
- **fix the tool automatically in the MDI operation.**

Job Sequence

TASK 1: **Identify the pockets with respect to tool number in the program**

- Set the tool in the tool holder as per the requirement in the program
- Number the tool as per program
- Check the tool list in the magazine with respect to tool number and pocket position

  Press custom graphic - press magazine

The tool list will display

Example

Tool No	Pocket number
1	6
2	7
3	9
Etc.	

Note down the tool number and pocket number

— — — — — — — —

TASK 2: **Fixing tool manually in the magazine**

- Clean the tool holder and the pocket number with suitable cleaner
- Manual index the magazine and bring the required pocket to load the tool manually
- Remove tool in any in the pocket, using special spanner provided with the machine
- Insert the required tool in position with little pressure
- Check the tool is grippled properly in the pocket
- Similarly fix the other tool in the pockets

— — — — — — — —

TASK 3: **Fixing tool automatically in the MDI mode**

MDI - MODE-OPERATION

Press the MDI key

Press the " program" key

The following screen appear (Fig 1)

Fig 1

PROGRAM (MDI)
O0000
%

10:26:11 S M OT DI
PROGRAMMING NUMBER O0000 IS GENERATED AUTOMATICALLY

MA20N24183H1

- Prepare the program blocks with a block M99 at the end to return to the beginning the block
- To erase the program created in MID either press "Reset" key or enter address O0000 then press the "Delete" key in the MDI panel.
- Place cursor in the first blok and push cycle start key for executing position
- To stop the operation press "Reset" key (or) rotate the "Feed holed" key to "0" position (or) press the emergency switch.

Call the required tool number on the spindle

- Unclamp the tool by pressing unclamp tool switch, LED Glow
- Insert the required tool on to the spindle press clamp tool switch it will hold the tool on the to the spindle and the LED will put off
- Similarly fix the other tools
- After fixing all the tools, call the tools and check for the correct tool and its number.

— — — — — — — —

Measurement of work and tool offset (Fanuc)

Objectives : At the end of this exercise you shall be able to
- **measure the work offset in X and Y axis**
- **measure the work offset in Z axis**
- **measure the tool offsets**
- **enter the work offset**
- **enter the tool offset.**

Job Sequence

- A CNC machine tool is provided with a fixed position. Normally tool change and programming and absolute zero point as described. This position is called the reference position (Fig 1)
- Machine zero point is fixed some where in the left corner of the table as shown in Fig 1.
- Work place is fixed on the machine table. The distance between the machine zero to work zero (Fig 1) is the work zero point

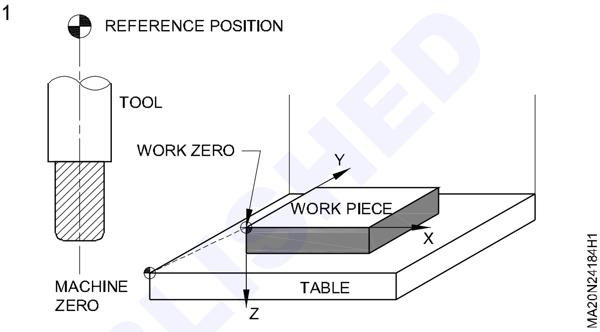


Fig 1

— — — — — — — —

TASK 1: **Measurement of work offset**

- Call tool number one in MDI mode.
- Go to jog mode. Remove the available tool if any/
- Mount one referencce tool / cylindrical pin of about 100mm length / position finder. Make the zero offset value G54 X0, Y0, Z0.
- Just touch the X surface of the workpiece as shown in Fig1 and take the absolute display reading of X-axis.
- Just touch th Y surface of the workpiece as shown in Fig 1 and take the absolute display reading of Y-axis.

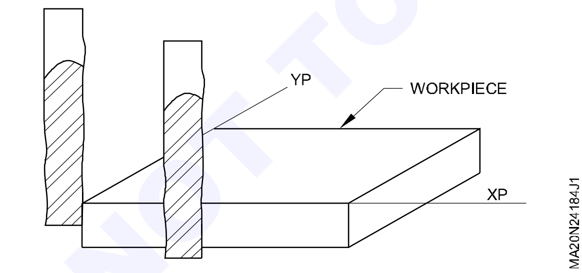


Fig 1

- Just touch the Z surface of the workpiece as shown in Fig 2 and take the absolute display reading of Z-axis.

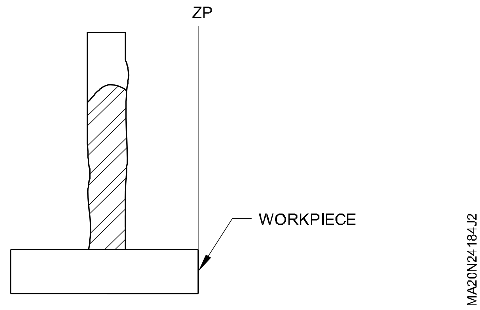


Fig 2

- Reduce the radius value of the tool / cylindrical pin from the noted X and Y value. This gives the zero offset of the corner of the job as shown. This may be entered under X and Y in one of the zero offset value i.e., in G55-G59.
- Enter the same absolute display value in 'Z'.
- Every time a new job is mounted, new 'Z' value should be taken through the reference tool.

  As explained above and must be entered in the appropriate zero offset number i.e., G55 to G59.

— — — — — — — —

TASK 2: **Tool offset measurement**

- **If tool measuring system is available we can measure the tool offset through the tool pre setter.**
- **If the tool pre setter is not available then reference tool method can be used to find out the tool offset.**

- Designate always. Tool No 01 as reference tool and avoid changing the tool.
- Bring this tool and just touch the machined surface of the job.
- Note down the absolute value of Z at the time when the G54 Z-value is '0' - say A

- Use this 'Z' value for subsequent measurement of tool - (A)
- Bring the next tool (No 02) for which measurement is to be set.
- Touch the tool on the same workpiece surface.
- Note the display value and subtract the ref. Tool 'Z' value (A).
- This will be the Tool offset value for Tool No 02.
- The same procedure can be adopted for other tools.
- Enter all the value of length offsets in the CNC, against the Tool No.,

The tool-offset can be measurent directly if the value A is entered against G54 Z value.

Tool Name, offset number along with the length offset may be kept seprarately & safely in a tool register for further use and reference.

- The value A should be taken as zero offset Z-value for G55-G59
- To set zero offset of 'Z' for a new job just touch the reference tool in 'Z' direction and note down the Z value in the required offset number from G55 to G59. Please note that G54, z-value is 0. This avoids measuring of Tool offset for other Tools again.

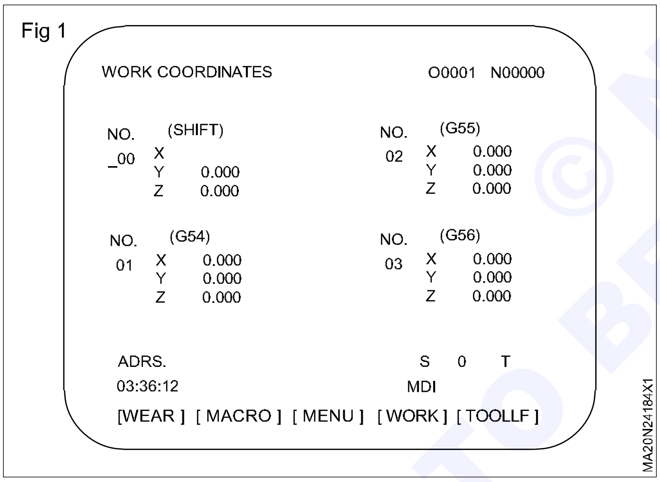
— — — — — — — —

TASK 3: **Enter work offset value**

Note: Displays the workpiece origin offset for each workpiece coordinate system (G54 to G59 P1 to G54) and external workpiece origin offset. The offset can be set on this screen.

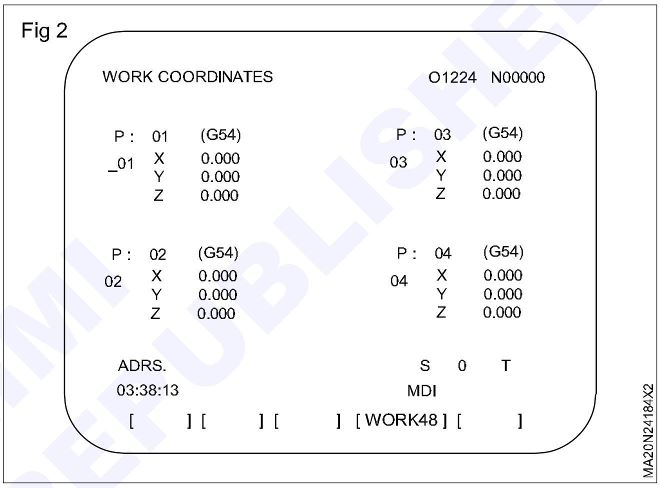
- Press function key [Menu offset]
- Press soft key [WORK] or [WORK48]
- The workpiece corordinate system setting screen is displayed in Fig 1& 2.

Fig 1



- The screen for displaying the workpiece origin offset values consist of two or more pages. Display a desired page in either of the following two ways
- Press the page up or page down key.

Fig 2



- Press NO key and enter the workpiece coordinate system number (0: external work piece origin offset, 1 to 6: workpiece coordinate systems G 54 to G59, 1 to 48: workpiece coordinate systems G54 P1 to G54 P48), and then press INPUT key
- Move the cursor to the work piece origin offset to be changed
- Press the address key corresponding to the desired axis
- Enter a desired value by pressing numeric keys, then press INPUT key.
- The entered value is specified in the workpiece origin offset value.
- To change other offset values move the cursor to the desired offset and enter the new value.

— — — — — — — —

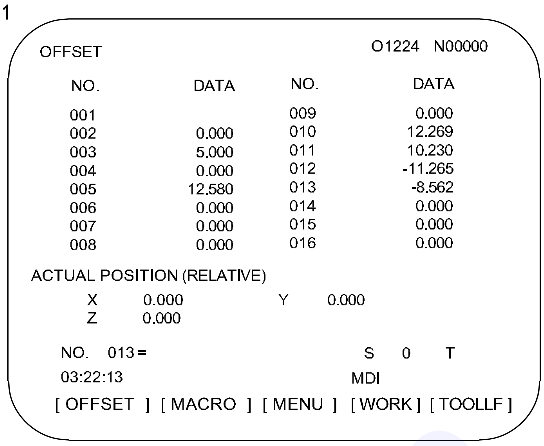
TASK 4: **Enter the tools offset values**

Tool offset values, tool length offset values, and cutter compensation values are specified by D codes or H codes in a program. Compensation values corresponding to D codes or H codes are displayed or set on the screen

- Press function key Menu Offset
- Press soft key [OFFSET] (Fig 1)

  (The screen varies according to the type of tool offset memory.)
- Move the cursor to the compensation value to be set or changed using page keys and cursor keys. Press NO key and enter the compensation number for the compensation value to be set or changed and then press INPUT key.
- Enter a compensation value, and INPUT key.

Fig 1



TOOL OFFSET MEMORY A

MA20N24184Y1

Explanations

- **Decimal point input** — A decimal point can be used when entering a compensation value.
- **Other method** — An external input/output device can be used to input or output a cutter compensation value.
- **Tool offset memory** — There are tool offset memories A, B, and C, which are classified as follows:

  Tool offset memory A

  Tool geometry compensation and tool wear compensation are treated the same.

  Tool offset memory B

  D codes and H codes are treated the same. Tool geometry compensation and tool wear compensation are treated differently

  Tool offset memory C

  Offset memory is divided into two areas: one area for tool length offset date (H), and the other for cutter compensation date (D)
- **Disabling entry of Compensation values** — The entry of compensation values may be disabled by setting bit 0 and 1 of parameter 078 (not applied to tool offset memory A)
- **Incremental input** — The current compensation value can be incremented or decremented if bit 4 of parameter 001 is specified accordingly. If this is selected, key in a desired increment or decrement.

Skill Sequence

Setting up the work piece

Objectives: This shall help you to
- **measure the work offset for work piece edge**
- **measure the work offset for rectangular work piece**
- **measure the work offset for circular work piece**
- **enter the and edit the offset values.**

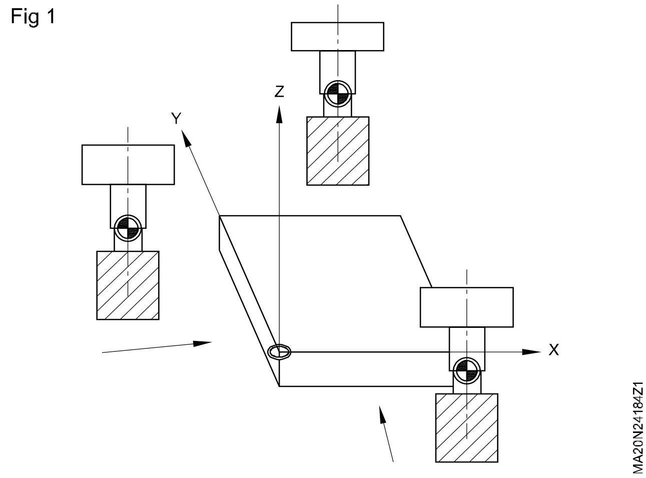
Measuring the workpiece

Overview

You must select the relevant offset panel (for example, G54) and the axis you want to determine for the offset first (Fig 1)

Before measuring you can start the spindle by following the steps in Section " Activating the tool and the spindle"

Fig 1



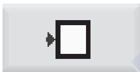
Operating sequence

Work edge measurement

1 Select the machining operating area 

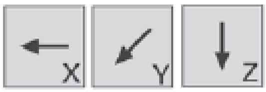
2 Switch to "JOG" mode 

3 Open the window for workpiece measurement 

4 Press the vertical softkey to open the window for measurement at the workpiece edge. 

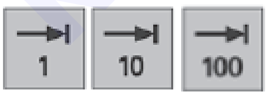
5 Press this softkey to measure in the X direction 

6 Traverse the tool, which has been measured previously to approach the workpiece in the X direction

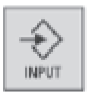


7 Switch to handwheel control mode. 

8 Select a suitable override feed rate and then use the handwheel to move the tool to scratch the required workpiece edge.

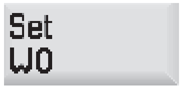


9 Select the offset plane to save in and the measuring direction (for example, "G54" and " ") 

10 Enter the distance (for example "0") in the following window 

Press this key or move the cursor to confirm your print Fig 2

Fig 2

11 Press this vertical softkey. The workpiece offset of the X axis is calculated automatically and displayed in the offset field 

12 Repeat the above operations to measure and set the workpiece offsets in axis Y and Z respectively

Rectangular workpiece measurement

1 Select the machining operating area 

2 Switch to "JOG" mode. 

3 Open the lower-level menu for workpiec e measurement. 

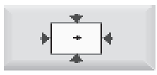
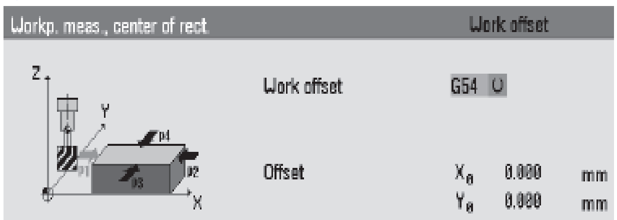
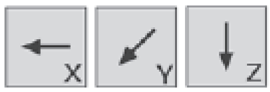
4 Open the window for measurement of a rectangular workpiece (Fig 3) 

Fig 3



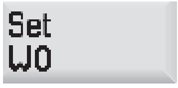
5 Traverse the tool, which has been measured previously, in the direction of the orange arrow P1 shown in the measuring window, in order to scratch the workpiece edge with the tool tip.



6 Save the tool position P1 in the coordinate system.

7 Repeat steps 5 and 6 to save the other three positions P2, P3 and P4.

8 Save the workpiece offsets in axes X and Y after measuring all four positions

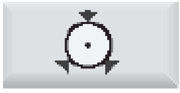


Circular workpiece measurement

1 Select the machining operating area 

2 Switch "JOG" mode. 

3 Open the lower-level menu for workpiece measurement

4 Open the window for measurement of a circular workpiece. 

5 Traverse the tool, whicch has been measured previously, in the direction of the orange arrow P1 shown onn the measuring window, in order scratch the workpiece edge with the tool tip (Fig 4)

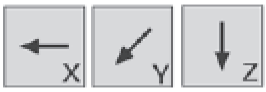
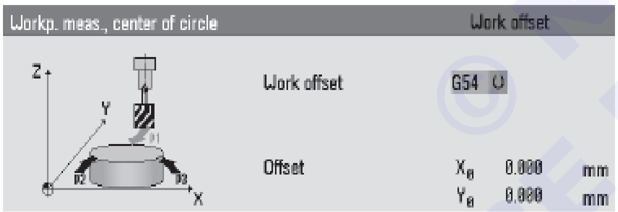
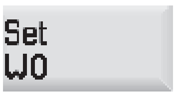


Fig 4



6 Save the tool position P1 in the coordinate system

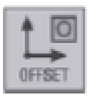
7 Repeat steps 5 and 6 to save the other two positions: P2 and P3

8 Save the workplace offsets in axes X and Y after measuring all three positions. 

Entering/modifying workpiece offsets

Operating sequence

In case of any problems found when testing the tool offset result, you can proceed through the following steps to make tiny adjustment values.

1 Select the offset operating area. 

2 Open the list of workpiece offsets. The list contains the values of the basic offset of the programmed workpiece offset and the active scaling factors, the mirror status display the total of all active wokpiece offsets.

3 Use the cursor keys to position the cursor bar in the input fields to be modified and enter the values (Fig 5)

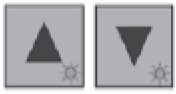


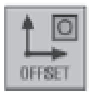
Fig 5

	X mm	Y mm	Z mm	X	Y	Z
G500	0.000000	0.000000	0.000000	0.000	0.000	0.000
G54	0.000000	0.000000	0.000000	0.000	0.000	0.000
G55	0.000000	0.000000	0.000000	0.000	0.000	0.000
G56	0.000000	0.000000	0.000000	0.000	0.000	0.000
G57	0.000000	0.000000	0.000000	0.000	0.000	0.000
G58	0.000000	0.000000	0.000000	0.000	0.000	0.000
G59	0.000000	0.000000	0.000000	0.000	0.000	0.000
Program	0.000000	0.000000	0.000000	0.000	0.000	0.000
Scale	1.000000	1.000000	1.000000			
Mirror	0	0	0			
Total	0.000000	0.000000	0.000000	0.000	0.000	0.000

4 Confirm your entries. The changes to the workpiece offsets are activated immediately 

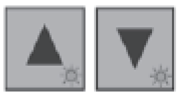
Entering/modifying the setting data

Operating sequence

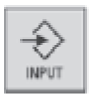
1 Select the offset operating area 

2 Open the setting data window 

3 Open the window for setting the general data

4 Position the cursor bar in the input fields to be modified and enter the values. 

5 Use this key or move the cursor to confirm your entries.



Tool change in the CNC VMC

Objectives : At the end of this exercise you shall be able to
- **change tool in machine spindle manually**
- **change tool using tool change command in MDI mode.**

Job Sequence

Normally any machining operation last operation tool will be in the spindle.

TASK 1: **Change the tool in machine spindle**

- Switch on the machine
- Perform the reference operation
- The LED bulb above the button will glow

Normally reference point and tool change position will be the same if not move the spindle position to tool change point.

- Hold the tool in left hand
- Press the tool release button (unclamp tool) and it will release the tool from the spindle

Careful not to allow the tool to fall down - Hold it carefully.

- Insert the required tool in to the spindle by aligning slot position into the tang position and then press the tool clamps button. The LED bulb goes off.
- Run the spindle in MDI mode and check the true running of the tool on the spindle.

— — — — — — — —

TASK 2: **Tool change in MDI mode**

- Start the machine
- Reference the machine
- Locate the tool and pocket number that you want to call the tool into the spindle
- Identify the tool number by viewing custom graph → magazine. It will show the list

Example

Tool Number	Pocket number in Magazine
1	3
2	6
3	10
4	12
6	6

Example Tool number is in pocket 6

- Go to MDI mode, type command as T02, input into MDI screen and press cycle start button

  Now the spindle tool will go to pocket number 6, and the tool in pocket 6 that is tool number 2 will be fixed on to the machine spindle

— — — — — — — —

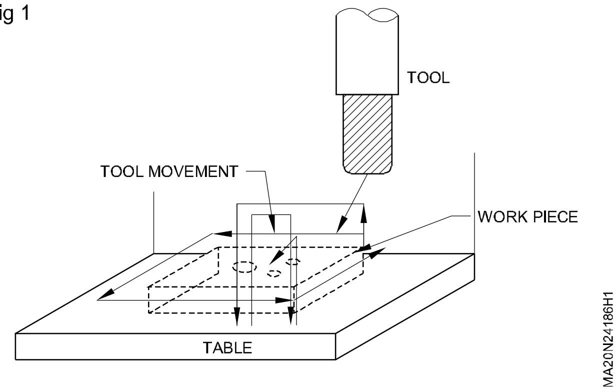
Program checking in dry run, and single block modes

Objectives : At the end of this exercise you shall be able to
- **verify the CNC program by dry run mode**
- **verify the CNC program by single block mode.**

Job Sequence

TASK 1: **Verify the program in dry run mode (Fig 1)**

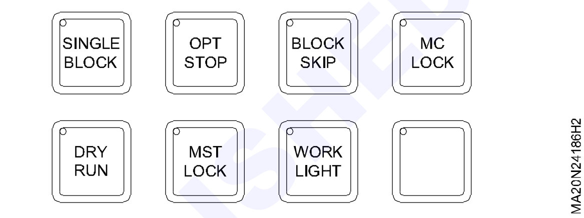
Fig 1



Note: The tool is moved at the federate specified by a parameter regardless of the federate specified in the program. This function is used for checking the movement of the tool under the state that the workpiece is removed from the table.

- Press dry run switch on the machine operator's panel during automatic operation (Fig 2)

Fig 2



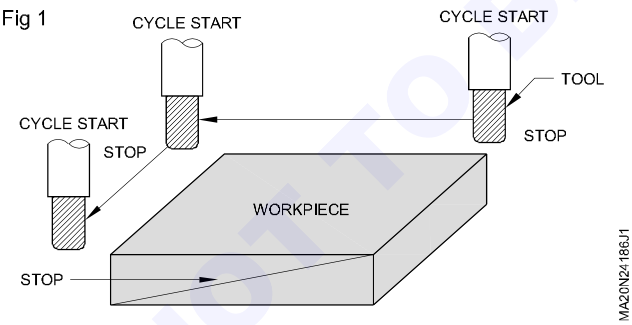
The tool moves at the federate specified in a parameter. The rapid traverse switch can also be used for changing the federate.

Refer to the appropriate manual provided by the machine tool builder for dry run.

— — — — — — — —

TASK 2: **Verify program by single block (Fig 1)**

Fig 1



Note: Pressing the single block switch starts the single block mode. When the cycle start button is pressed in the single block mode, the tool stops after a single block in the program executed. Check the program in the single block mode by executing the program block by block

- Press the single block switch on the machine operator's panel, during automode/dry run mode. The execution of the program is stopped after the current block is executed.
- Press the cycle start button to execute the next block. The tool stops after the block is executed.

Note: Refer to the appropriate manual provided by the machine tool builder for single block execution.

— — — — — — — —

Checking finished size through tool offset

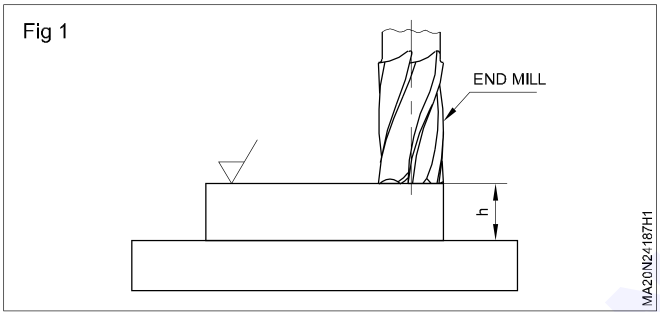
Objectives : At the end of this exercise you shall be able to
- **check the finished size by micrometer/Vernier caliper**
- **determine size whether + or - to the drawing dimensions**
- **adjust + or - in tool offset depending upon the position.**

Job Sequence

- When a programmed is executed, it is quite possible that the actual dimension achieved may be different from desired value and it may be due to error in tool offset measurements. This error may be corrected as described below in the following cases.

CASE 1

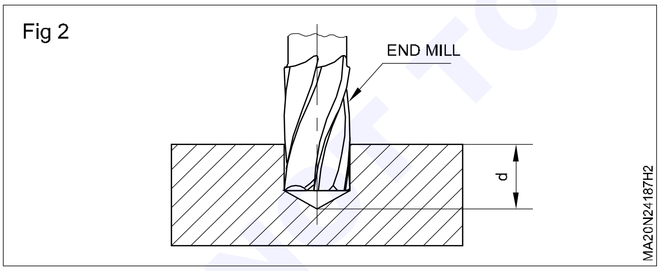
Height dimension (h) Fig 1)



- If 'h' is more than the desired value reduce the tool offset by the difference of measured value and desired value
- If 'h' is less than the desired value increase the tool offset (length) by the difference between the desired value and measured value.

CASE 2

Depth dimension (d) (Fig 2)

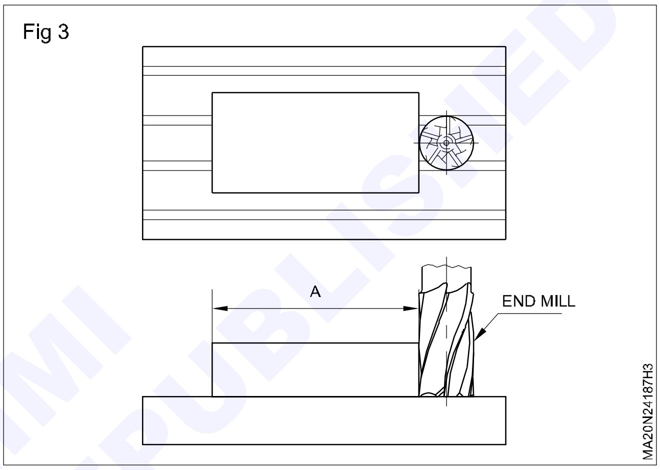


- If 'd' is more than the desired value increase the tool offset by the difference of measured depth and desired length
- If 'd' is less than the desired value reduce the tool offset by the difference between the desired value and measured value.

CASE 3

Dimension of external profile

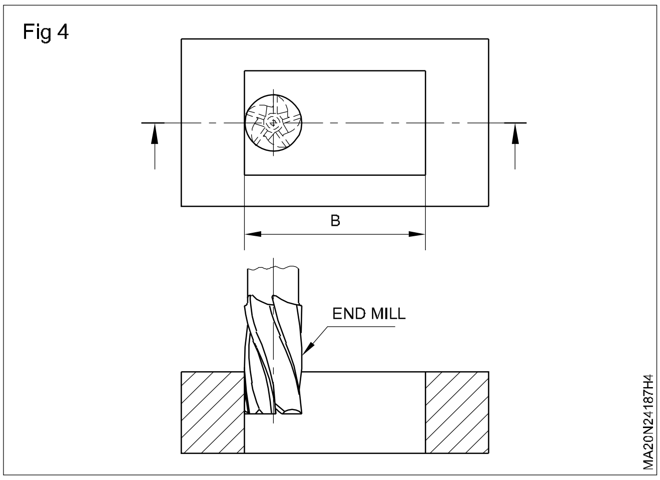
- If A (fig 3) is more than the desired value reduce the tool offset (radius) by half the difference between measured value and desired value
- If A is less than the desired value increase the tool offset (radius) by half the difference between the desired value and measured value



CASE 4

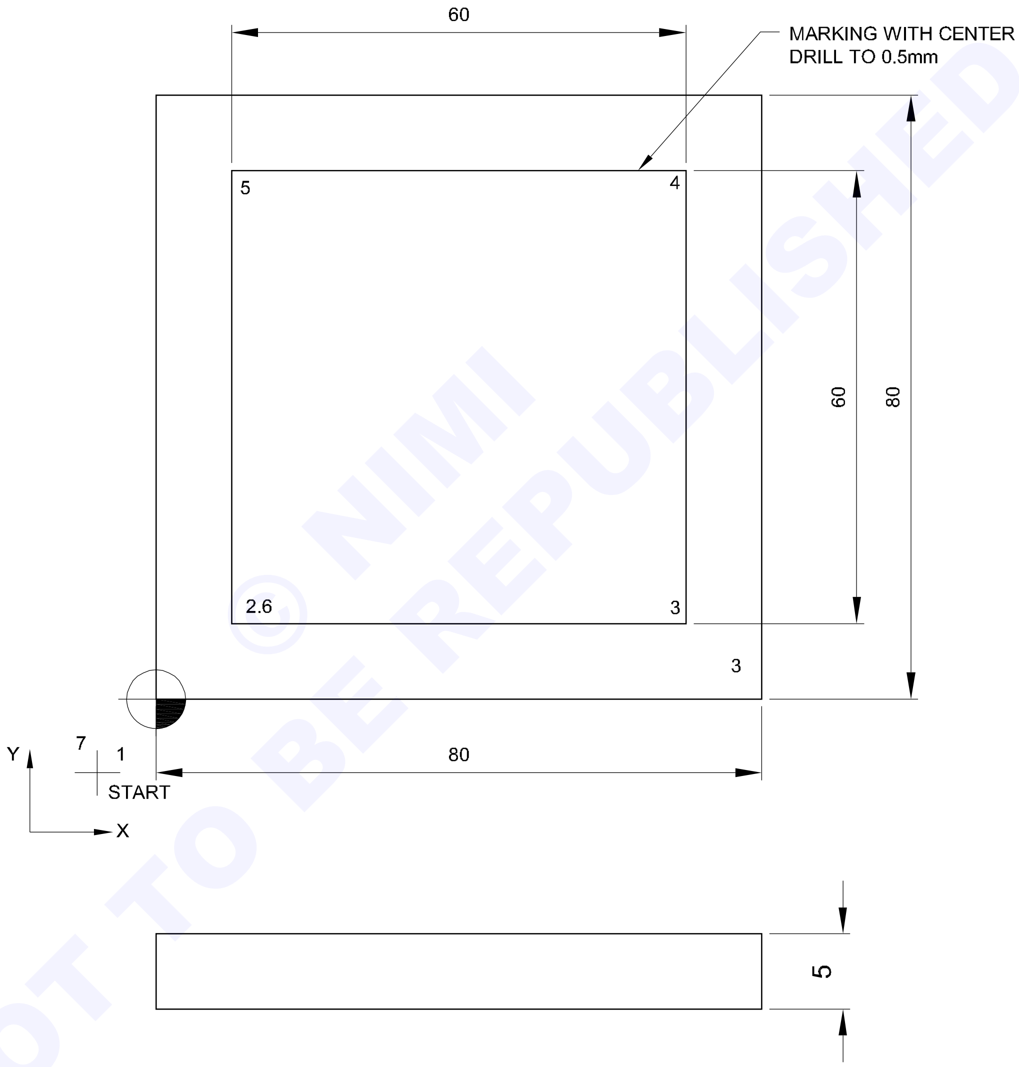
Dimension of internal profile

- If B is more than the desired value increase the offset by half the difference between the measured value and desired value. (Fig 4)
- If B is less than the desired value reduce the tool of (radius) by half the difference between desired value and measured value.



Prepare part program, enter, edit and simulate

Objectives : At the end of this exercise you shall be able to
- **write part programming marking to 0.5 mm depth**
- **input the program in the simulator and edit it**
- **execute the program in the simulator.**



ALL DIMENSIONS ARE IN mm

-	-	-	-	-	-	2.4.188
NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.

SCALE 1:1	**PART PROGRAMMING AND PRACTICE WITH SIMULATOR (POINT TO POINT) SINUMERIC**	DEVIATIONS ±1	TIME :
		CODE NO. MA20N24188E1	

Job Sequence

- Write the CNC program for marking to a depth of 0.5 mm
- Enter the program in CNC simulator
- Verify the program by simulation

Program (Fanuc)

O 0204

N5 G40 G49 G50 G80 G69;

N10 G90 G21 G94;

N15 T04 M19, (center drill or pointed tool)

N20 S600 M03

N25 G00 G55 G43 H04 X0 Y0 Z2;

N30 G00 X 10 Y10;

N35 G01 Z-0.5 F100;

N40 G01 X70 Y10;

NN45 G01 X 70 Y70;

N50 G01 X10 Y10

N55 G01 X10 Y10

N60 G00 Z100;

N65 M05;

N70 G00 G91 G28 X0 Y0 Z0;

N75 M30;

Note: Enter the program in CNC simulator and verify it by your trainer

Skill Sequence

Editing programs

Objectives: This shall help you to
- **inserting, altering, and deleting a word**
- **replacing words and addresses**
- **deleting blocks.**

Editing → search for part program to be edited

- Select EDIT mode
- Press function PRGRM key and display the program screen.
- Select a program to be edited
- If a program to be edited is selected, perform the operation
- Search for a word to be modified
  - Scan method
  - Word search method
- Perform an operation such as altering, inserting, or deleting a word

Word search

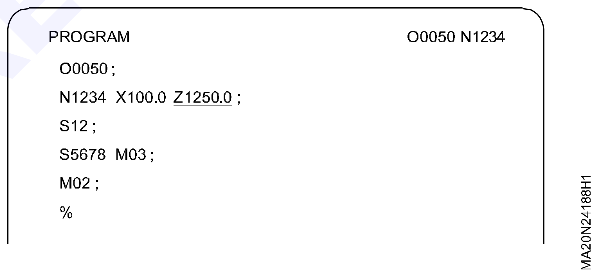
- Press the cursor key ↓

  The cursor moves forward word by word on the screen; the cursor is displayed at a selected word. The cursor is positioned to the address of the selected word.
- Press the cursor key ↑

  The cursor moves backward word by word on the screen; the cursor is displayed at a selected word.

Example: When Z1250.0 is scanned (Fig 1)

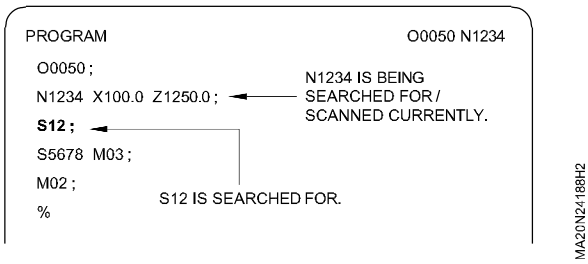
Fig 1



- Holding down the cursor ↓ key or ↑ scans words continuously.
- Pressing the page key ↓ displays the next page and searches for the first word of the page
- Pressing the page key ↑ displays the previous page and searches for the first word of the page.
- Holding down the page key ↓ or ↑ displays one page after another

Example of searching for S12 (Fig 2)

Fig 2

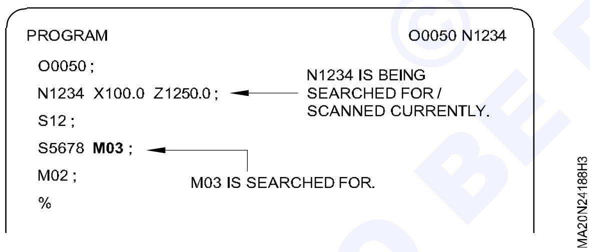


- Key in address [S]
- Key in [1] [2]
  - S12 cannot be searched for if only S1 is keyed in.
  - S09 cannot be searched for by keying in only S9.
  - To search for S09, be sure to key in S09.
- Pressing the cursor key [↓] starts search operation.
- Upon completion of search operation, the cursor is displayed at "S" of S12. Pressing the cursor key [↑] rather the cursor [↓] key performs search operation in the reverse direction

Searching an address

Example of searching for M03 (Fig 3)

Fig 3



- Key in address [M]
- Press the cursor key … [↓]
- Upon completion of search operation, the cursor is displayed at "M" M03. Pressing the [↑] Key rather than the [↓] key performs search operation in the reverse direction

Heading a program

Method 1

1 Press [RESET] when the program screen is selected in EDIT mode.

When the cursor has returned to the start of the program, the contents of the program are displayed from its start on the screen.

Method 2

1 Select AUTO or **EDIT** mode.

2 Press function **PROG** key and display the program

3 Press the address key [O]

4 Press the cursor key… [↑]

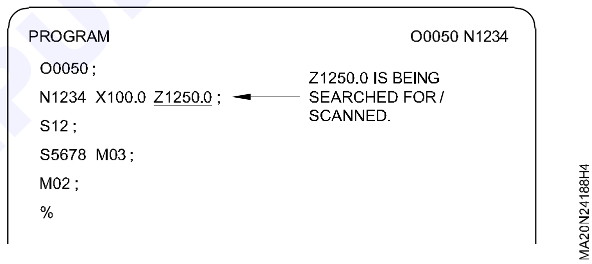
Inserting a word

- Search for or scan the word immediately before a word to be inserted
- Key in an address to be inserted.
- Key in data
- Press the [INSRT] key.

Example of inserting T15

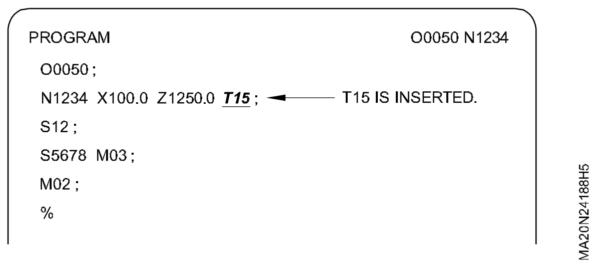
- Search for or scan Z1250.0 (Fig 4)

Fig 4



- Key in [T] [1] [5]
- Press the [INSRT] key (Fig 5)

Fig 5



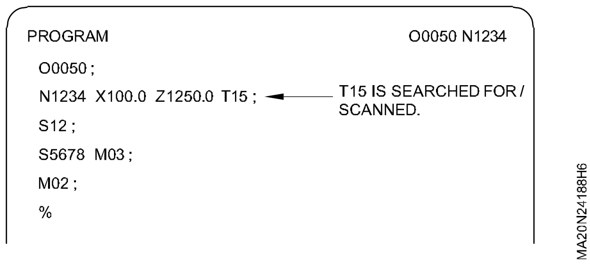
Altering a word

- Search for or scan a word to be altered
- Key in an address to be inserted
- Key in data.
- Press [ALTER] key

Example of changing T15 to M15

- Search for or scan T15 (Fig 6)

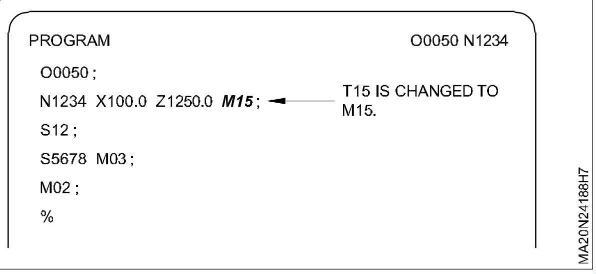
Fig 6



- Key in M 1 5

- Press the ALTER key. (FIg 7)

Fig 7



Deleting a word

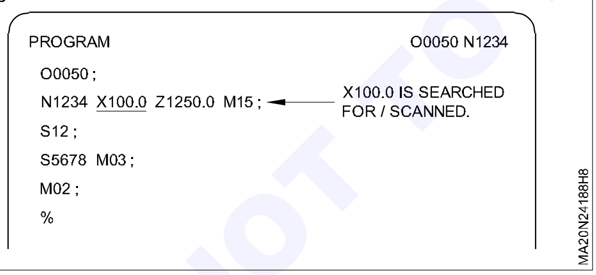
- Search for or scan a word to be deleted

- Press the DELET key

Example of deleting a word

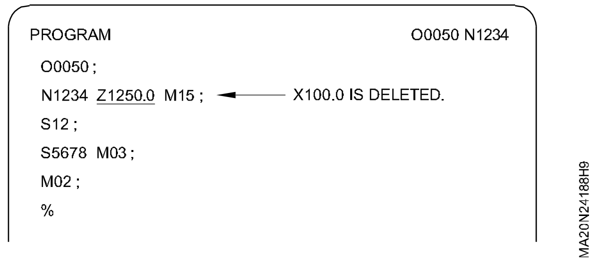
- Search for or scan X100.0 (Fig 8)

Fig 8



- Press the DELET key (Fig 9)

Fig 9



Deleting a block

- Search for or scan address N for a blok to be deleted

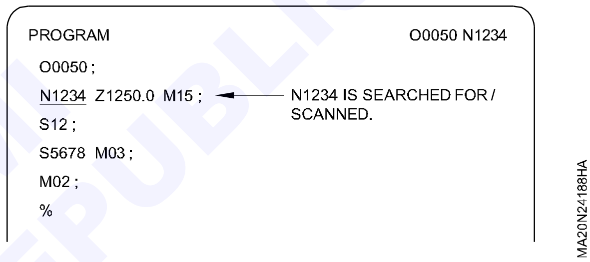
- Key in EOB

- Press the EOB key

Example of deleting a block No.1234

- Search for or scan N1234 (Fig 10)

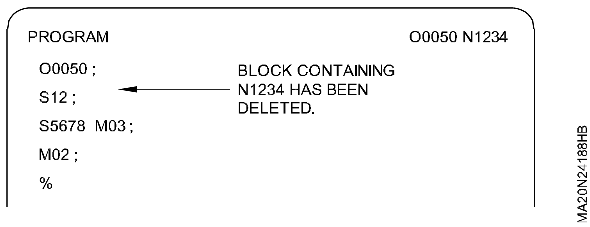
Fig 10



- Key in EOB

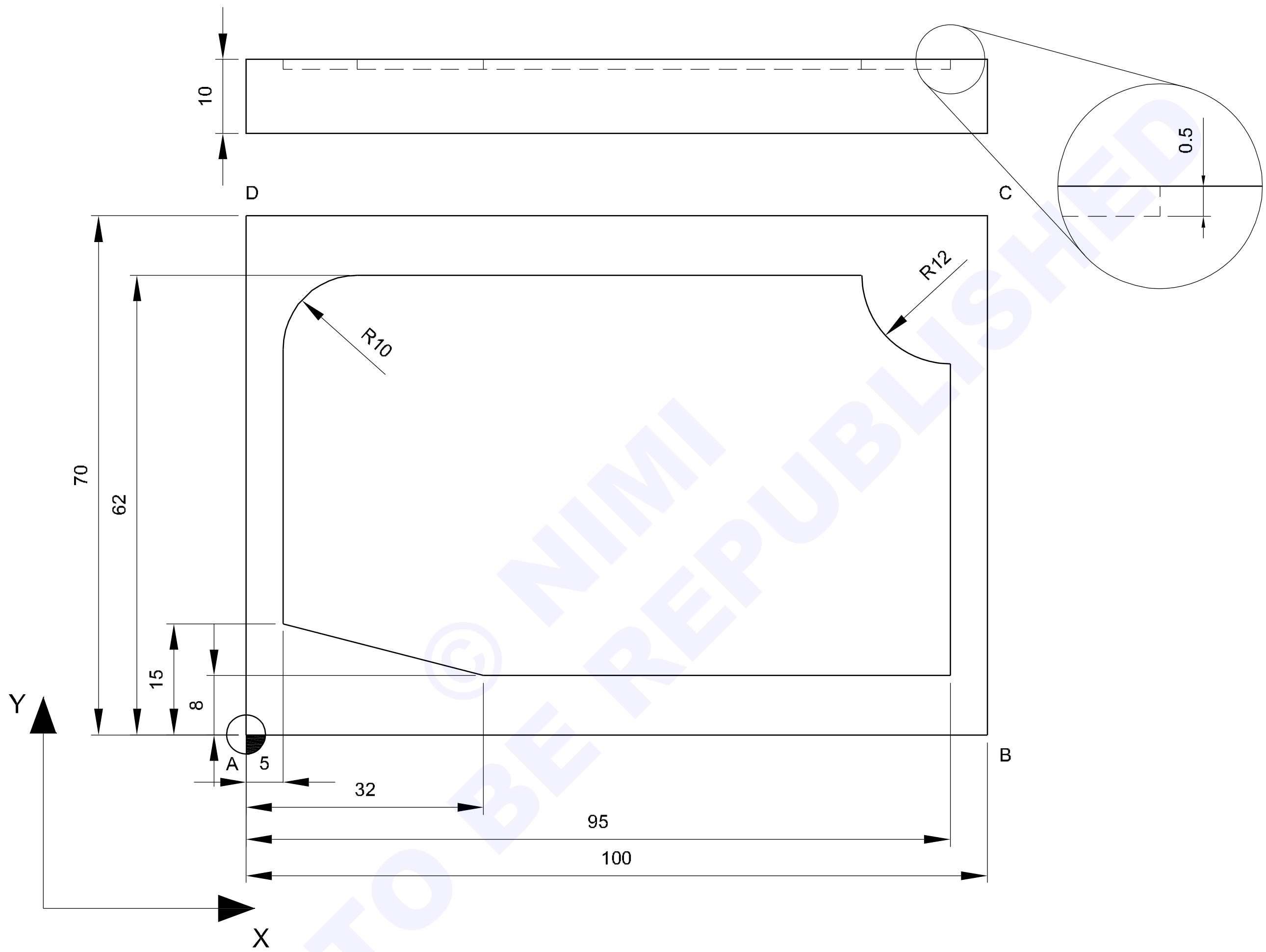
- Press the EOB key (Fig 11)

Fig 11



Tool path simulation

Objectives : At the end of this exercise you shall be able to
- **write the tool path for marking profile to 0.5 mm depth**
- **input the program in the simulator**
- **execute the program in the simulator.**



Job Sequence

- Write the tool path using linear and circular interpolation
- Enter the tool path in CNC simulator
- Edit and simulate the tool path
- Get is verified by your trainer.

ALL DIMENSIONA ARE IN mm

-	-	-	-	-	-	2.4.189
NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.

SCALE 1:1	**TOOL PATH SIMULATION**	DEVIATIONS ±1	TIME :
		CODE NO. MA20N24189E1	

Recovering from axis over travel

Objectives : At the end of this exercise you shall be able to
- **identify the over travel axis**
- **release the over travel by following the machine builders recommendation.**

Job Sequence

- Over travel: When the tool tries to move beyond the stroke end set by the machine tool limit switch, the tool decelerates and stops because of working the limit switch and over travel displayed.

Over travel during automatic operation

When the tool touches a limit switch along an axis during automode operation, the tool is decelerated and stopped along all axes and an over travel alarm is displayed.

Over travel during manual operation.

In manual operation, the tool is decelerated and stopped only along the axis for which the tool has touched a limit switch.

Releasing over travel

Press the reset button to reset the alarm after moving the tool to the safety direction by manual operation

For detail on operation, refer to the operator's manual of the machine builder.

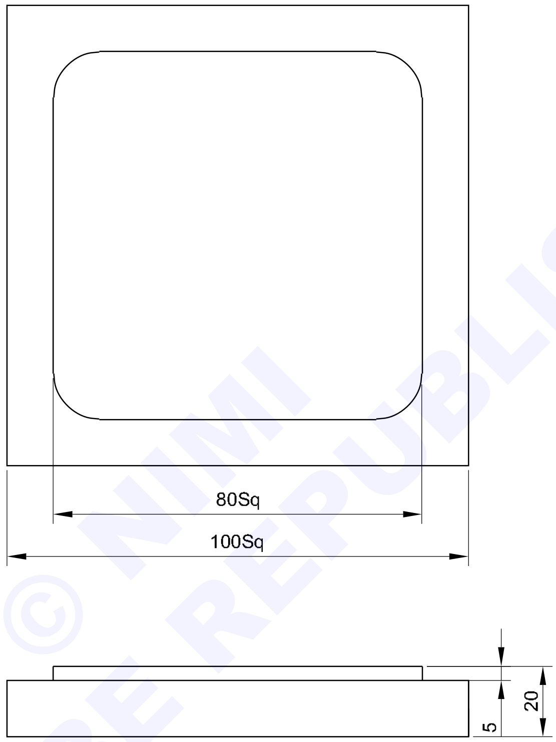
Procedure to disable this limit switches

- Find which axis is over travelled by observing the alarms
- Turn off the control
- Press (P) and the (cancel) key simultaneously in the keyboard
- Do not release these keys and turn on the control
- Hold the keys until the control turn on completely
- Release the keys
- Try to move the axis in Jog mode in the opposite direction of respective axis minimum of 25 mm
- Then do the homing.

In latest contract there is a O.T button (Over travel key) Press the O.T key and move the slide away from the limit switch. Then perform homing operation.

Side milling with CRC

Objectives : At the end of this exercise you shall be able to
- **write the CNC part program for side milling with CRC**
- **set the work piece tool and work offset on the machine**
- **enter the program in CNC VMC**
- **run the program in Automatic mode.**



Job Sequence

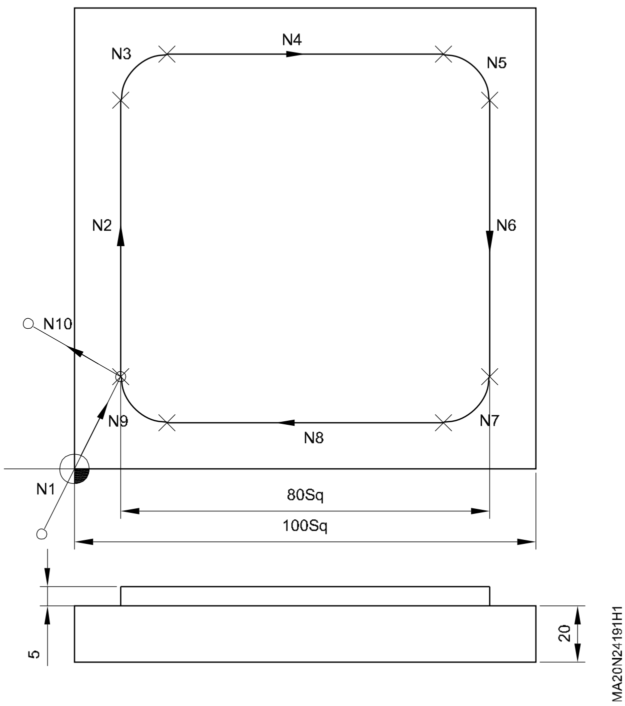
- Write the part program for 80 mm rectangular milling to a depth of 5 mm
- Enter the program in CNC simulator and verify it by simulating
- Set the work piece on the VMC machine table
- Set the work offset and enter into the work offset area
- Set 20 mm end mill on the tool magazine
- Call the tool in to the spindle by MDI mode
- Measure the tool offset and enter the tool offset and end mill radius in appropriate parameter
- Enter the program in CNC machine verify the program by dry run or shifting 'Z' work offset value
- Run the program in auto mode.
- Check the dimensions.

-	PRE MACHINED 110x100x20	-	ALUMINIUM		-	2.4.191
NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.
SCALE	SIDE MILLING WITH CRC				DEVIATIONS	TIME :
					CODE NO. MA20N24191E1	

Program for rectangle drilling with CRC (Fig 1)

Note: N indicate the line number in the programme and its movement. (Fig 1)

Fig 1



O0002 (rectangle milling)

N1 G40 G49 G80 G69;

N2 G90 G55;

N3 T05 M90;

N4 G94 S750 M03;

N5 G55 G43 X-50 Y-50 Z50M07 H05;

N6 G01 Z-5 F150;

N7 G41 G01 X 10 Y20 F100 D04

N8G01 Y80

N9 G02 X 0 Y90 R10

N10 G01 X80;

N11 G02 X90 Y80 R10

N12 G01 Y20 F100

N13 G02 X80 Y10 R10 F150

N14 G01 X 20;

N15 G02 X 10 Y20 R10 G01Z5;

N16 G01 ZG;

N17 G40 G0 X-50 Y40 F300

N18 G0Z100 M09;

N19 G00 G28X0 Y0 Z0M05 M30

N20 M30;

Skill Sequence

Automatic mode operation/memory operation

Objectives: This shall help you to
- **load the program to execcute in auto mode operation**
- **execute the program in auto mode.**

Memory operation

Programs are registered in memory in advance. When one of these programs is selected and the cycle stat switch on the machine operator's panel is pressed, automatic operation starts, and the cycle start lamp goes on.

When the feed hold switch on the machine operator's panel is pressed during automatic operation, automatic operation is stopped temporarily.

When the cycle start switch is pressed again, automatic operation is restarted.

When the reset switch on the CRT/MDI panel is pressed, automatic operation terminates and the reset state is entered.

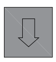
The following procedure is given as an example. For actual operation refer to the manual supplied by the machine tool builder

Automatic operation

- Press "Auto" mode switch
- Select the program number required. To select the particular program number.

  A Press" program" key to display the program.

  B Press address "O" and the enter the program number using numerical keys.

  C Press this  Curser key. Now the selected program will appear on the screen.
- Press cycle start switch to start the program.

  A By pressing Single block key the program will run in single block

B By pressing 

key the optional stop MDI in activated.

C By pressing 

Simulation of the program may be seen by releasing the feed hold.

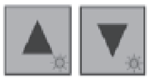
Executing a part program in auto mode(Siemens)

Objective: This shall help you to
- **executing a part program in auto mode (Siemens).**

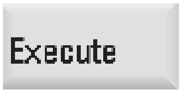
Executing a part program

Before starting a program, make sure that both the control system and the machine are set up, and the part program is verified with simulation and test. Observe the relevant safety notes of the machine manufacturer.

1 Select the program management operating area.

2 Select the desired program directory. 

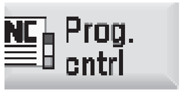
3 Select the program that you desire to execute.



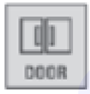
4 Press this soft key. For some directions, press the following softket instead. 

The system automatically changes the "AUTO" mode in the machining operation area after you press the softkey.

5 If desired, you can use this softkey to specify how you want the program to be executed (for more information, see section "Program control functions.)



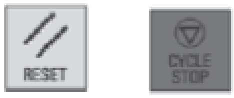
6 Make sure the federate override is 0%. Press this key on the MCP to close the safety door if this function ios not available, close the door on the machine manually.



7 Press this key on the machine control panel to start the machining of the program. 

8 Turn the feed rate override switch slowly to the desired value

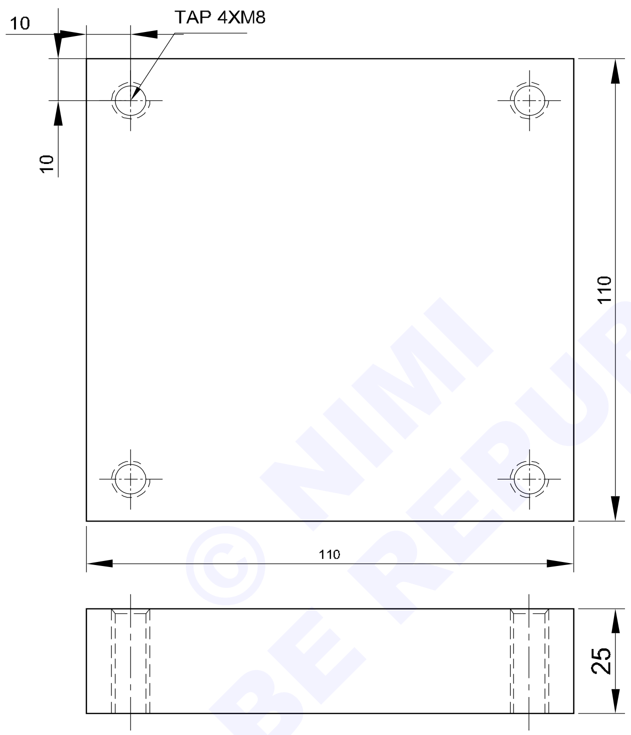
Pressing this stops the execution of a part program. The program currently running is aborted. On the next program start, the machining starts from the beginning.



Pressing this key suspends the execution of a part program. The axes stop running while the spindle continues running. Press the following key again, and the program continues to run. 

Face milling and canned cycle program

Objectives : At the end of this exercise you shall be able to
- **set work offset and tool offset**
- **write the program for face milling operation**
- **write the program for milling and tapping operation**
- **enter the program in CNC VMC and simulator and verify**
- **run the program in auto mode operation.**



Job Sequence

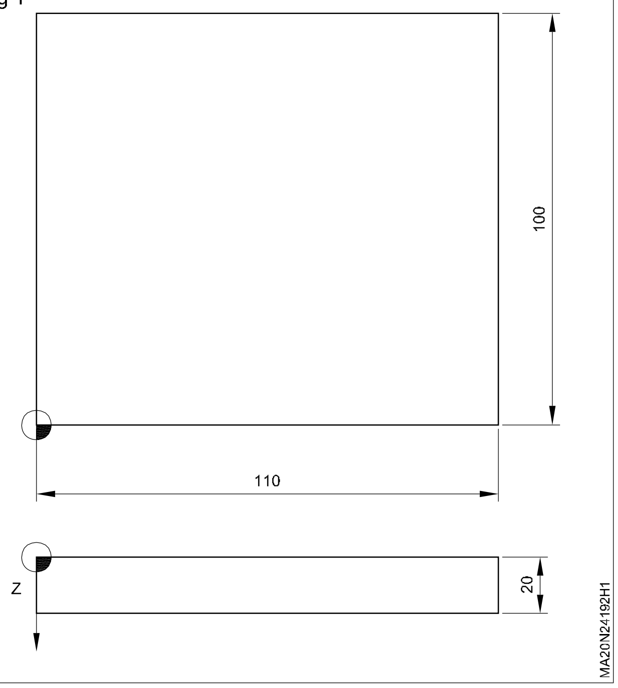
- Write the face milling program, enter CNC simulator and verify it.
- Prepare the program of centre drilling, drilling, counter sinking and tapping operation using canned cycles at specified locations in the drawing.
- Enter the program in CNC simulator and verify it.
- Fix the work piece on the table using suitable work holding device
- Set the work offset and tool offset for all the tools. Example Face mill, centre drill, drill, counter sink and tap
- Enter the both the program in the CNC VMC
- Check the program by dry run
- Execute the program in automode
- Check the dimension

Note: While taking work offset in Z axis shift the Z value 0.5mm for facing allowance.

-	PRE MACHINED 110x100x20	-	ALUMINIUM		-	2.4.192
NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.
SCALE	FACE MILL & DRILLING OPERATION				DEVIATIONS	TIME :
					CODE NO. MA20N24192E1	

Task 1 - Face milling program (Fig 1)

Fig 1



O0001

N5 G40 G49 G50 G80 G69;

N10 G90 G21 G94;

N15 T06 M19;

N20 S600 M03;

N25 G00 G55 G43 H06 X20 Y-30 Z50;

N30 G00 Z0 F100 M07;

N35 G01 X20 Y130

N42 G00 Z5;

N40 G00 X65 Y-30;

N45 G00 Z0;

N50 G01 X65 Y130;

N60 G00 X 110 Y-30;

N65 G00 Z0;

N70 G01 X 110 Y 130

N 75 G00 Z50 M05;

N80 G91 G28 X0 Y0 Z0 M09;

N85 M30;

Task 2 - Canned cycle program

(center drilling, drilling counter sinking and tapping operation)

O0002;

N1 G40 G15 G69 G80:

N2 G90 G21 G94:

N3 T06 M06; (CENTER DRILL)

N4 S600 M03;

N5 G00 G55 G43 H06 X 10 Y10 Z10;

N6 G81 X10 Y10 Z-6 T2 F50;

N7 X100 Y10;

N8 X100 Y100;

N9 X10 Y100;

N10 G00 G80 Z100;

N11 M05;

N12 G28 G91 X0 Y0 Z0;

N13 G90;

N14 T07 M06; (DRILLS DIA 6.8)

N15 S500 M03;

N16 G00 G55 G43 H07 X10 Y10 Z10;

N17 G83 X10 Y10 Z-25 Q10 R2 F50;

N18X100 Y10;

N19 X100 Y100;

N20 X10 Y100

N21 G00 G80 Z100;

N22 M05;

N23 G28 G91 X0 Y0Z0;

N24 G90;

N25 T08 M06; (COUNTERSINK BIT 90 DEGREE, DIA 16)

N26 D500 M03;

N27 G00 G55 G43 H08 X 10 Y10 Z10;

N28 G81 X10 Y10 Z-5 R2 F40;

N29 X100 Y10;

N30 X100 Y100;

N31 X10 Y100;

N32 G00 G80 Z100;

N33 M05;

N34 T09 M06; (M8 TAP);

N35 S150 M03;

N36 G00 G55 G43 H09 X10 Y10 Z10;

N37 G95 G84 X10 Y10 Z-25 R2 F1.5;

N38 X100 Y10;

N39 X100 Y100;

N40 X10 Y100;

N41 G00 G80 Z100;

N42 M05;

N43 G91 G28 G00 X0 Y0 Z0;

N44 M 30;

Block search and restart

Objectives : At the end of this exercise you shall be able to
- **identify the block numbers in the program where you want to restart the program**
- **move the tool to restart position**
- **restart the program.**

Job Sequence

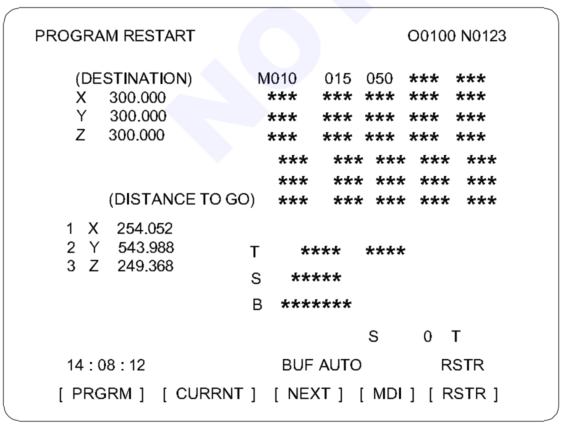
Generally the following operations are required when machining is interrupted due to a reason such as damaged tool/power and is restarted.

- Remove the cause which interrupted the machinery
- Locate the interrupted block number in the program
- Restore the machine suitable status (including the auxiliary functions like coolant on, tool call and spindle status by MDI mode)
- Move the tool to the position suitable for restarting the machine
- Resume automatic operation from the interrupted block.

When the power is turned ON or emergency stop is released, perform all necessary operation at that time, including reference position return

- Load the program to run in auto mode
- Locate the block number where you want to restart the program
- Turn on the program restart button on the machine operator panel
- Input 'Q' and sequence number of the block to be resumed (press the cursor down ⇩ Key in older version)
- Once the search to resume the program has been completed, the CRT screen displays the program resume screen (Fig 1)

Fig 1



MA20N24193H1

DESTINATION shows the position at which machining is to restart

DISTANCE TO GO shows the distance from the current tool position to the position where machining is to restart. A number to the left of each axis name indicates the order of axes (determined by parameter along which the tool moves to the restart position).

M: Thiry-five most recently specified M codes

T: Two most recently specified T codes

S: Most recently specified S codes

B: Most recently specified B code

- Turn the program to re-start switch OFF. At this time, the figure at the left side of axis name DISTANCE TO GO blinks
- Check the screen for M, S, T and B codes to be executed. If they are found, execute the M, S, T and B functions by the MDI mode. After execution, restore the previous mode.
- Check that the distance indicated under DISTANCE TO GO is correct. Also check whether there is the possibility that the tool might hit a work piece or other objects when it moves to the machining restart position. If such a possibility exists, move the tool manually to a position from which the tool can move, to the machining restart position without encountering any obstacles.
- Press the cycle start button. The tool moves to the machining restart position at the dry run feed rate sequentially along axes in the order specified by parameter settings. Machining is then restarted

Warning: Don't interrupt/restart during program restart operation.

Mounting of work piece

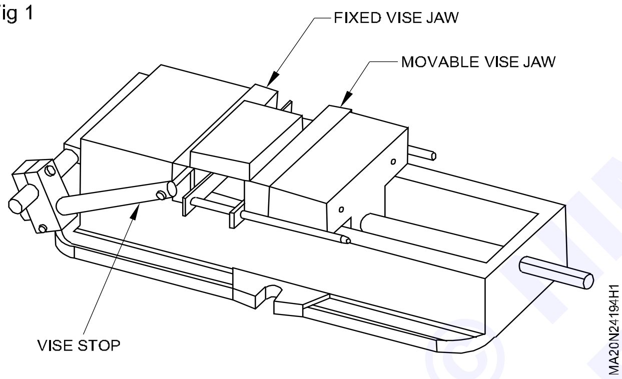
Objectives : At the end of this exercise you shall be able to
- **set, align and hold work piece on machine vice**
- **clamp larger work piece on machine table**
- **grip the work piece with the help of strip and stop pin**
- **grip the work piece with the help of locator and clamps**
- **set the fixture on machine table and grip the work piece.**

Job Sequence

TASK 1: **Setting vice on machine table (Fig 1)**

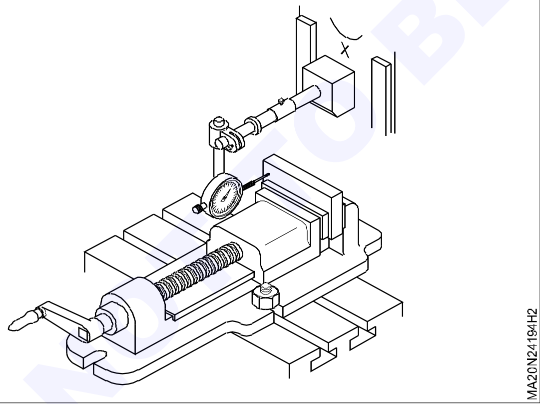
- Place the vice on the machine table
- Position the vice jaw parallel or perpendicular to X or Y axis
- Bolt the vice with suitable 'T' bolts and nuts

Fig 1



- Align the fixed vice jaw parallel or perpendicular to the X or Y axis (Fig 2)

Fig 2

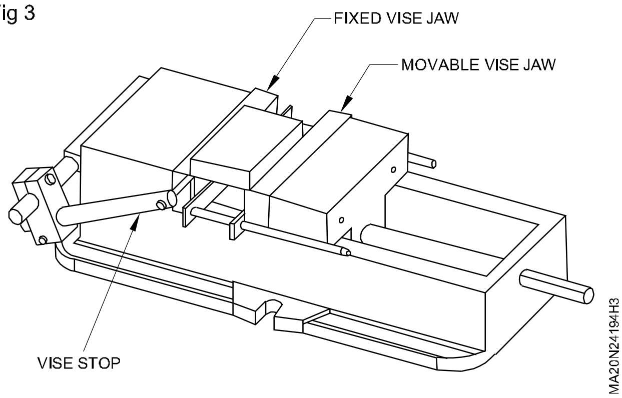


- Fix the parallel block in between jaws
- Fix the dial test indicator
- Align the vice by moving in jog mode
- Tighten the 'T' bolts and nuts
- Check the alignment once again

Setting work piece in the vice (Fig 3)

- Deburr the work piece
- Open the vice jaw to accommodate the work piece
- Place the suitable parallel blocks on the vice so that the work piece should project 3 mm to 5 mm above the jaw surface
- Position the work piece and tight the work piece with help of handle provided on the vice

Fig 3



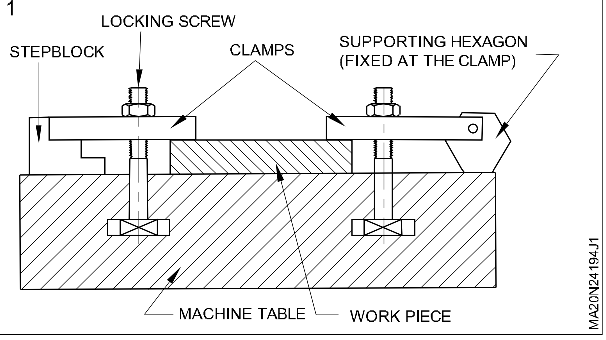
— — — — — — — —

TASK 2: **Clamping of Work piece (Fig 1)**

Larger work pieces are mounted directly on machine table and clamped

- Clean and deburr the work piece
- Clean the machine table
- Place the work piece directly on the table position the work piece reference surface parallel to X axis
- Insert the 'T' bolt with suitable clamp and support block. The resting of the support block height should be slightly less than the work piece
- slightly tighten the clamps to hold the work piece
- Align the reference surface with the help of dial test indicator
- Fully tighten the clamps to hold the work piece firmly

Fig 1



Note: The clamps are positioned parallel and it opposite sides of the work piece.

1 Clamps

2 Step block

3 Locking screw

4 Supporting hexagon (fixed at the clamp)

5 Work piece

6 Machine table

Hints on clamping

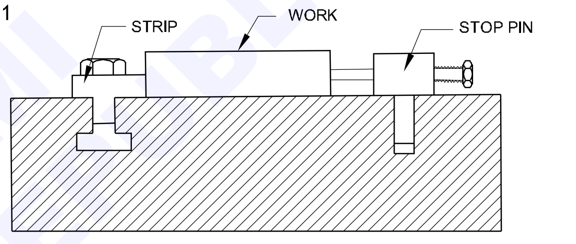
- A clamping system must hold the work piece firmly against the locating elements and the cutting forces developed during operation without causing damage to it
- Clamping system should be positioned at thick section of the work piece
- Clamping should be positioned to direct the clamping force on a strong supported part of the work piece
- Clamping system should not obstruct the parts of cutting tool
- Clamping force shall be directed towards support/ locators.

— — — — — — — —

TASK 3: **Griping the work piece for entire top surface machining (Fig 1)**

- Place the strip on the table align and clamp
- Position the stop pin arrangement to accommodate the work piece
- Clean and position work piece in between strip and stopping
- Tighten the stop pin hold to hold grip the work piece against the strip
- Check the reference surface for parallel to the axis

Fig 1



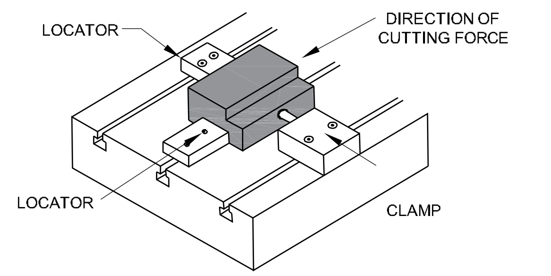
— — — — — — — —

TASK 4: **Griping work piece with locators (Fig 1)**

- Fix and align the locator in X and Y axis and align it
- Support the work piece against the locators
- Clamp the work piece by pressing the clamp in the work piece and tighten it in both the direction
- Check for the rigidity of gripping the work piece

Note: The cutting force should act towards the locator.

Fig 1

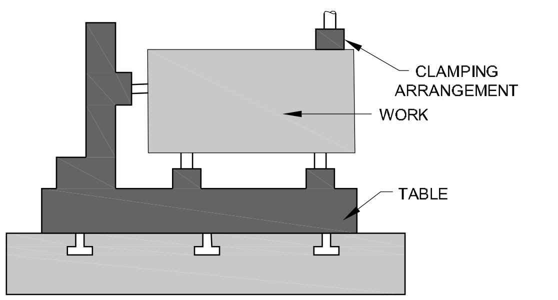


— — — — — — — —

TASK 5: **Holding the work piece in a fixture for machining (Fig 1)**

- Fix the fixture on the machine table
- Align the reference surface parallel or perpendicular to the X or Y axis
- Clean the work piece and insert in to the fixture
- Clamp the work piece with clamps provided in the fixture.

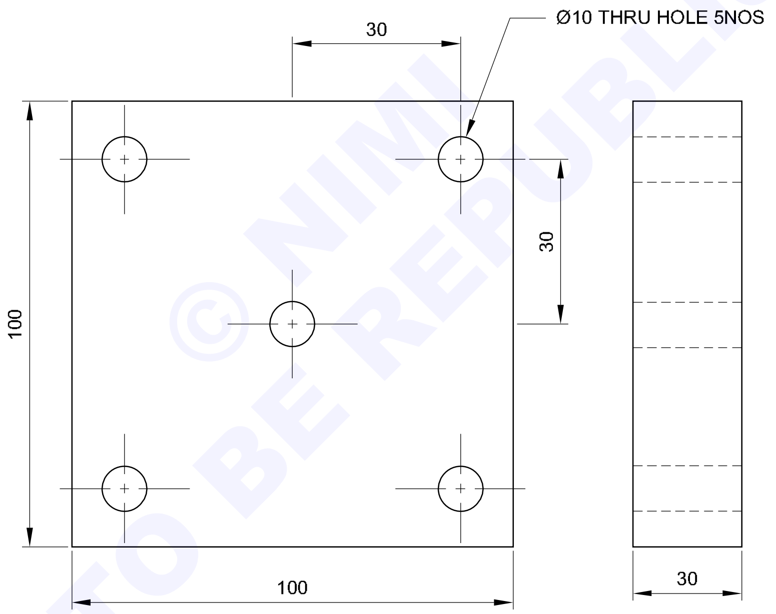
Fig 1



— — — — — — — —

Machining Part on CNC VMC with face milling, drilling (siemens)

Objectives : At the end of this exercise you shall be able to
- **set the work piece on VMC machine for machining**
- **set the tools in tool magazine as per requirement**
- **measure work and tool offset and enter in offset page**
- **prepare and verify the program on simulator**
- **run the program in auto mode operation**
- **check the dimensions and correct it if required.**



–	PRE MACHINED105 X 105 X 35 mm	–	–	–	1	2.4.195
NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.
SCALE 1 : 1	VMC FACE MILL AND DRILLING				TOLERANCE	TIME
					CODE NO. MA20N24195E1	

Job Sequence

- Make the Job Setting on the Machine.
- Make the Tool Setting as per the given Job.
- Take the Work offset in X & Y Axes.
- Take the Tool Offset in Z - Axis.
- Write the Part Program for Face Milling & Drilling Operations as per the given Job.
- Enter the Program in the Machine.
- Verify the Program in the Simulator Option on Machine.
- Make the wear offset corrections for achieving the Job Size in Tolerance as per drawing.

Program for face mill and drilling (siemens)

```
MSG ("FACE MILLING");
G17G71G90G40;
T1M6; FACE MILL CUTTER DIA 50;
D1;
M03S600;
G0G90G54X0Y0;
G0Z20;
M07;
CYCLE61(10,0,5,-2,-50,-50,50,50,1,75,0,200,31,0,1,11010);
G0Z20;
M9;
G75Z0;
G75X0Y0;
MSG ("DRILLING DIA 10MM X 4 PLACES");
T2M6; DRILL DIA 10
D1;
M03S1000;
G0G90G54X0Y0;
G0Z20;
M07;
MCALL CYCLE82(10,5,1,-32,,0.6,0,10001,12);
CYCLE802(111111111,111111111,0,0,30,30,-30,30,- 30,-30,30,-30,,,,,,,,,0,0,1);
MCALL;
G0Z20;
M9;
G75Z0;
G75X0Y0;
M30;
```

Skill Sequence

Face milling using cycle

Objective: This shall help you to
- **prepare the face mill program using face milling cycle.**

Face milling using cycle

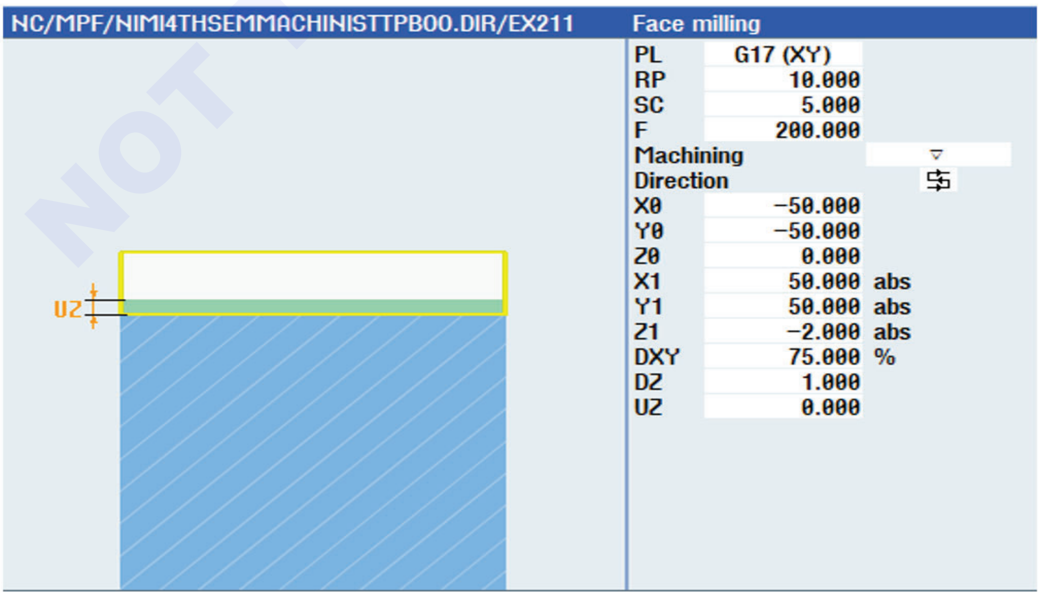
CYCLE61(10,0,5,-2,-50,-50,50,50,1,75,0, 200,31,0,1, 11010);

1 Click on the “Milling” option.

2 Click on the “Face milling” option.

3 In the parameter window enter the following parameters as per the Fig 1.

Fig 1



Drilling cycle with positions

Objective: This shall help you to
- **prepare the drilling program using drilling cycle with positions.**

MCALL CYCLE82(10,5,1,-32,,0.6,0,10001,12);

CYCLE802(111111111,111111111,0,0,30,30,-30,30,-30,-30,30,-30,,,,,,,,,0,0,1);

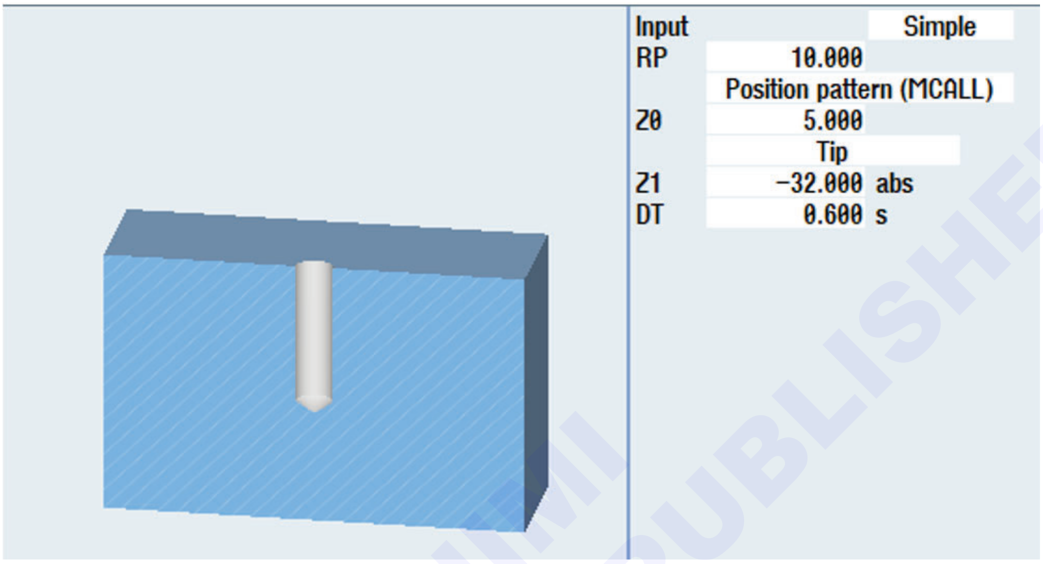
1 Click on the Drilling Option.

2 Click on the "Drilling Reaming" Option

3 Click on the "Drilling Option" & fill the Parameters as shown in the Fig 1.

4 Then Press "Accept" the Following Programming Line appears in the Program

Fig 1



5 MCALL CYCLE82(10,5,1,-32,,0.6,0,10001,12);

6 Click on the Drilling Option.

7 Click on the Positions Option

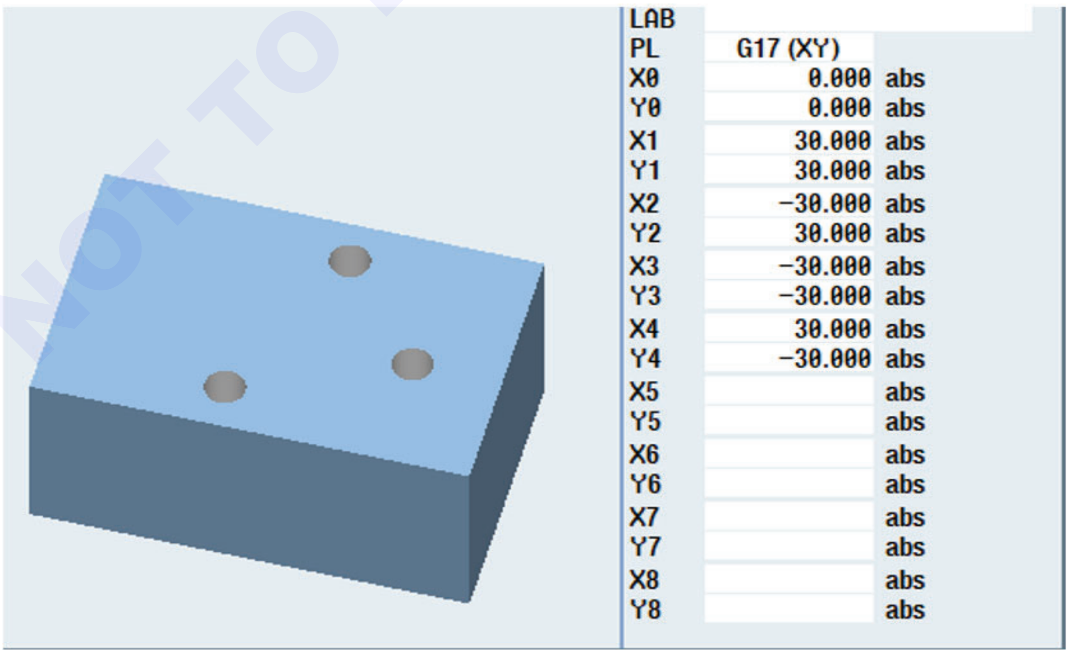
8 Click on the Option 

9 In the Position window fill the Co - Ordinates of the Holes as shown in the Fig 2.

10 Then Press " Accept" the following Programming line appears in the Program.

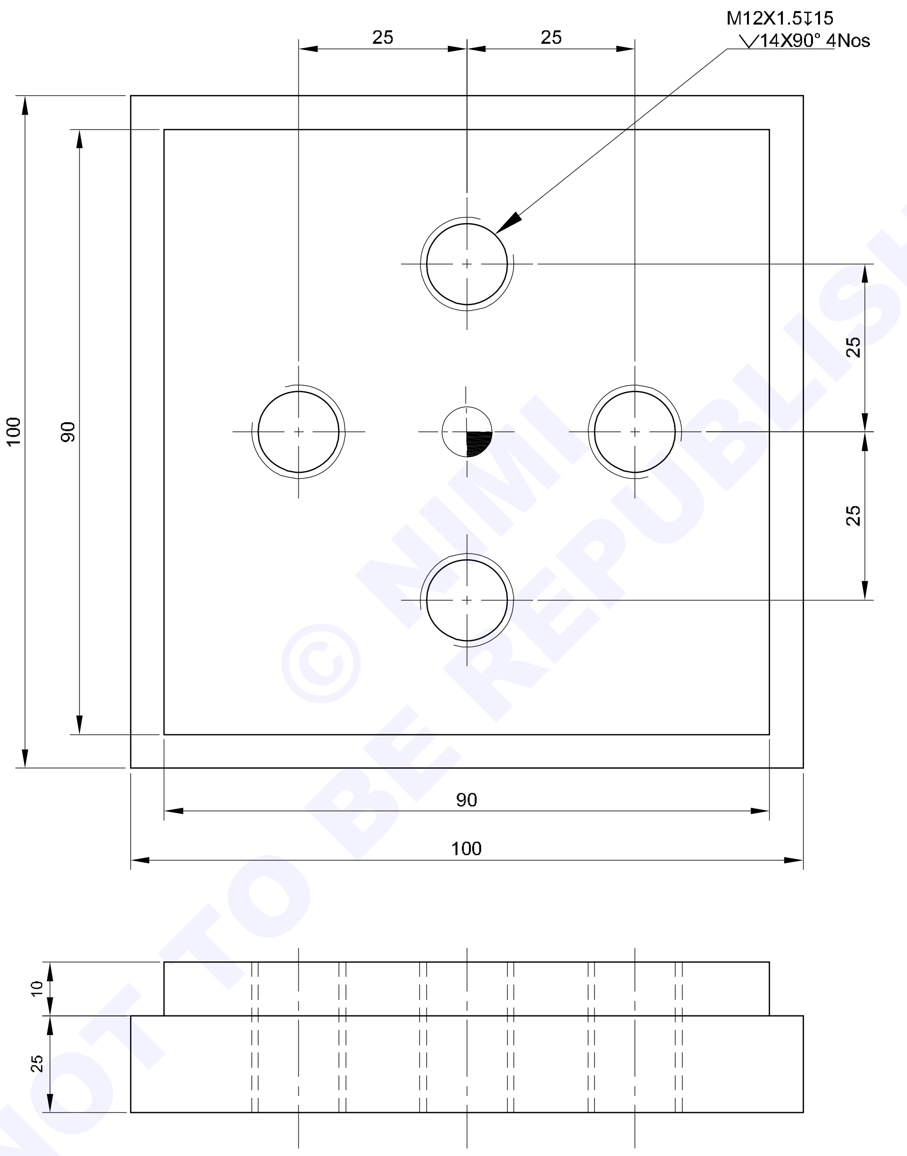
CYCLE802(111111111,111111111,0,0,30,30,-30,30,-30,-30,30,-30,,,,,,,,,0,0,1);

Fig 2



Machining part with face mill, side mill and drilling operation (Siemens)

Objectives : At the end of this exercise you shall be able to
- **perform face milling operation in VMC**
- **machine side milling with CRC**
- **carryout drilling operations using cycles.**



ALL DIMENSIONS ARE IN MM

1	PRE MACHINED 90 X 90 X 25 mm	–	Fe310	–	–	2.4.196
NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.

SCALE 1 : 1	TITLE : FACE MILLING,SIDEMILL AND DRILLING OPERATION	TOLERANCE	TIME
		CODE NO. MA20N24196E1	

Job Sequence

TASK 1: **Face milling**

- Prepare the Part Program for Face Milling, using face milling cycle61(cutter dia 50 mm)
- Verify the Program in the Simulator Option on Machine.
- Set the work piece on the Machine.
- Set the Tool in tool magazine as per the program
- Take the Work offset in X & Y Axes.
- Take the Tool Offset in Z - Axis.
- Enter the offset values in offset pages
- Check the tool offset for its correctness
- Run the program in auto mode
- Verify the dimension with respect to the drawing , if required do the wear correction

Program for face milling

MSG("FACE MILLING")

G17G71G90G40;

T1M6; FACE MILL CUTTER DIA 50;

D1;

M03S600;

G0G90G54X0Y0;

G0Z20;

M07;

CYCLE61(10,0,5,-2,-50,-50,50,50,1,75,0,200, 31,0,1,11010);

G0Z20;

M9;

M05;

G75Z0;

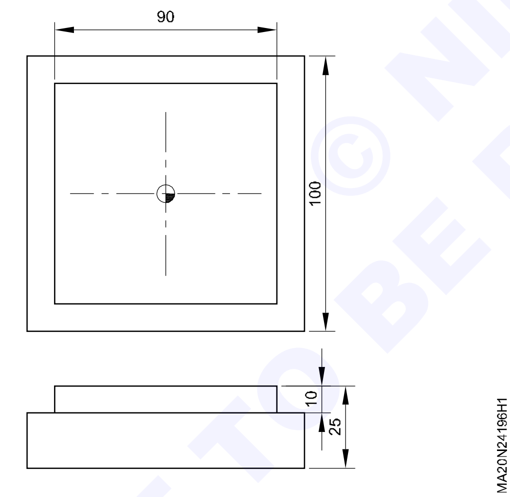
G75X0Y0;

M30;

— — — — — — — —

TASK 2: **Side milling (Fig 1)**

Fig 1



- Prepare the Part Program for side Milling, using side milling cycle76(end mill dia 20 mm)
- Verify the Program in the Simulator Option on Machine.
- Set the work piece on the Machine.
- Set the Tool in tool magazine as per the program
- Take the Work offset in X & Y Axes.
- Take the Tool Offset in Z - Axis.
- Enter the offset values in offset pages
- Check the tool offset for its correctness
- Run the program in auto mode
- Verify the dimension with respect to the drawing, if required do the wear correction

Program for side milling

MSG("SIDE MILLING")

G17G71G90G40;

T2M6; END MILL CUTTER DIA 20;

D1;

M03S2000

G0G90G54X0Y0;

G0Z20;

M07;

CYCLE76(10,0,5,,10,90,90,5,0,0,0,2.5,0,0,100,50,0,1,100, 100,1, 2,1100,1,101);

G0Z20;

M9;

M05;

G75Z0;

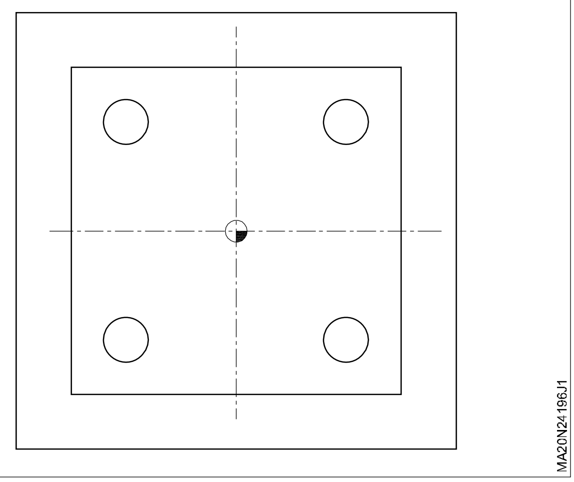
G75X0Y0;

M30;

— — — — — — — —

TASK 3: **Drilling (Fig 1)**

Fig 1



- Prepare the Part Program for drilling, using cycle76(drill dia 10.5mm)
- Verify the Program in the Simulator Option on Machine.
- Set the work piece on the Machine.
- Set the Tool in tool magazine as per the program
- Take the Work offset in X & Y Axes.
- Take the Tool Offset in Z - Axis.
- Enter the offset values in offset pages
- Check the tool offset for its correctness
- Run the program in auto mode
- Verify the dimension with respect to the drawing , if required do the wear correction

Drilling program

MSG ("PRE DRILLING HOLE DIA 10.5MM X 22MM DEEP X 4 PLACES")

T3M6; DRILL DIA 10.5

D1;

M03S1000;

G0G90G54X0Y0;

G0Z20;

M07;

MCALL CYCLE82(10,2,1,-22,,0.6,0,10001,12);

CYCLE802(111111111,111111111,25,0,0,25,-25,0,0,-25,,,,,,,,,,,0,0,1);

MCALL;

G0Z20;

M9;

M05;

G75Z0;

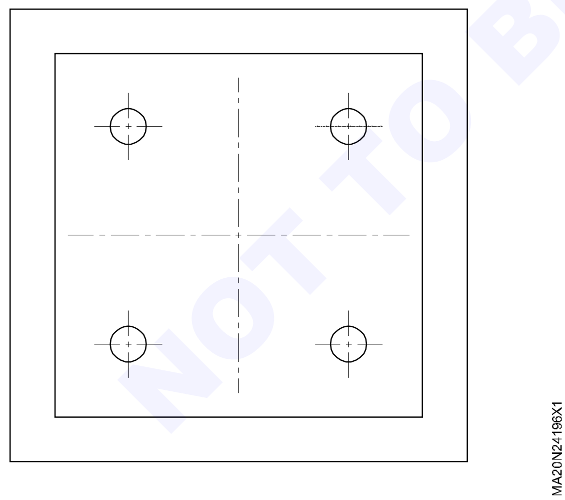
G75X0Y0;

M30;

— — — — — — — —

TASK 4: **Chamfering (Fig 1)**

Fig 1



- Prepare the Part Program for chamfering on the pre drilled holes using cycle 81&82
- Verify the Program in the Simulator Option on Machine.
- Set the work piece on the Machine.
- Set the Tool in tool magazine as per the program
- Take the Work offset in X & Y Axes.
- Take the Tool Offset in Z - Axis.
- Enter the offset values in offset pages
- Check the tool offset for its correctness
- Run the program in auto mode
- Verify the dimension with respect to the drawing, if required do the wear correction

Chamfering program

MSG ("CHAMFER DIA14 X 90DEG X 4 PLACES")

T4M6; CHAMFER DIA 16;

D1;

M03S1000;

G0G90G54X0Y0;

G0Z20;

M07;

MCALL CYCLE81(25,10,20,,20,0.6,0,1,11);

CYCLE802(111111111,111111111,25,0,0,25,-25,0,0,-25,,,,,,,,,,0,0,1);

MCALL;

G0Z20;

M9;

M05;

G75Z0;

G75X0Y0;

M30;

— — — — — — — —

TASK 5: **Tapping (Fig 1)**

- Prepare the Part Program for tapping using cycle 84
- Verify the Program in the Simulator Option on Machine.
- Set the work piece on the Machine.
- Set the Tool in tool magazine as per the program
- Take the Work offset in X & Y Axes.
- Take the Tool Offset in Z - Axis.
- Enter the offset values in offset pages
- Check the tool offset for its correctness
- Run the program in auto mode
- Verify the dimension with respect to the drawing, if required do the wear correction

Tapping Program

MSG ("M12X1.5X15 DEEP TAP HOLES X 4 PLACES")

T5M6; M12X1.5 TAP

D1;

M03S1000;

G0G90G54X0Y0;

G0Z20;

M07;

MCALL CYCLE84(20,5,1,-15,,0.6,5,,1.5,0,100,100,0,1,0,0,5,1.4,,,,,11001,1001002)

CYCLE802(111111111,111111111,25,0,0,25,-25,0,0,-25,,,,,,,,,,0,0,1);

MCALL;

G0Z20;

M9;

G75Z0;

G75X0Y0;

M30;

— — — — — — — —

Skill sequence

Side milling

Objectives : At the end of this exercise you shall be able to
- **prepare the side milling with spigot option program using cycle 76**
- **prepare the chamfering program using cycle 81 &82**
- **prepare the tapping program using cycle 84.**

Side milling with spigot option

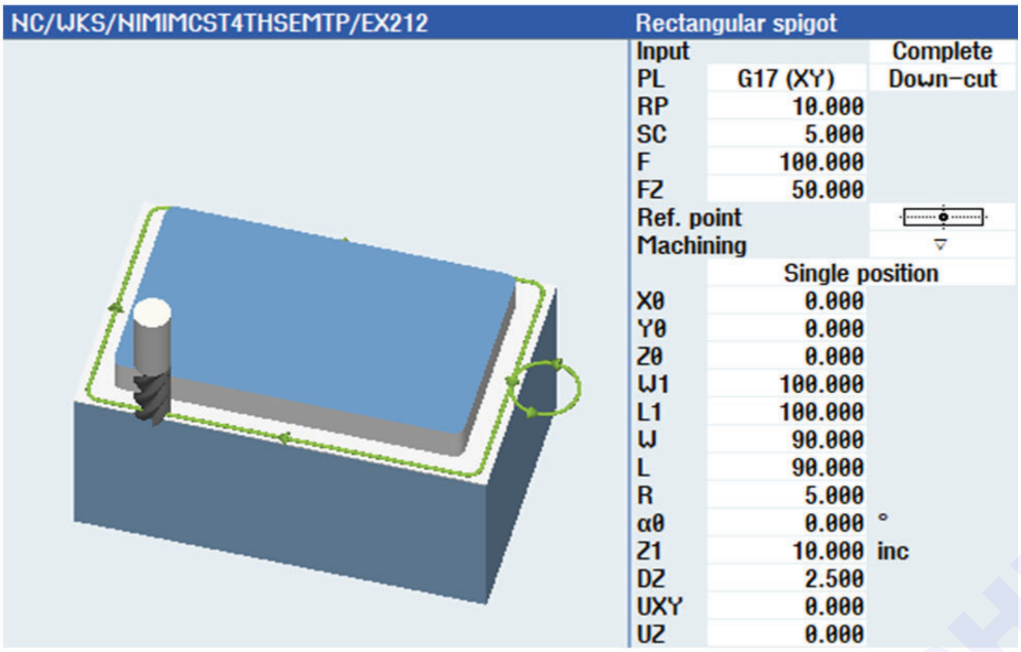
- Click on the "milling" option
- Click on the "multi edge spigot" option
- Click on the "rectangle spigot" option
- In the parameter window enter the following parameters as per the Fig 1.

Press accept key

Then the tool path will be automatically created

CYCLE76(10,0,5,,10,90,90,5,0,0,0,2.5,0,0,100,50,0,1,100,100,1,2,1100,1,101)

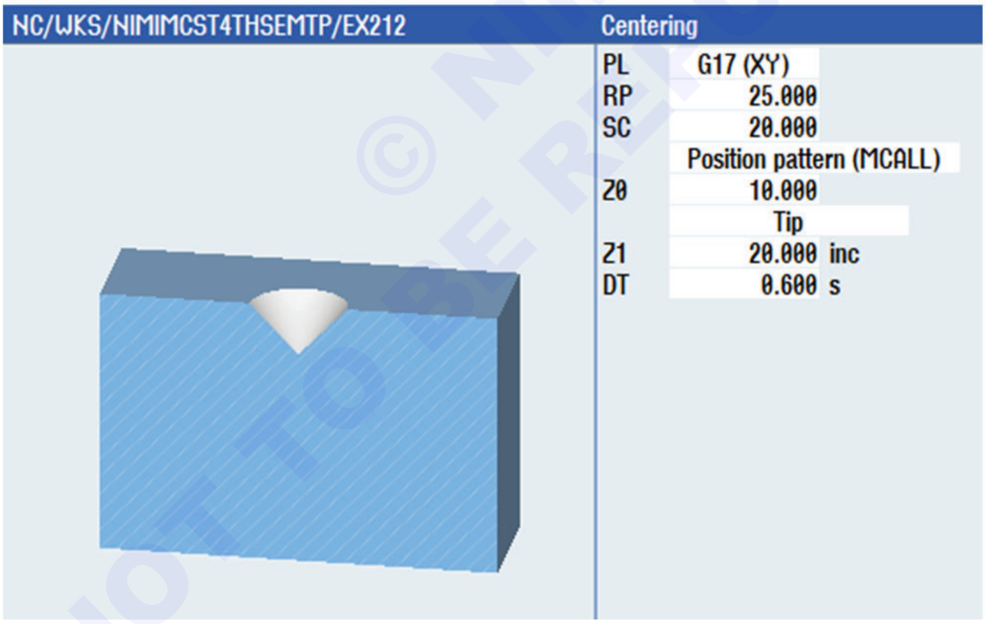
Fig 1

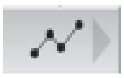


Chamfering on the predrilled holes

- Click on the "Drilling" Option.
- Click on the "Centering" Option
- Fill the Parameters as shown in the Fig 2
- Then Press "Accept" the Following Programming Line appears in the Program MCALL CYCLE81 (25,10,20,,20,0.6,0,1,11)

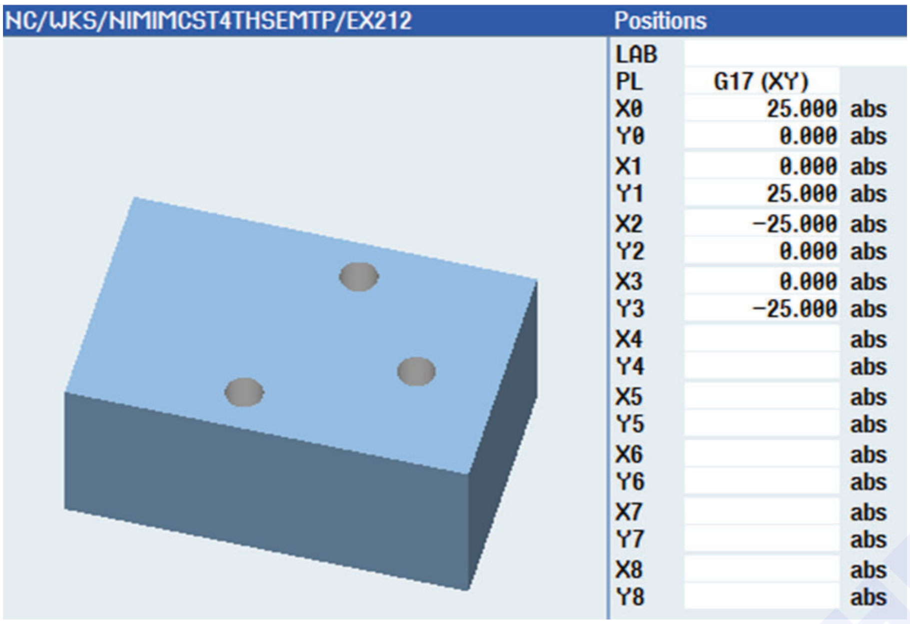
Fig 2



- Click on the Drilling Option.
- Click on the Positions Option
- Click on the Option 
- In the Position window fill the Co - Ordinates of the Holes as shown in the Fig 3.
- Then Press " Accept" the following Programming line appears in the Program.

CYCLE802(111111111,111111111,25,0,0,25,-25,0,0,-25,,,,,,,,,,,0,0,1);

Fig 3

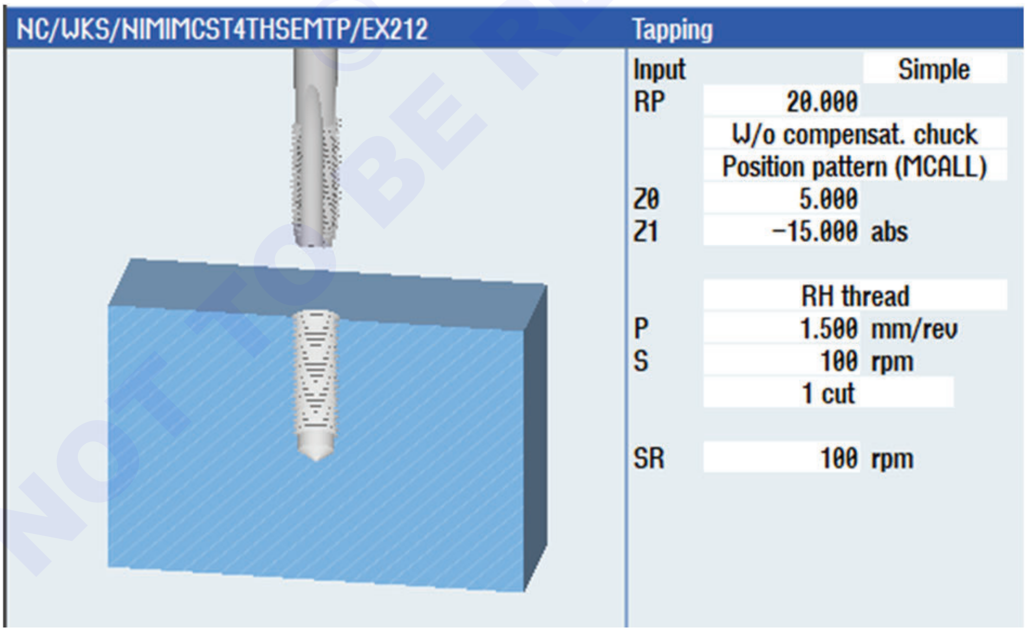


Tapping operation on the holes

Note: CYCLE802 IS SAME FOR DRILLING, COUNTERSINKING / CHAMFERING, TAPPING SINCE, IT IS THE CO - ORDINATES OF THE HOLES OR POSITIONS.

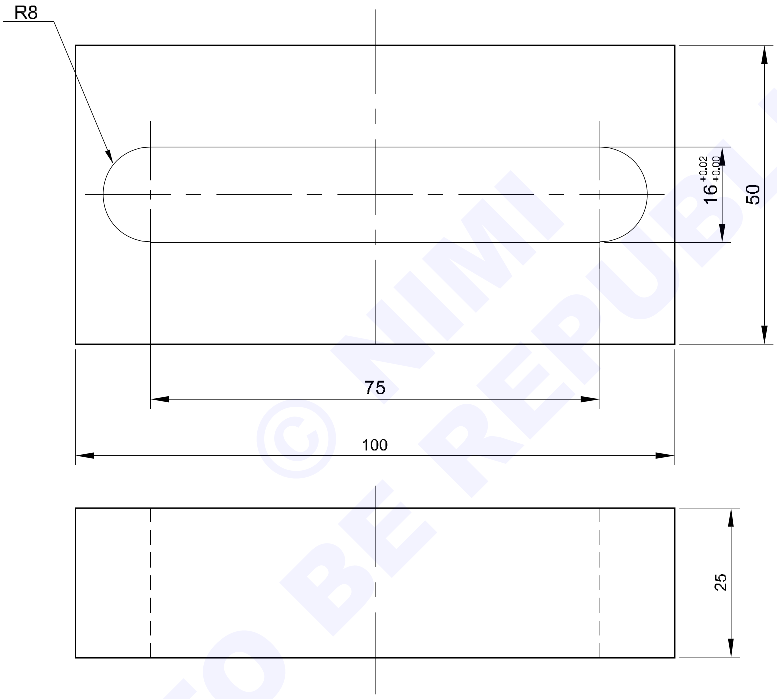
- Click on the "Drilling" Option.
- Click on the "Thread" Option
- Click on the " Tapping" Option
- Fill the Parameters as shown in the Fig 4 and click on the "accept" option and the following program line appears in the Program.
- MCALL CYCLE84(20,5,1,-15,,0.6,5,,1.5,0,100,100,0,1,0,0,5,1.4,,,,,11001,1001002)
- CYCLE802(111111111,111111111,25,0,0,25,-25,0,0,-25,,,,,,,,,,,0,0,1);

Fig 4



Slot machining using CRC

Objectives : At the end of this exercise you shall be able to
- **prepare the program for slot milling**
- **execute the program**
- **correct the dimension using wear compensation.**



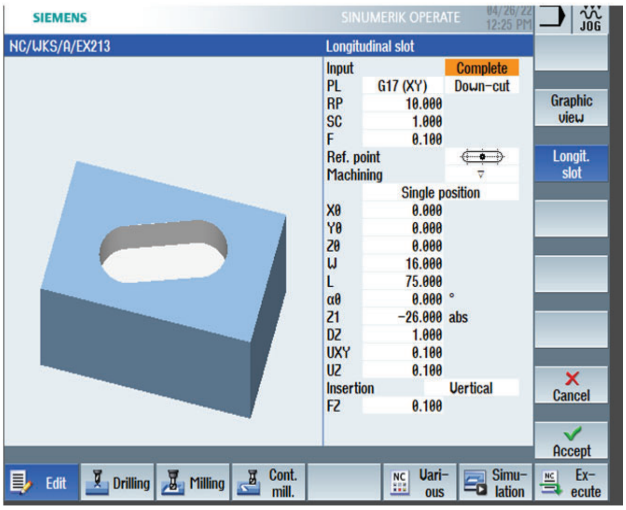
–	PRE MACHINED 100 X 50 X 25 mm	–	ALUMINIUM- LM25	–	–	2.4.197
NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.

SCALE 1 : 1	SLOT MILLING	TOLERANCE	TIME
		CODE NO. MA20N24197E1	

Job Sequence

- Read the drawing
- List out the tools required to do the practical
- Prepare the CNC program for longitudinal slot milling, select the parameter as in Fig 1.

Fig 1



- Verify the program
- Set the work piece on the machine table
- Measure work offset and tool offset enter in offset page
- Execute the program and verify the slot size
- If any variation in size correct in wear off set

For internal dimension

Example

The programmed value is say 16mm

Actual; measures size is 15 mm

The difference in programmed size and actual size 1mm

Divide the difference by 2 that is equal 0.5 mm

Enter this 0.5mm in wear radius as -0.5mm

Re execute the program

And recheck the size until you get the required tolerance

Example program for slot milling

WORKPIECE (," ",,'RECTANGLE",64,25,-25,80,100,50)

T= "CUTTER 10" (End cutting end mill dia 10 mm)

M06

M03 S700

G00 X0 Y0 Z100

SLOT1(10,0,1,-25,,1,75,16,0,0,5,0,0,0,1,0.1, 11,0.1,15,15,0.1,0,2,0,1,2,100,1011,102) SLOT MILLING PARAMETER)

G00 Z100

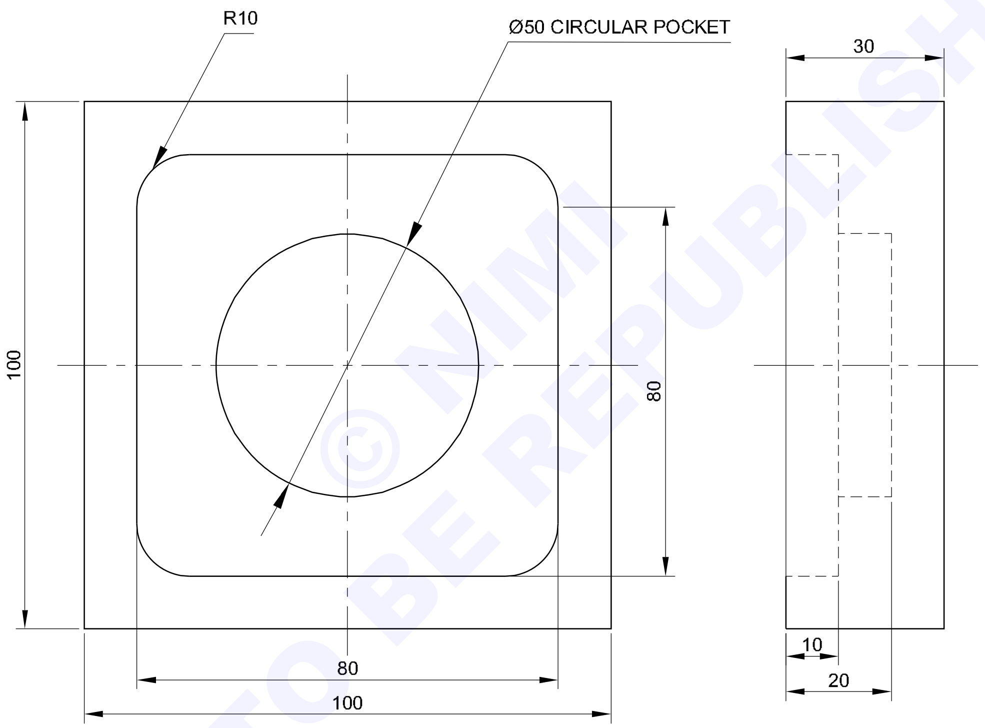
M05

MCALL

M30

Machining parts with pockets

Objectives : At the end of this exercise you shall be able to
- **prepare the part program for rectangular pocket milling using cycle pockets**
- **prepare the part program for circular pocket milling using cycle pockets**
- **machine the pockets in automode operation.**



ALL DIMENSIONS ARE IN mm

1	PRE MACHINED 100 X 100 X 30 mm	–	ALUMINIUM- LM25	–	–	2.4.198
NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.

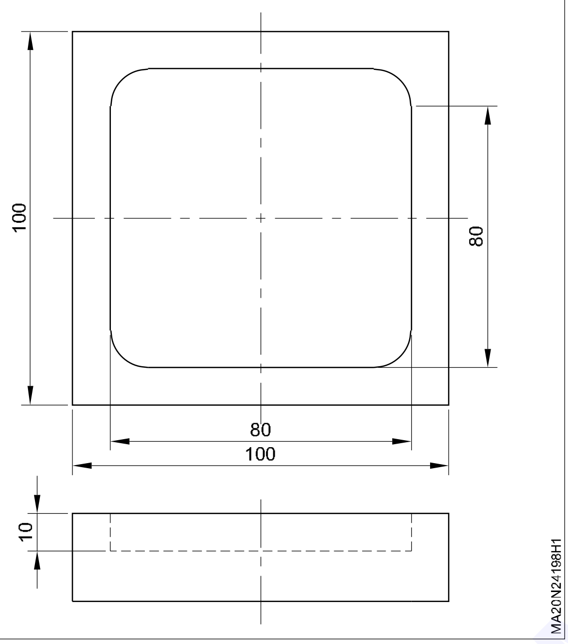
SCALE 1 : 1	POCKET MILLING	TOLERANCE	TIME
		CODE NO. MA20N24198E1	

Job Sequence

TASK 1: **Rectangle pocket milling**

- Prepare the part program for rectangular pocket milling, using pocket 3 (end mill dia 20 mm) (Fig 1)

Fig 1



- Verify the program in the simulator option on machine
- Set the work opiece on the machine
- Set the tool in tool magazine as per the program
- Take the work offset in X & Y axes
- Take the tool offset in Z-axis.
- Enter the offset values in offset pages
- Check the tool offset for its correctness
- Run the program in auto mode
- Verify the dimension with respect to the drawing, if required do the wear correction

Program for rectangular pocket milling

MSG (" POCKET MILLING")

G17G71G90G40

T2M6; END MILL CUTTER DIA 20

D1

M03S2000

H0H90H54X0Y0

G0Z20

M07

MSG ("RECTANGULAR POCKET 90 X 90MM")

POCKET3 (10,0,5,-10, 80,80,0,0,0,0,2,0,0,400,0.1,0,21,75, 8,3,15,1,2,0,1,2,11100,11,110)

MSG (" CIRCULAR POCKET DIA 50 MM")

POCKET4 (10,0,5,-20, 50,0,0,2.5,0,0,400,0.1,0, 1011,50,9,15,0,2,0,1,2,10100,111,110)

G0Z20

M9

G75Z0

G75X0Y0

M30

— — — — — — — —

TASK 2: **Circular pocket milling**

- Prepare the part program for circular pocket milling, using pocket4 (end mill dia 20 mm) (Fig 1)
- Verify the program in simulator option on machine.
- Set the work piece on the machine.
- Set the tool in tool magazine as per the program
- Take the work offset in X & Y axes
- Take the tool offset in Z-Axis
- Enter the offset values in offset pages
- Check the tool offset for its correctness
- Run the program in auto mode
- Verify the dimension with respect to the drawing, if required do the wear correction

Follow the rectangular pocket milling steps for the development of the program.

— — — — — — — —

Skill sequence

Pocket milling for rectangular & circular pockets

Objective : This shall help you to

- **generate the program for rectangle and circular pocket using cycles.**

Rectangular pocket

1 CLICK ON THE "MILLING "OPTION

2 CLICK ON THE "POCKET" OPTION

3 CLICK ON THE "RECTANGLE POCKET" OPTION

4 In the parameter window enter the folowing parameters as per Fig 1

5 Then press "Accept" the following programming line appears in the Program

POCKET3 (10,0,5,-10, 80,80,0,0,0,0,2,0,0,400, 0.1,0,21,75,8,3,15,1,2,0,1,2,11100,11,110)

Similarly follow for circular pocket

1 CLICK ON THE "MILLING" OPTION

2 CLICK ON THE "POCKET" OPTION

3 CLICK ON THE "CIRCULAR POCKET" OPTION

4 In the parameter window enter the folowing parameters as per Fig 2.

5 Then press "Accept" the following programming line appears in the program

POCKET4(10,0,5,-20,50,0,0,2.5,0,0,400,0.1,0, 1011, 50,9,15,0,2,0,1,2,10100,111,110)

Fig 1

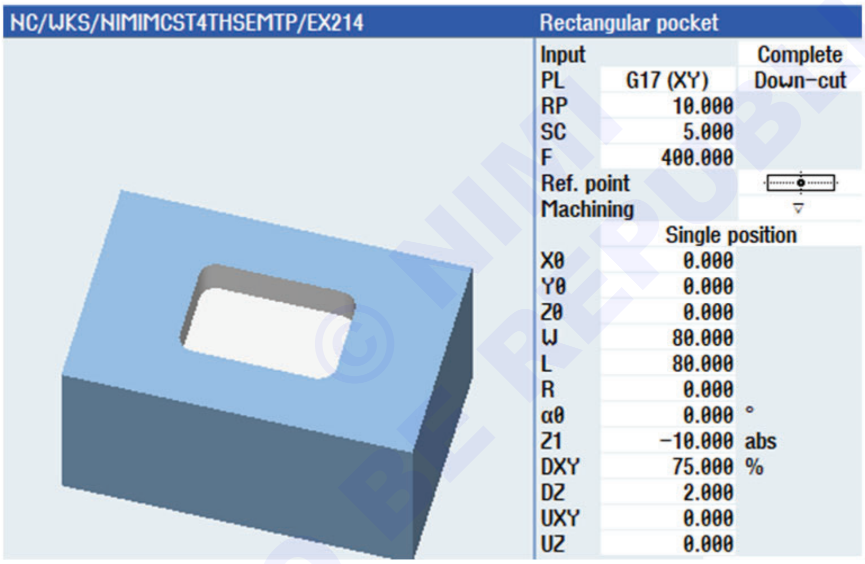
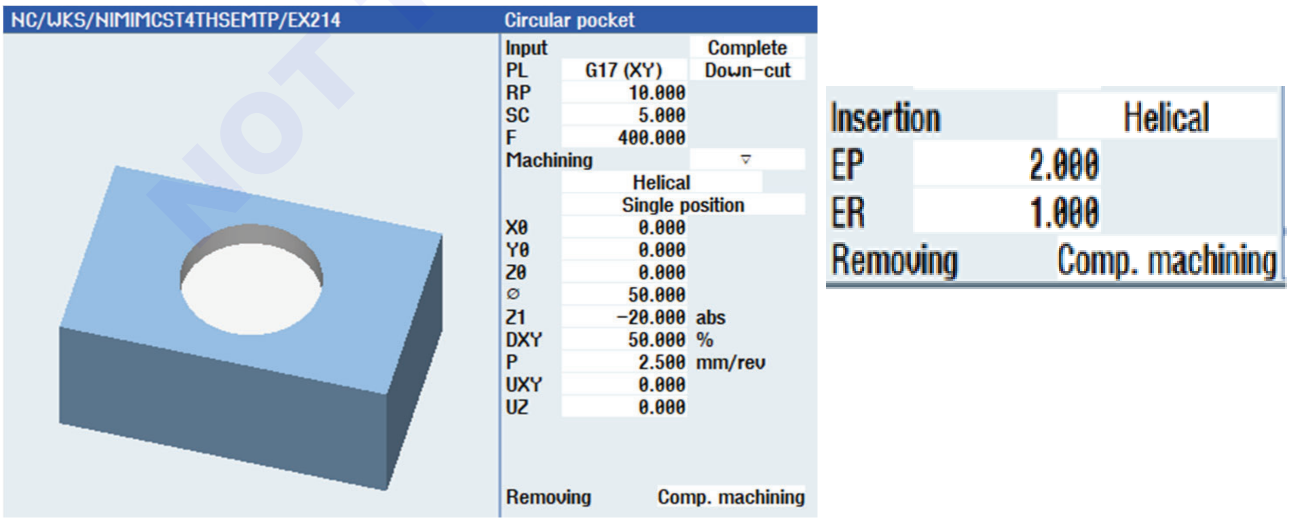
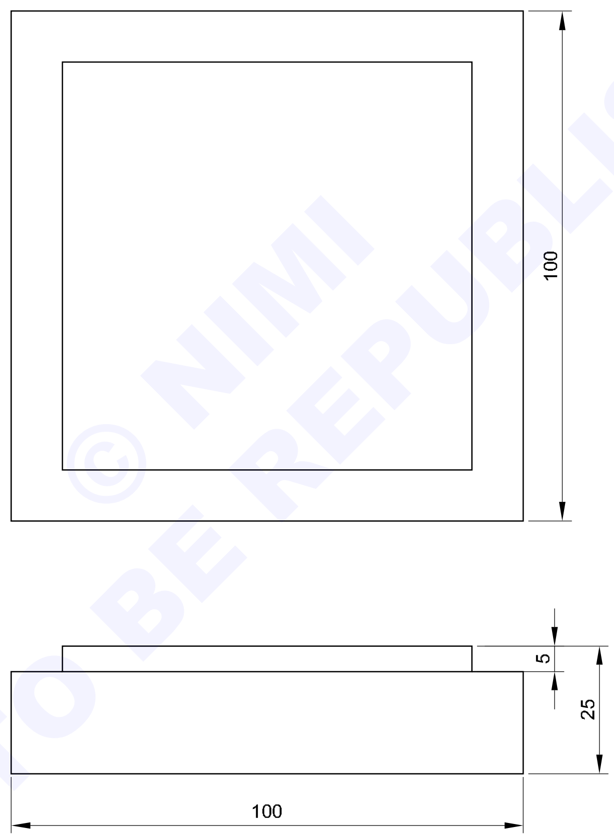


Fig 2



End milling with polar co-ordinates (Siemens)

Objectives : At the end of this exercise you shall be able to
- **prepare the end milling program using polar co-ordination**
- **decide the lead in lead out for CRC**
- **execute the program in auto mode - single block.**



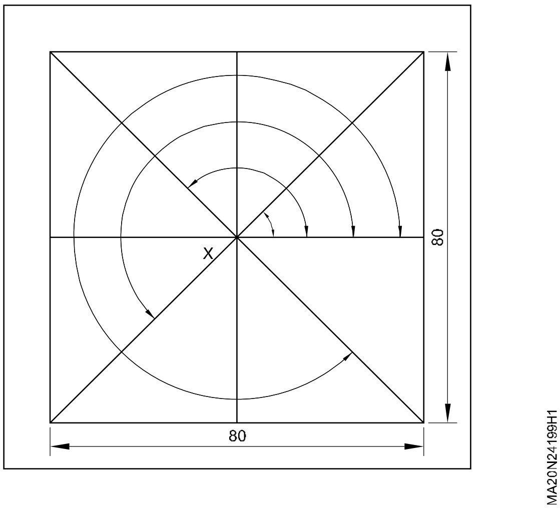
–	PRE MACHINED 100 X 100 X 25 mm	–	ALUMINIUM-LM25	–	–	2.4.199
NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.
SCALE 1 : 1	POLAR CO-ORDINATES (SIEMENS)				TOLERANCE	TIME
					CODE NO. MA20N24199E1	

Job Sequence

Polar program in Siemens

- Read the drawing
- List out the tool required to do the practical
- Fix the work offset at the center of the work piece Fig 1

Fig 1



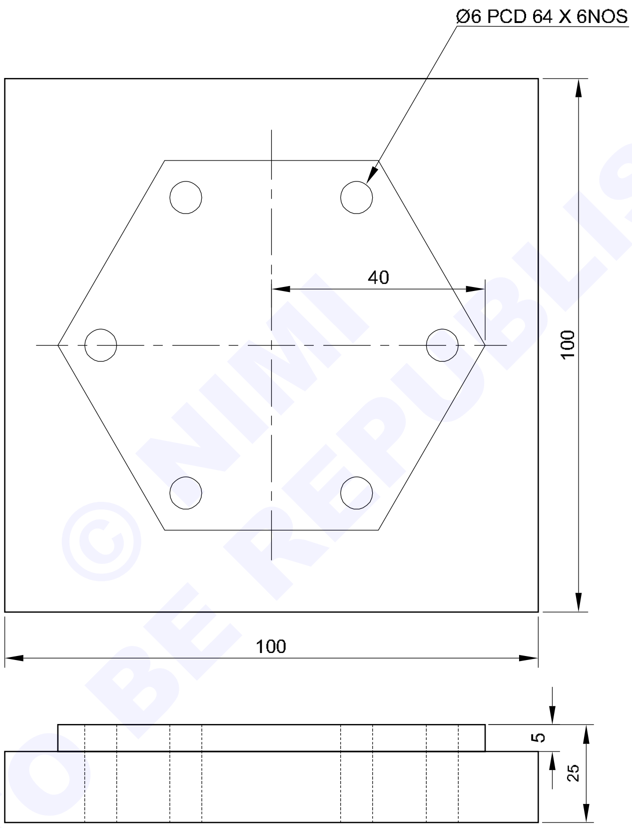
- Calculate the polar radius and polar angles
- Decide the lead in lead out for cutter radius compensation
- Prepare the program
- Verify the program
- Measure offset values and enter in offset page
- Execute the program auto mode in single block
- Verify the dimensions
- Calculate the polar radius
- Calculate the polar radius

Program for polar end milling

```
WORKPIECE (," " ,,"RECTANGLE",64,25,-25,-80,100,100)
T="CUTTER 20"
M06
M03 S500
G00 X-100 Y-100 Z100
G01 Z-5 F10
G42 G01 X-80 Y-50 F100
G111 X0 Y0
AP=225 RP=56.577
AP=315
AP=45
AP=135
AP=225
G40 G01 X-80 Y-80 F100
G00 Z100
M30
```

Polar co-ordinate program for end milling and drilling (Siemens)

Objectives : At the end of this exercise you shall be able to
- **prepare the program for the given drawing in polar co-ordinate with Siemens control**
- **verify and execute the program in auto mode.**



–	PRE MACHINED 100 X 100 X 25 mm	–	ALUMINIUM-LM25	–	–	2.4.200
NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.
SCALE 1 : 1	POLAR PROGRAM				TOLERANCE	TIME
					CODE NO. MA20N24200E1	

Job Sequence

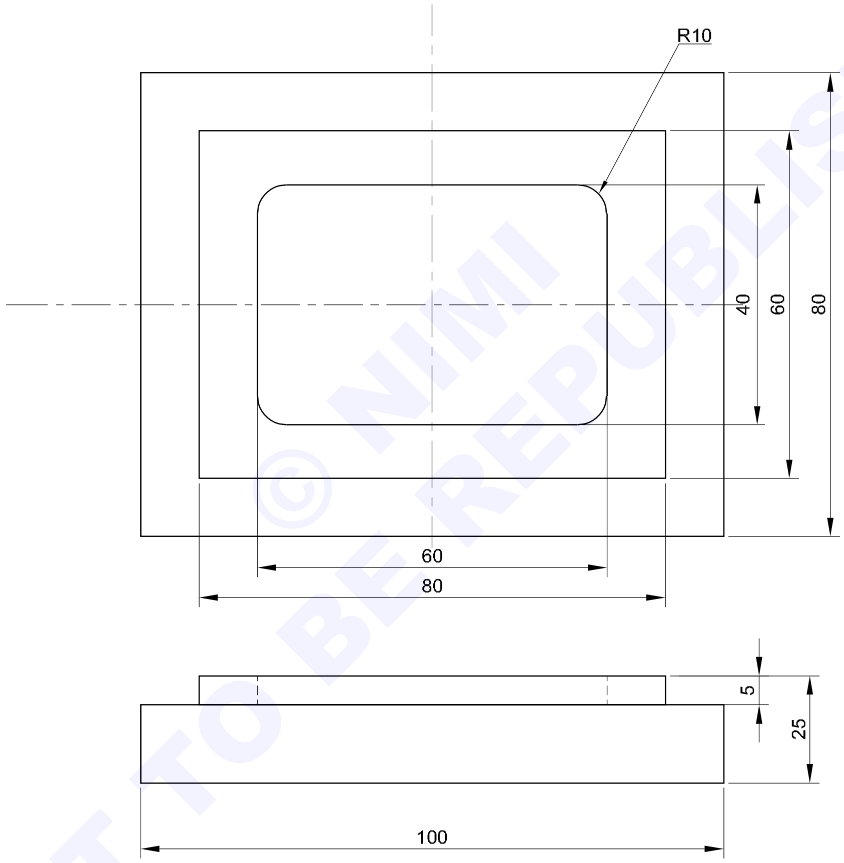
- Read the drawing
- Prepare the part program to mill hexagon to a depth of 5mm using polar co-ordinator with CRC
- Prepare the part program to drill 6 holes as shown in drawing using G81 with polar co-ordinate
- Verify the program by simulation
- Set the work piece on the machine
- Fix work offset at the left hand lower most corner.
- Measure tool offsets and enter the offsets in relevant offset page.
- Execute the program in single block
- Check size of hexagon, if required correct the size.

Sample program for hexagon and simple drilling.

```
NC/WKS/POL/POL
-"CUTTER 60"
Workpiece (,"",, "Box", 49, 0, -20, -80,0,0,100,100)
G17 G90
M3 S250
G00 X-30Y -90
G00 Z100
G01 Z-5
G42 G01 X-0 Y-60
G111 X50 Y50
AP = -60 RP=40
G01 Z-5 F100
AP-0
AP-60
AP-120
AP-180
AP-240
AP-300
AP-330
G00 Z100
G40X120 Y120
M05
T= "CENTERDRILL 12"
M06
S400 M03
G00 X50 Y50
G00Z10
G17 G90 G111 X50Y50
MCALL CYCLE (10, -30.0, 1,5,,0.6,10,1,11)
RP=32 AP=0
AP=60
AP=120
AP=180
AP=240
AP=300
G00 X100 Y100 Z100
M05
M30
```

Determining and entering wear offset

Objectives : At the end of this exercise you shall be able to
- **prepare the rectangular slot and spigot**
- **measure the size for internal slot and external spigot**
- **correct the dimension by wear offset correction.**



–	PRE MACHINED 100 X 80 X 25 mm	–	ALUMINIUM-LM25	–	–	2.4.201
NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.
SCALE 1 : 1	DETERMINING THE WEAR OFFSETS				TOLERANCE	TIME
					CODE NO. MA20N24201E1	

Job Sequence

- Prepare the CNC part program for the given drawing
- Set the work piece on the machine
- Measure work offset and tool offset
- Enter the values in respective offset page
- Execute the program in auto mode - single block
- Check the dimensions and compare with programmed value
- If it is in 'Z' direction correction to be done on tool length wear offset
- If it is less add the difference value that '+' sig
- If it is more subtract th difference values that '-' sig
- If it is in X and Y direction, correction to be done on cutter radius/diameter wear off set
- For external dimension '+' means '-' the wear offset
- For internal dimension '-' means '+' the wear offset
- After determining wear offset enter the value in wear tool wear page

Skill Sequence

Entering the tool wear offset in Siemens control

Objective: This shall help you to
- **enter the tool wear offset in respective offset page.**

Tool list

In the tool list all parameters and functions that are required to create and set the tools are displayed

Each tool is identified by the location number, the tool type, the tool name and the replacement of tool number

The most common tools and probes for turning, drilling and milling are reachable over the tool list

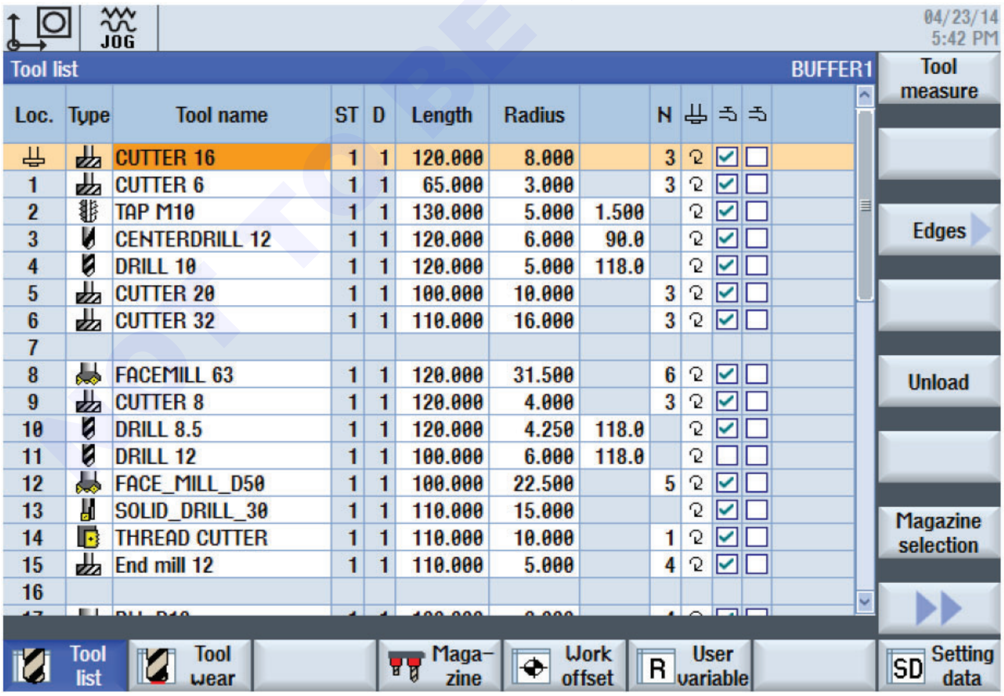
Geometrical and technological tool date can be assigned to each tool type. Depending on the tool type difference correction data are necessary

Selecting the function "Tool list"

After pressing the "MENU SELECT" key on the MCP, press the HSK 2 "Parameters". Alternatively, you can also press the "OFFSET" key on the MCP

By pressing the HSK 1 "Tool list" the "Tool list" window opens (Fig 1)

Fig 1



Loc.	Type	Tool name	ST	D	Length	Radius		N
		CUTTER 16	1	1	120.000	8.000		3
1		CUTTER 6	1	1	65.000	3.000		3
2		TAP M10	1	1	130.000	5.000	1.500	
3		CENTERDRILL 12	1	1	120.000	6.000	90.0	
4		DRILL 10	1	1	120.000	5.000	118.0	
5		CUTTER 20	1	1	100.000	10.000		3
6		CUTTER 32	1	1	110.000	16.000		3
7								
8		FACEMILL 63	1	1	120.000	31.500		6
9		CUTTER 8	1	1	120.000	4.000		3
10		DRILL 8.5	1	1	120.000	4.250	118.0	
11		DRILL 12	1	1	100.000	6.000	118.0	
12		FACE_MILL_D50	1	1	100.000	22.500		5
13		SOLID_DRILL_30	1	1	110.000	15.000		
14		THREAD CUTTER	1	1	110.000	10.000		1
15		End mill 12	1	1	110.000	5.000		4
16								

Note: The display in the tool list can be changed to either diameter…. or "Radius" in the following general setting.

Tool Wear

Tools that are in use for long periods are subject to wear. You can measure this wear and enter it in the tool wear list

The control then taken this information into account when calculating the tool length or radius compensation. This ensures a consistent level of accuracy during workpiece machining

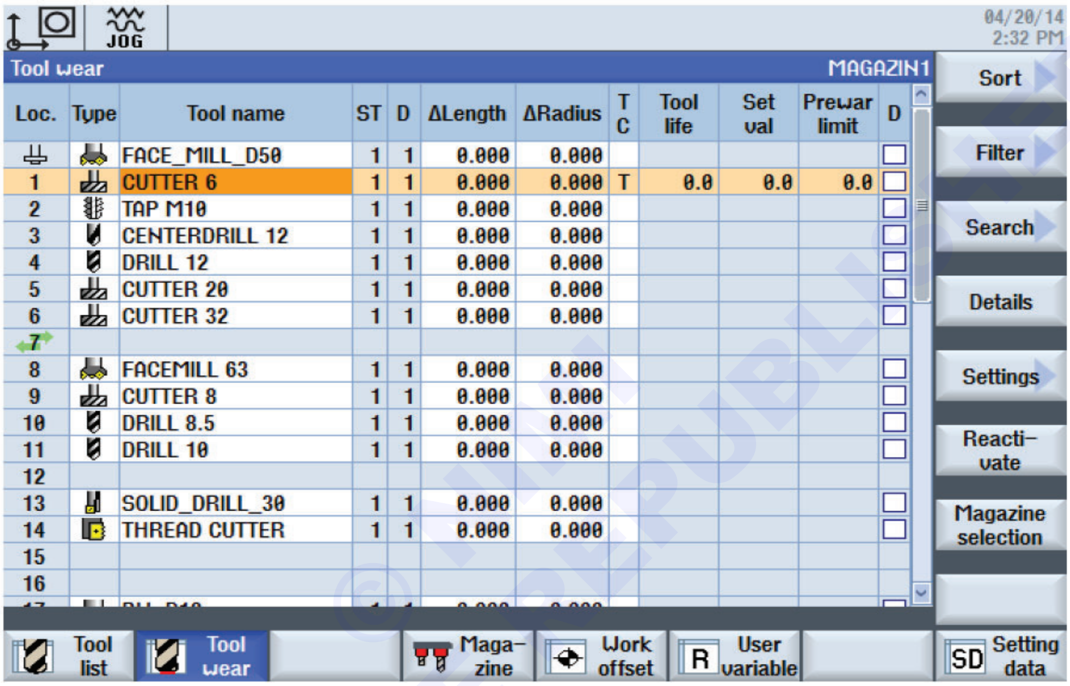
You can automatically monitor the tool working times via the workpiece count, tool life or wear.

In addition, you can disable tool when you no longer wish to use them

Selecting the function "Tool wear"

By pressing the HSK 2 "Tool wear" window opens (Fig 2)

Fig 2



Parameters for " Tool wear"

Parameter	Meaning
LOC.	Magazine/location number:
Type	Tool Type:
Tool Name	Tool Name:
ST	Replacement tool number:
D	Cutting Edge number:
Δ Length	Wear of length in Z direction
Δ Radius	Wear of radius or wear of diameter
Δ ∅	(refer to machine data setting)

For entering the values

Keep the pointer on the tool number

Enter the 'Z' difference in Δ length

Enter the radius difference in Δ radius

Note: if you make any change in program or parameter, press reset button

Restarting machine from sudden stoppage

Objectives : At the end of this exercise you shall be able to
- **restart the machine from sudden stoppage in Siemens control**
- **restart the machine from sudden stoppage in fanuc control.**

Job Sequence

TASK 1: **Resart in siemens control**

Executing specified blocks

> **If you would only like to perform a certain section of a program on the machine, then you do not need to start the program from the beginning. You can start the program from a specified program block in the following cases:**
> - **After you stop or interrupt the program execution**
> - **When you need to specify a target position, e.g. during remachining**

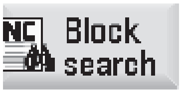
Operating sequence

Proceed as follows to start machining from the last interruption point in the program:

1 Select the machining operating area.

2 Switch to "AUTO" mode.

3 Press this softkey to open the block search window.



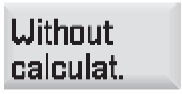
4 Press this softkey to load the interruption point, and the cursor moves to the beginning of the target block which is interrupted last time.

5 Press one of the following softkeys to set the condition for the block search:

- The program will continue from the line before the target block. The same calculations of the basic conditions (e.g. tool and cutting edge numbers, M functions, feedrate and spindle speed) in the previous blocks are carried out as during normal program operation, but the axes do not move.

- The program will continue from the target block containing the interruption point. The same calculations of the basic conditions in the previous blocks are carried out as during normal program operation, but the axes do not move.

- Block search without calculation of the basic conditions. All settings required for execution have to be programmed from the target block (e.g. feedrate, spindle speed, etc.).



6 Make sure the feedrate override is 0%.

7 Press this key on the MCP, and then an alarm 010208 appears for your confirmation whether to continue.

8 Press this key again to execute the program.

9 Turn the feedrate override switch on the MCP slowly to the desired value.

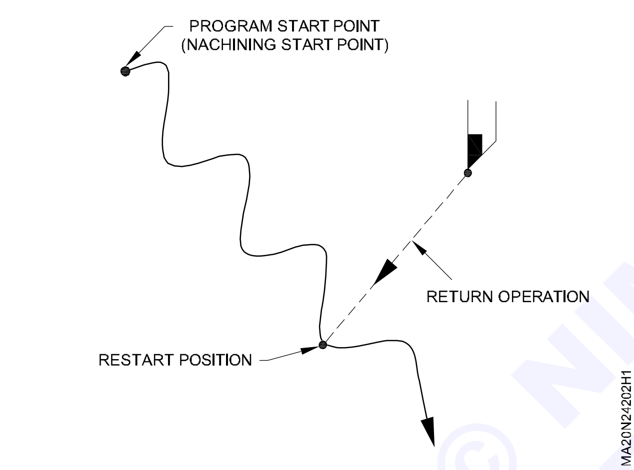
TASK 2: **Resart in fanuc control**

- **This function specifies sequence No of a block to be restarted when a tool is broken down or when it is desired to restart machining operation after a day off, and restarts the machining operation from that block. It can also be used as a high- speed program check function**
- **There are two restart methods: the P-type method Fig 1 and Q-typed method Fig 2.**

P Type - Operation can be restarted anywhere. This restart method is used when operation is stopped becase of a broken tool.

Fig 1

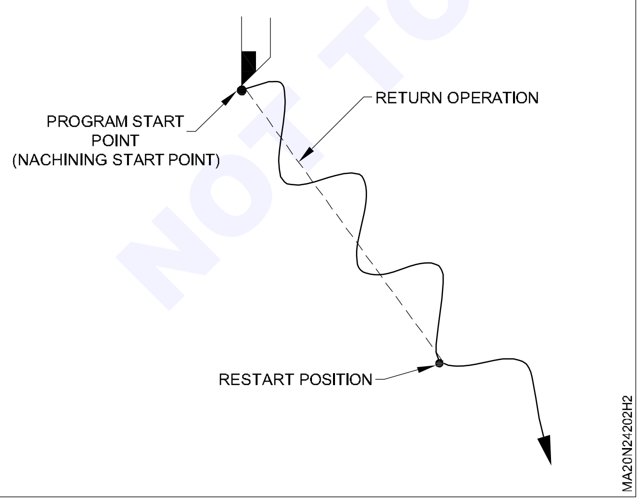
OPERATION CAN BE RESTARTED ANYWHERE. THIS RESTART METHOD IS USED WHEN OPERATION IS STOPPED BECAUSE OF A BROKEN TOOL



Q Type - Before operation can be restarted, the machine must be moved to the programmed start point (machining point)

Fig 2

BEFORE OPERATION CAN BE RESTARTED, THE MACHINE MUST BE MOVED TO THE PROGRAMMED START POINT (NACHINING START POINT)



Procedure for program restart by specifying a block number

Procedure 1

[P Type]

1 Retract the tool and replace it with a new one. When necessary, change the offset. (Go to step 2)

[Q type]

1 When the power is turned ON or emergency stop is released, perform all necessary operation at that time, including the reference position return

2 Move the machine manually to the program starting point (machine start point), and keep the modal data and coordinate system in the same conditions as at the machining start.

3 If necessary, modify the offset amount. (Go to step 2)

Procedure 2

[COMMON TO P TYPE/Q TYPE]

1 Turn the program restart switch on the machine operator's panel ON.

2 Press PROG Key to display the desired program

3 Find the program head. Press RESET key

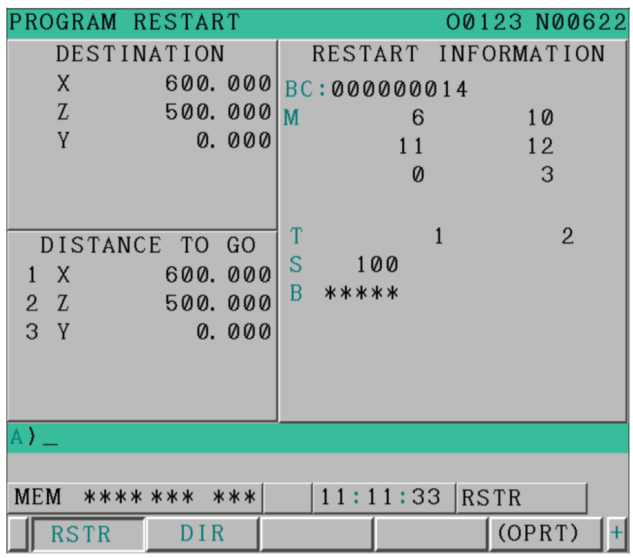
4 Enter the number of the block to be restarted then press the [P TYPE] or [Q TYPE] soft key. The block number cannot exceed eight digits

B xxxxxxx [Q TYPE] or [P TYPE]

→ Block number

5 The block number is searched for, and the program restart screen appears on the LCD display Fig 3

Fig 3



DESTINATION shows the position at which machining is to restart

1 DISTANCE TO GO shown the distance from the current tool position to the position where machining is to restart. A number to the left of each axis name indicates the order of axes (determined by parameter setting) along which the tool moves to the restart position

2 The coordinates and amount of travel for restarting the program can be displayed for upto four axes. For a path with five axes, when softkey [RSTR] is pressed again to display the fifth axis.

M: Upto 35 most recently specified M codes. The maximum number of displayed M codes differs depending on the size of display.

With 10.4 inch LCD/MDI panel up to :30 M codes

With 8.4 inch LCD/MDI panel up to : 6 M codes

T : Two recently specified T codes

S : Most recently specified S Code

B : Most recently specified B Code

Codes are displayed in the order in which they are specified. All codes are cleared by a program restart command or cycle start in the reset state

6 Turn the program re-start switch off. At this time, the figure at the left side of axis name DISTANCE TO GO blinks.

7 Check the screen for the M,S,T, and B codes to be executed. If they are found, enter in the MDI mode, then execute the M, S, T, and B functions. After execution, restore the previous mode. These codes are not displayed on the program restart screen

8 Check that the distance indicated under DISTANCE TO GO is correct. Also check whether there is the possibility that the tool might hit a work piece or other objects when it moves to the machining restart position. If such a possibility exists, move the tool manually to a position from which the tool can move to machining restart positon without encountering any obstacles.

9 Press the cycle start button. The tool moves to the machining restart position at the dry run feed rate sequentially along axes in the order specified by parameter No. 7310 settings. Machining is then restarted.

Program transfer to machine - USB and flash drive

Objectives : At the end of this exercise you shall be able to
- **transfer the CNC part program from USB/flash drive to CNC machine in fanuc control**
- **transfer the CNC part program from USB/Flash drive to CNC machine in Siemens control.**

Job Sequence

TASK 1: **Program input through USB/Flash drive**

Fanuc I series control

(Latest control have the facility to use USB/Flash drive)

- Make sure that you are in edit mode.
- Press MDI mode
- Press offset and setting page on the softkey to collect the setting page.
- In setting page i/o channel set to use the USB channel.
- Insert the USB stick then select the edit mode
- Click on program - that display the program directory screen
- Click directory and operation then > (arrowkey) then notice it says devise USB
- Select USB memory and it automatically reads what's on the card
- Now by pressing  down words arrow select the program that you want to load it into your CNC control.
- Click on the right arrow key until you see the file input file/output file.
- Click on input now.

Notice the program numbers, type the exact name of the program on to the field and then select a program number O1 and click on set and then press execute and then it will load that program under number one and can see it's active at the top of the screen.

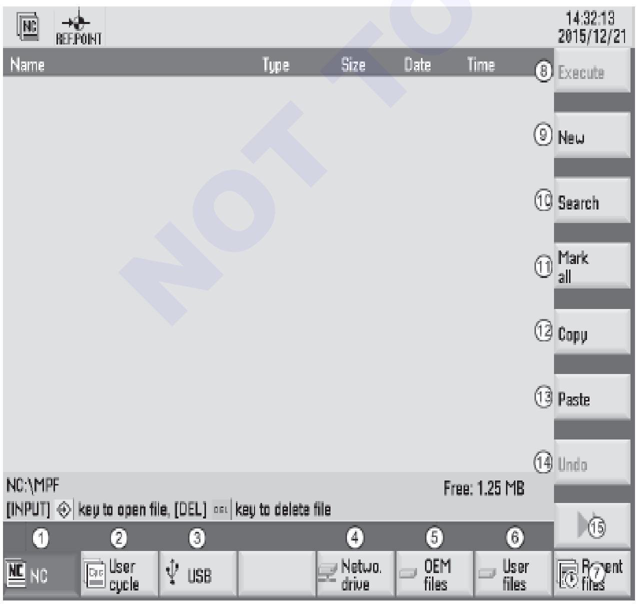
— — — — — — — — —

TASK 2: **Program transfer from USB/Flash drive to CNC machine (Siemens Control)**

Softkey functions

Pressing this key on PPU allows you to open the following window Fig 1 

Fig 1



1 Stores the NC programs for subsequent operations
2 Manages and transfers the manufacturer cycles
3 Reads in/out files via the USB drive and executes part programs from the external storage media
4 This soft key is valid on the PPU160.2 only and displays as follows **... RS232**
5 Reads in/out files via Ethernet interface and executes part programs from a computer
6 Backs up manufacturer files
7 Backs up user files
8 Shows the recently accessed files
9 Executes the selected file. No editing is allowed in the execution process.
10 Creates new files or directions
11 Searches for files
12 Selects all files for the subsequent operations
13 Copies the selected files(s) to the clipboard
14 Pastes the selected files(s) from the clipboard to the current directory
15 Restores the deleted file(s)

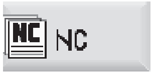
16 Opens the lower-level ment for more options:

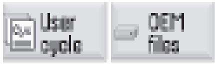
- Rename the part programs
- Cut the part programs

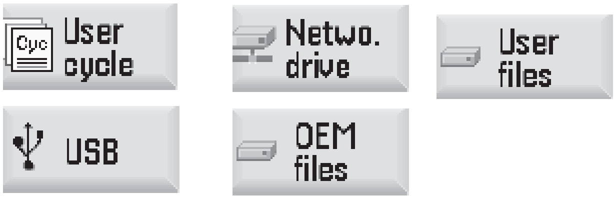
Note Softkeys 2 and 6 are visible only with the manufacturer password

Searching for programs

1 Select the program management operating area 

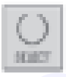
2 Select he storage directory in which you wish to perform the search 

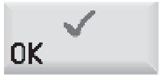
Note: The following two folders are visible only with the manufacturer password: 



3 Press this vertical softkey to open the search window. 

4 Enter the complete name with extension of the program file to be searched in the first input field in the search window. To narrow your search, you can enter the desired text in the second field

5 Use this key to choose whether to include subordinate folders or observe upper/lower case. 

6 Press this softkey to start the search 

Transferring from external (through USB interface)

Prerequsite: A USB memory stick (which includes the part program to be transferred) is inserted in the front USB interface of the PPU

Proceed as follows to transfer a part program from external through the USB interface

1 Select the program management operating area 

2 Press this softkey to enter the USB directory 

3 Select the program file you desire to transfer

4 Press this softkey to copy the file to the buffer memory on the control system 

5 Enter the program directory 

6 Press this soft key to past the copied file into the program directory 

Programmable work zero and tool geometry

Objectives : At the end of this exercise you shall be able to
- **modify the program to effect the work Zero through program**
- **modify the program to effect the tool offset through program.**

Job Sequence

TASK 1:

- Prepare a drilling program to a particular depth (blind hold)
- Execute the program and check the depth
- Increase the tool offset by 1 mm through program and execute it.
- Check the depth. It should be one mm more than the programmed value.

Similarly decrease the depth by 1 mm and repeat the same and measure it and understand the logic of programmable tool offset both in Siemens and fanuc control.

TASK 2:

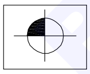
Similarly develop any simple program and shift the work zero value in X and Y, to a new location program and verify it effects.

Skill Sequence

Merging work zero with program zero points/programmable work offset

Objective: This shall help you to
- **prepare the program with programmable work offset in Fanuc and Siemens control**

Merging work zero with program zero point (Siemens Control)

Machine co-ordinate system MCS 

Machine co-ordinate system comprises all the physically existing axes.

Reference point and tool and pallet point (fixed machine points) are defined in machine co-ordinate system.

If the programming is performed directly in the machine co-ordinate system (G53) the physical axes of the machine responded directly. (Fig 1)

Settable zero system (SZS)

The settable zero offset system results from the machine co-ordinate system through settable zero offset (G54 to G57 and G505 to G509) in all axes. (Fig 2)

Work piece co-ordinate system (WCS)

The geometry of the work piece is described in the work piece co-ordinate system. In other words the data in the NC program referes to the work piece co-ordinate system

Fig 1

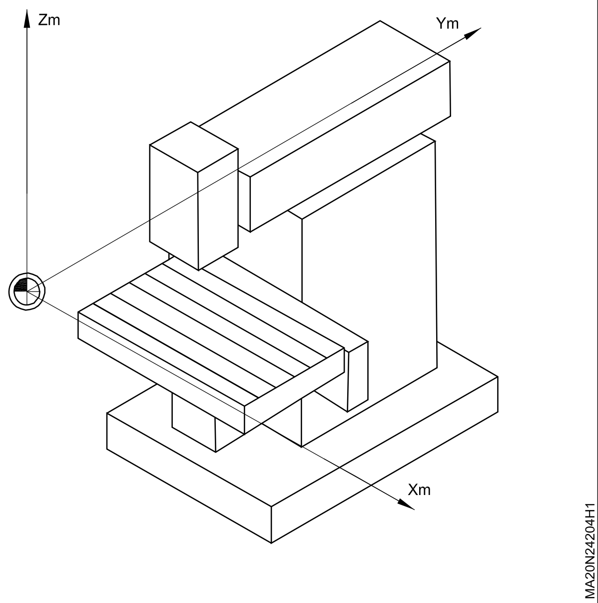
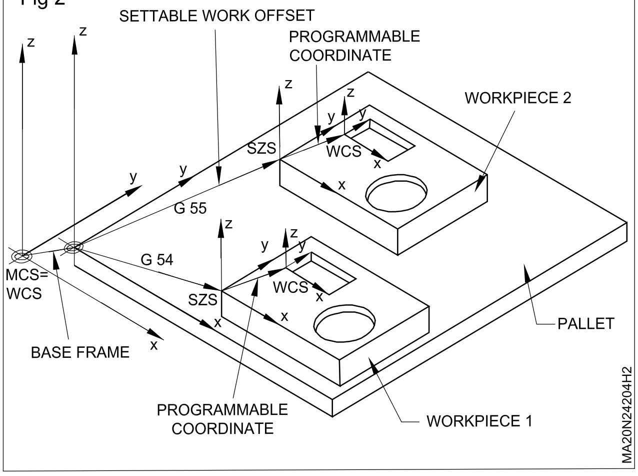
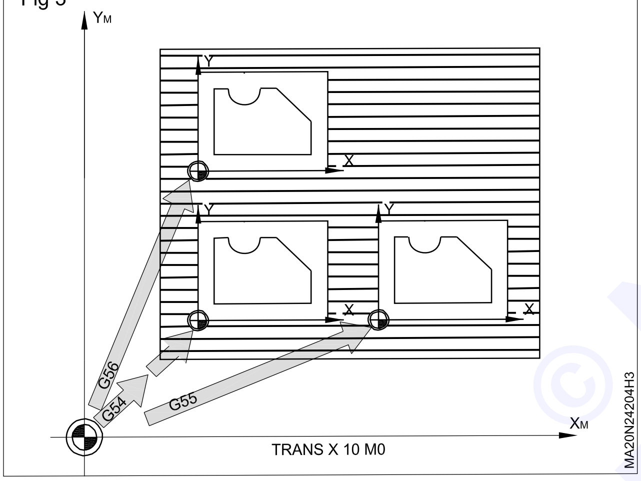


Fig 2



Three workpieces that are arranged on a pallet in accordance with the zero offset values G54 to G56 are to be machined in succession. The machining sequence is programmed in subprogram L47. (Fig 3)

Fig 3



Settable Zero offset (G54 to G57, G505 to G589, G53, G500, SUPA, G153)

Syntax

Activating settable zero offset;

G54

...

G57

G505

...

G599

Deactivating settable zero offset;

G500

G53

G153

SUPA

Meaning

G54 to G57;

Call of the 1st to 4th settable zero offset (ZO)

Call of the 5th to 99th settable zero offset

Deactivation of the current settable zero offset

G500= zero frame; (default setting;contains no offset, rotation, mirroring or scaling)	Deactivation of the settable until the next call, activation of the entire basic frame
G500 not equal to 0;	Activation of the first settable zero off set and activation of the entire basic frame or possibly a modified basic frame is activated.

G53: G53 supresses the settable zero offset and the programmable to offset non-modally

G153: G153 has the same effect as G53 and also suppresses the entire tool frame

SUPA: SUPA has he same effect as G153 and also suppresses;

- Handwheel offsets (DRF)
- Overlaid movements
- External zero offset
- PRESET offset

Program	Comment
N10 G0 G90 X10 Y10 F500 T1	; Approach
N20 G54 S1000 M3	;Call of the first Z0 spindle clockwise
N30 L47	; Program pass as subprogram
N40 G55 G0 X0, Y0	; Call of the second X0 Y0
N50 L47	; Program pass as subprogram
N60 G56 G0 X0 Y0	; Call of the third
N70 L47	: program pass as subprogram
N80 G53 X200 Z200	: Suppress zero offset, end of the program
Y300	
M30	

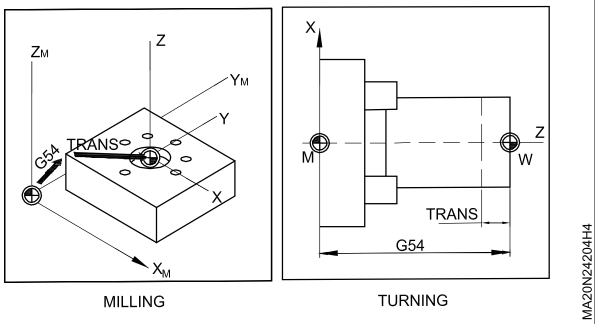
Programmable co-ordinate transformation (Frames)

- It is useful or necessary with in a NC program, to move the originally selected work piece coordinate system (settable zero offset, to another position and, it required to rotate it, mirror it and or scale it. This is performed using programmable co-ordinate system

Zero offset (TRANS, ATRANS) (Fig 4)

TRANS/ATRANS can be used to program zero offsets for all path and positioning axes in the direction of the axis specified in each case. This means that it possible to work with changing zero points, e.g during repetitive machining operations at different workpiece positions

Fig 4



Syntax

TRANS X… Y… Z…

ATRANS X… Y… Z…

Note: Each frame operation is programmed in a separate NC block

Fanuc - Programmable co-ordinate system

When a program is created in a work piece coordinate system, a child work piece coordinate system may be set for easier programming. Such a child coordinate system is referred to as a local coordinate system

Format

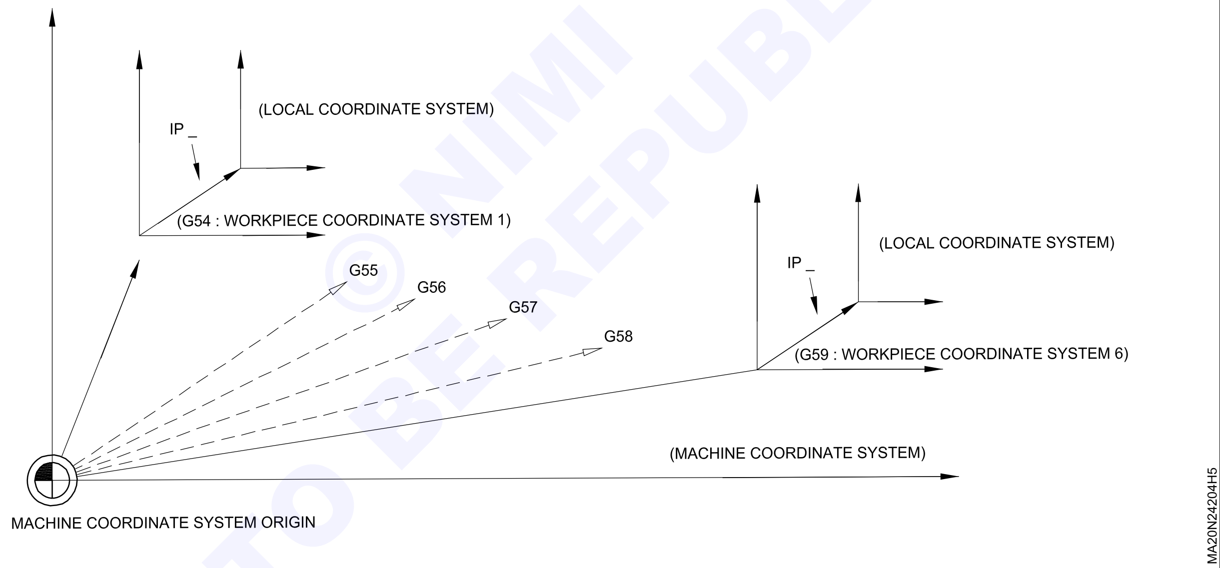
G52IP: Setting the local coordinate system

G52 IP 0; Cancelling of the local coordinate system

IP; Origin of the local coordinate system

Explanations: By specifying G52IP…; a local coordinate system can be set in all the workpiece system (G54 to G59). The origin of each local coordinate system is set at the position specified by IP… in the workplace coordinate system (Fig 5)

Fig 5



Setting the local coordinate system

When a local coordinate system is set, the move commands in absolute mode (G90), which is subsequently commanded, are the coordinate values in the local coordinate system. The local coordinate system can be changed by specifying the G52 command with the zero point of a new local coordinate system in the workpiece coordinate system.

To cancel the local coordinate system, and specify the coordinate value in work piece coordinate system match the zero point of the local coordinate system with that of the workpiece coordinate system

WARNING

1 When an axis returns to the reference point by the manual reference point return function, the zero point of the local coordinate system of the axis matches that of the work co-ordinate system. The same is true when the following command is issued; G52 X0, Y0, Z0.

2 The local coordinate system setting does not change the workpiece and machine coordi-nate

3 The local coordinate system is cancelled when the reset operation is performed.

4 When setting a workpiece coordinate system with the G92 command, the local coordinate systems are cancelled

5 G52 cancels the offset temporarily in cutter compensation

6 Command a move immediately after G52 block in the absolute mode.

7 The option for the workpiece coordinate system (G54 to G59) is required

Programmable tool offset

Objective: This shall help you to

- **prepare the program in Siemen and fanuc control to alter the tool offset values through program.**

Programmable tool offset

The programmable tool offset command is to modify the effective tool length or effective tool radius in the NC program without changing the tool offset in the compensation memory

The programmed offsets are deleted again at the end of the program

Siemens Control

Command

TOFFL/TOFF = Tool length offset

TOFFR = Tool radius offset

Syntax for TOFFL/TOFF

TOFFL = < values>

TOFFL (1) = < value>

TOFFL (2) = <Value>

TOFFL 3 = <value>

Syntax for TOFFR

TOFFR = <value>

Example: Positive tool length offset

- The active tool is a drill with length L1=100 mm
- The active plane is G17, the drill points in a 'Z' direction
- The effective drill length is to be increased by 1 mm. The following are the program of the tool length offset.

  TOFFL = 1

  Or

  TOFFL [1] = 1

  Or

  TOFFL [Z] = 1

Example for negative tool length offset.

The effective tool length to be reduced by 1 mm

Program

TOFFL = -1

TOFFL [1] = -1

Or

TOFFL [Z] = -1

Program code	Comment
N10 STC_DPI [1,1] =120	
N20 STC_ DP3 [1,1] =100	; Tool change L1 = 100 mm
N30 T1 D1 G17	
N40 TOFF[Z] = 1.0	; Offset in Z direction (corresponds to L1 for G17)
N50 G0 X0 Y0 Z0	; Machine axis position X0 Y100 Z101.
N60 G18 G0 X0Y0Z0	; Machine axis position X0 Y100 Z1.
N70 G17	
N80 TOFFL = 1.0	; Offset in L1 direction (corresponds to Z for G17)
N90 G0 X0 Y0 Z0	; Machine axis position X0 Y0 Z101.
N100 G 18 G0 X0 Y0 Z0	; Machine axis position X0 Y101 Z0.

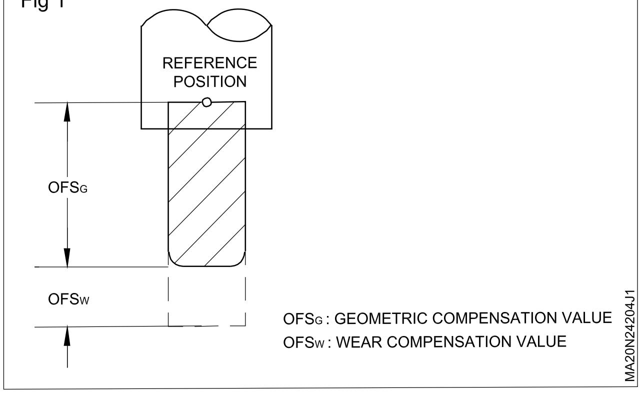
In this example, the offset of 1 mm in the Z axis is retained when changing to G18 in block N60; the effective tool length in the Y axis is unchanged tool length of 100 mm.

However, in block N100 the offset is effective in Y axis when changing to G18 as it was assigned to tool length L1 in the programming and this length component is effective in the Y axis with G18.

Fanuc Tool Compensation

Tool compensation values include tool geometry compensation value and tool wear composition (Fig 1)

Fig 1



Geometric compensation and wear compensation

Tool compensation values can be entered into CNC memory from the CRT/MDI panel or from a program. A tool compensation values is selected from the CNC memory when the corresponding code is specified after address H or D in a program. The value is used for tool length compensation, cutter compensation the tool offset.

Explanations

Valid range of tool Compensation values

Table 1 The valid input range of tool compensation value

Increment system	Metric input	Inch input
IS - B	0 to ±999.999mm	0 to ±99.9999inch
IS - C	0 to ±999.9999mm	0 to ±99.9999inch

Number of tool compensation values and the addresses to be specified

The memory can hold 32,64,99 or 200 tool compensation value (option)

Address D or H is used in the program. The address used depends on which of the following functions is used: tool length compensation tool offset cutter compensation or cutter compensation C. The range of the number that comes after the address (D or H) depends on the number of tool compensation values : 0 to 32, 0to 64,0 to 99,or … to 200.

Tool compensation memory and the tool compensation value to be entered

Tool compensation memory A or B can be used.

The tool compensation memory determines the tool compensation values that are entered (set) (Table 2)

Table 2 - Setting contents tool compensation memory and tool compensation value

Tool compensation memory	Tool compensation value
A	The tool compensation corresponding to the tool compensation number is used
B	Th tool geometry compensation, plus tool wear compensation, both corresponding to the tool compensation number, are used

Format

The programming format depends on which tool compensation memory

Input of tool compensation value by programming

Table 3 Setting format for tool compensation memory and tool compensation value

Tool compensation memory		Format
A	Tool compensation value	G10P_R_;
B	Geometry compensation value	G10L10P_R_;
	Wear compensation value	G10L11P_R_;

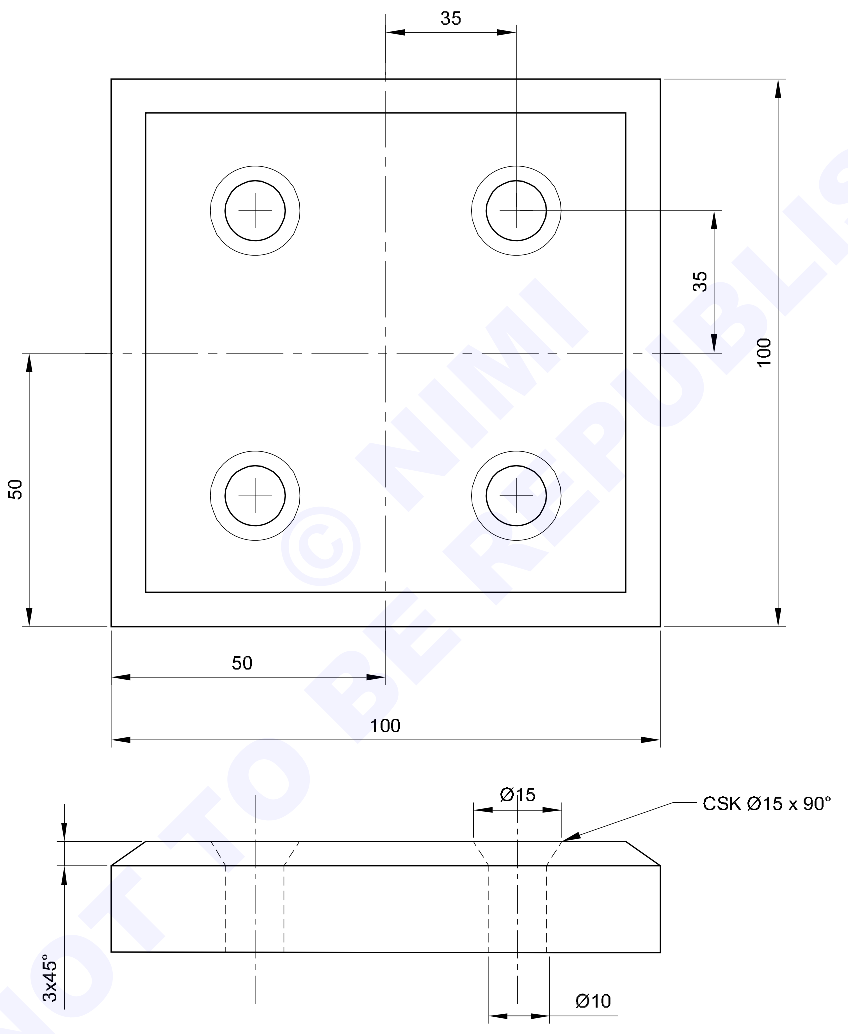
P: Number of tool compensation

R: Tool compensation value in the absolute command (G90) mode value to be added to the speified tool compensation value in the incremental command (G91) mode (the sum is also a tool compensation value).

> **Note: To provide compatibility with the format of older CNC programs, the system allows L1 to be specified instead of L11.**

Chamfering and counter, sinking

Objectives : At the end of this exercise you shall be able to
- **write a part program for Chamfering as per drawing**
- **write the program for drilling and chamfering as per drawing**
- **verify the program**
- **machine the work piece in auto mode.**



1	PRE MACHINED 100 x 100 x 25		ALUMINIUM			2.4.205
NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.

SCALE 1:1	**CHAMFERING AND COUNTER SINKING**	DEVIATIONS	TIME
		CODE NO. MA20N24205E1	

Job Sequence

Procedure

MSG ("FACE MILLING")

N3 G17G71G90G40;

N4 T="T1"M6; FACE MILL CUTTER DIA 50

N5 D1

N6 M03S600

N7 G0G90G54X0Y0

N8 G0Z20

N9 M07

N10 CYCLE61 (10,0,5,-2,-50,-50,50,50,1,75,0,200, 31,0,1,11010);

N11G0Z20

N12M9

N13 G75Z0

N14 G75X0Y0

N15 MSG ("DEEP HOLE DRILLINGDIA 8.5 MM X THRU DEEP X 4 PLACES")

N16 T = "T3" M6' DRILL DIA 8.5

N17D1

N18 M03S1000

N19 G0G90G54X0Y0

N20 G0Z20

N21 M07

N22 M CALL CYCLE 83 (10,0,5,-32,,-10,,25,0,.6,0.6,25,0,0,1,2,1,4,0.6,1.6,0,1,11221112)

N23 CYCLE802 (111111111,111111111,40,40,-40,40,-40,-40,40,-40,,,,,0,0,1);

N24 MCALL

N25 G0Z20

N26 M9

N27 G75Z0

N28 G75X0Y0

N29 MSG ("CHAMFER DIA 14 X 90 DEG X 4 PLACES")

N30 T = "T4" M6; CHAMFER DIA 16

N31 D1

N32 M03S1000

N33 G0G90G54X0Y0

N34 G0Z20

N35 M07

N36 MCALL CYCLE81 (25,-2,20,-13,,0.6,0,1,12)

N37 CHCLE802 (111111111,111111111,40,40,-40,40,-40,-40,40,-40,,,,,0,0,1);

N38 MCALL

N39 G0Z20

N40 M9

N41 G75Z0

N42 G75 X0Y0

N43 MSG ("CHAMFERING THE PROFILE BY 3X45DEG")

N44 T = "T4" M6; CHAMFER DIA 16

N45 D1

N46 M03S1000

N47 G0G90G54GX0Y0

N48 G0Z20

N49 M07

N50 CYCLE 76 (10,-2,1,,5,100,100,0,0,0,0,0.5,0.1,0.1,100,0.1,0,5,12,6,3,-6,1100,10001,1)

N51 G0Z20

N5 2M9

N53 G75Z0

N54 G75X0Y0

N55 M30

FACE MILLING CYCLE:

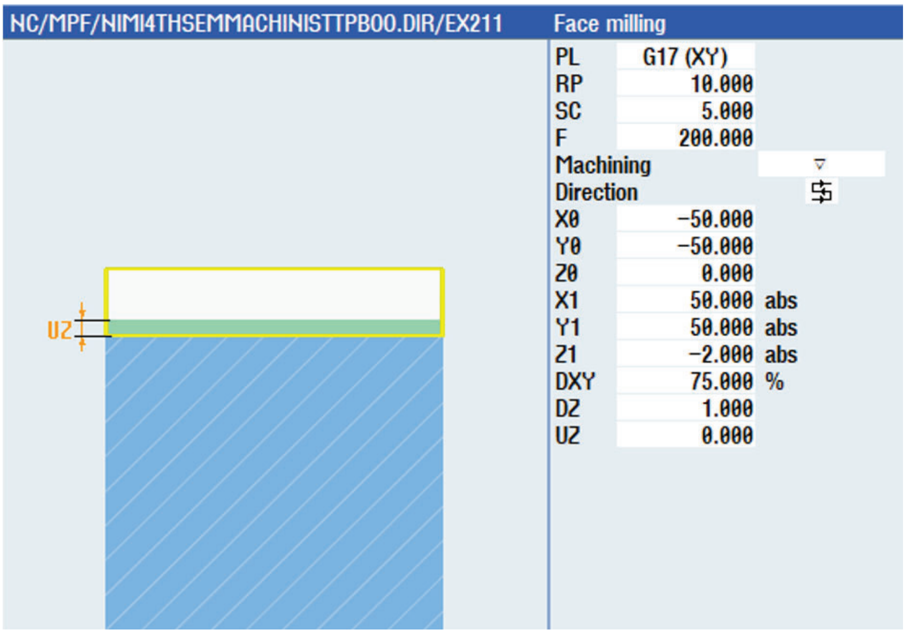
CYCLE 61 (10,0,5,-2,-50,-50,50,50,1,75,0, 200,31,0, 1,11010);

1 Click on the “Milling option”

2 Click on the “Face milling option”

3 In the parameter window enter the following parameters as per the Fig 1.

Fig 1



Drilling cycle with positions

MCALL CYCLE83 (10,0,5,-38,,-10,,25,0.6,0.6,25,0,0,1.2,1.4,0.6,1.6,0,1,11221112)

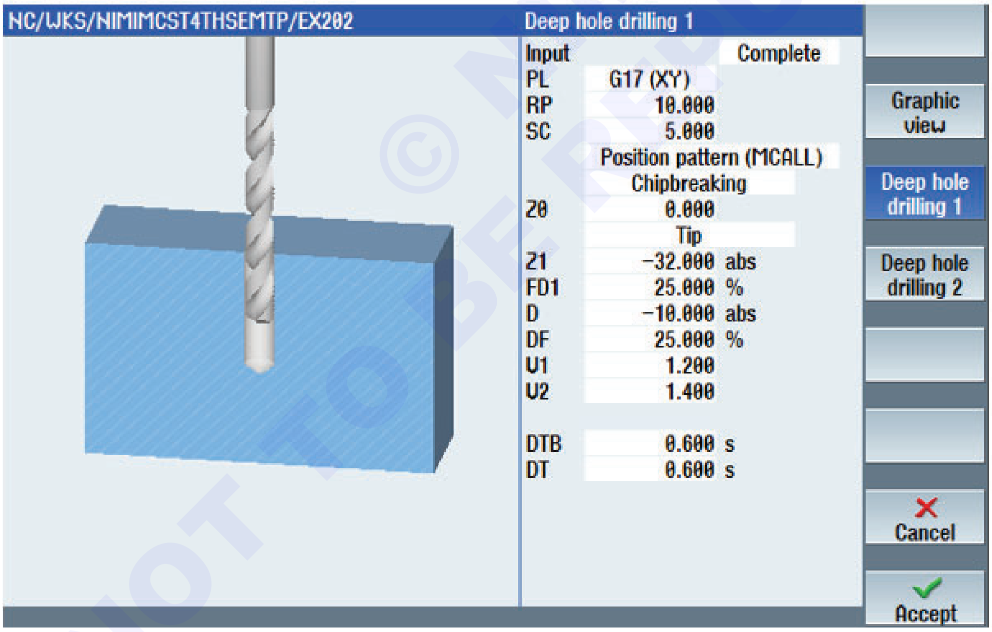
CYCLE802(11111111,11111111,40,40,-40,40,-40,-40,40,-40,,,,,0,0,1);

1 Click on the 'Drilling “option"

2 Click on the 'Deep hole drilling "Option”.

3 Click on the "Deep hole drilling1 "option” & fill the parameters as shown in the Fig 2.

Fig 2

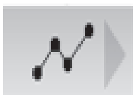


4 Then press "Accept" the following programming line appears in the program

5 MCALL CYCKE 82 (10,5,1,-32,,0.6,0,10001,12)'

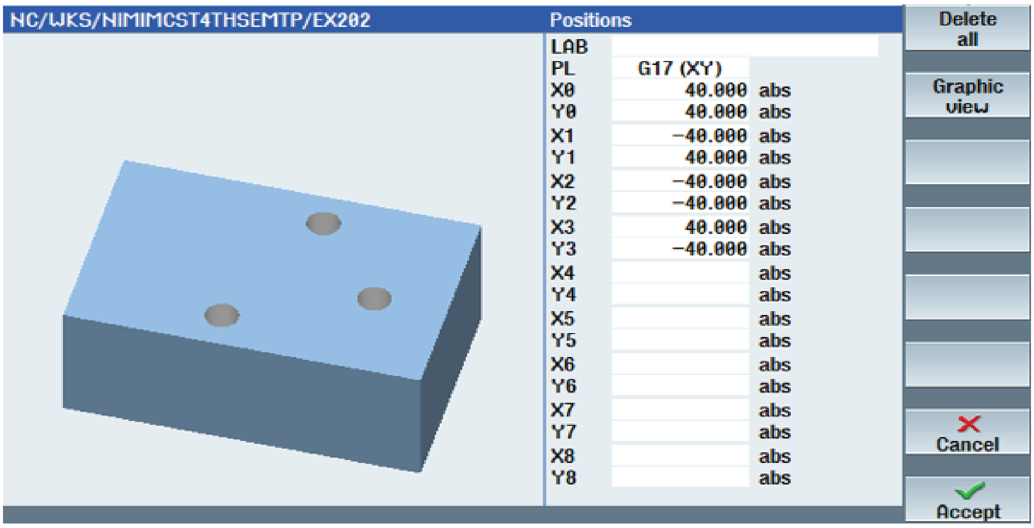
6 Click on the drilling option

7 Click on the positions option

8 Click on the option 

9 In the position window fill the co-ordinates of the holes as shown in the Fig 3.

Fig 3



10 Then press "Accept" the following programing line appears in the program

CYCLE802 (111111111,111111111,40,40,-40,40,-40,-40,40,-40,,,,,0,0,1)

Chamfering drilled holes with positions

MCALL CYCLE81 (25, -2,20,-13,,0.6,0,1,12)

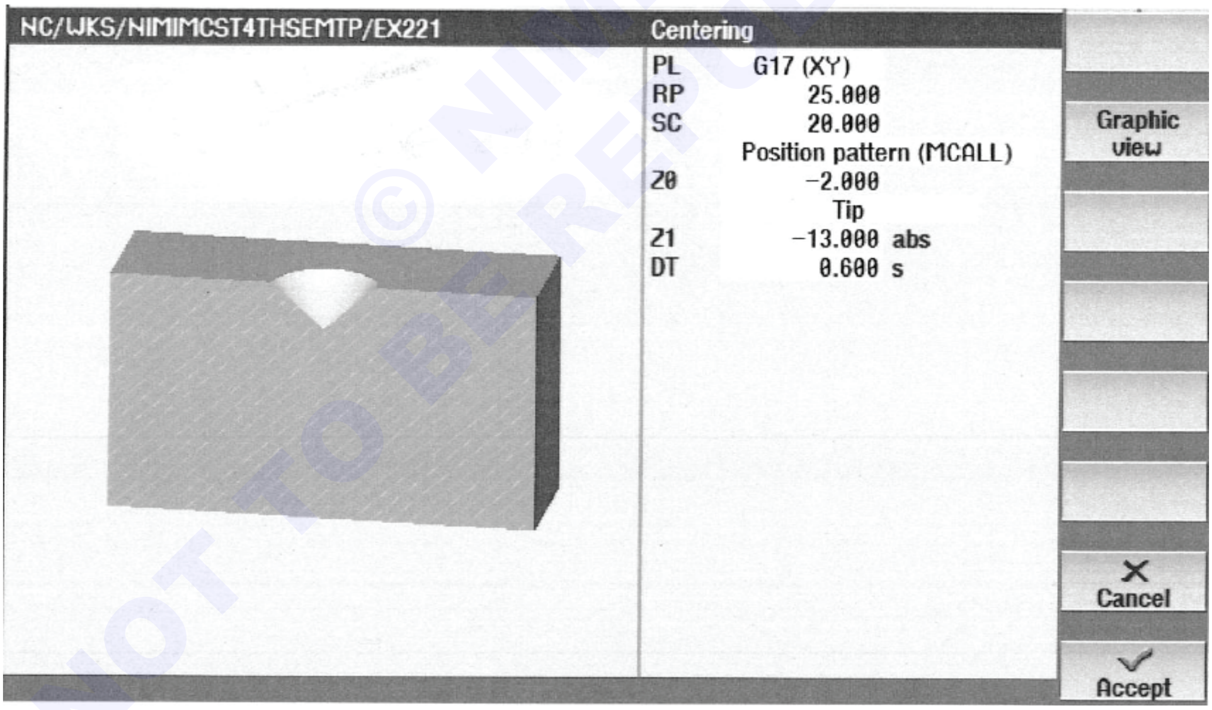
CYCLE802 (111111111,111111111,40,40,-40,40,-40,-40,40,-40,,,,,0,0,1)

11 Click on the "Drilling" Option.

12 Click on the "Centering" optioin

13 In the option window fill the parameters as shown in the Fig 4.

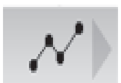
Fig 4



14 Then press "Accept" the following programming line appears in the program MCALL CYCLE81 (25,-2,20,-13,,0.6,1,1,12);

15 Click the drilling option

16 Click on the positions option.

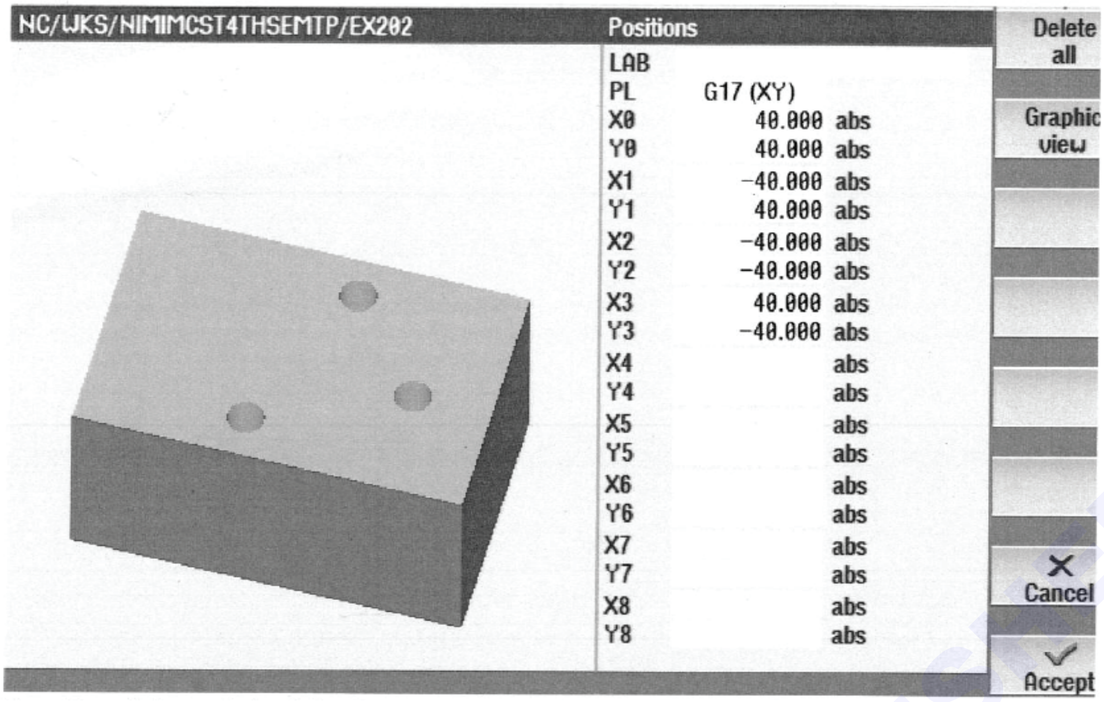
17 Click on the option 

18 In the position window fill the co-ordinates of the holes as shown in the Fig 5.

19 Then Press “Accept” the following programming line appears in the program

CYCLE802 (111111111,11111111,40,40,-40,40,-40,-40,40,-40….0,0,1);

Fig 5



CHAMFERING EDGES

CYCLE76 (10,-2,1,,5,100,100,0,0,0,0,0.5,0.1,0.1,100,00.1,0,5,2,6,3,-6,1100,10001,1)

20 Click on the "Milling option

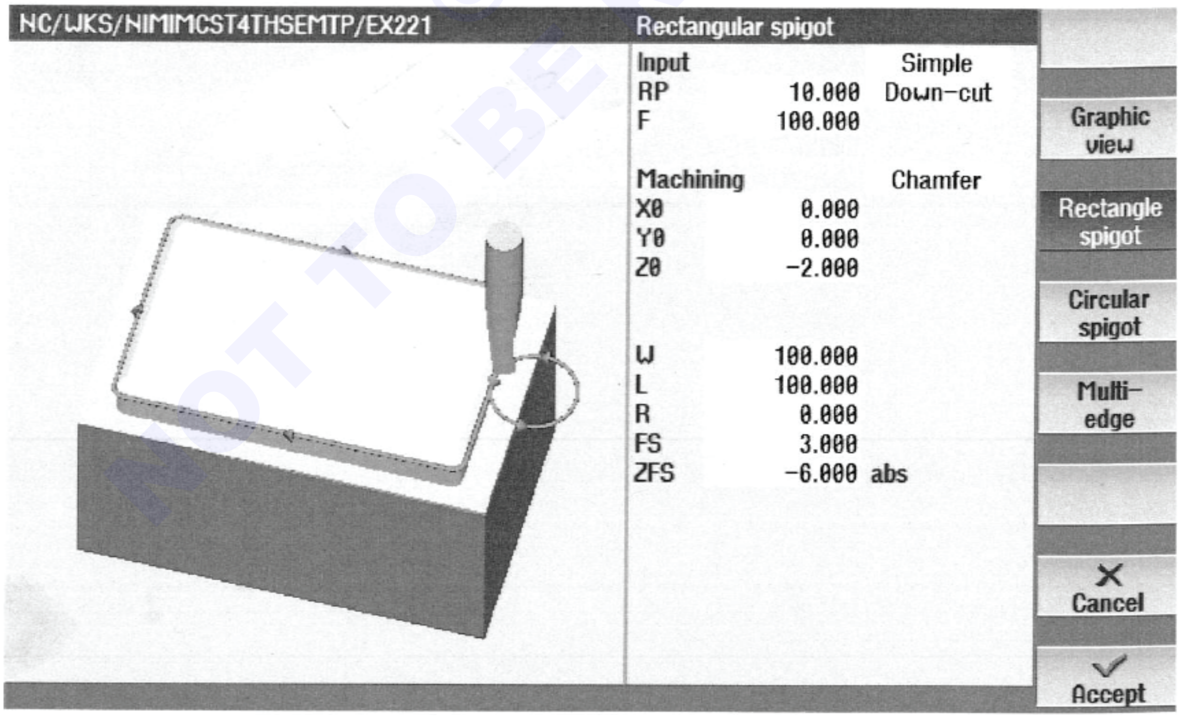
21 Click on the "Rectangle spigot" option

22 In the option window fill the parameters as shown in the Fig 6.

23 Then press "Accept" the following programming line appears in the program

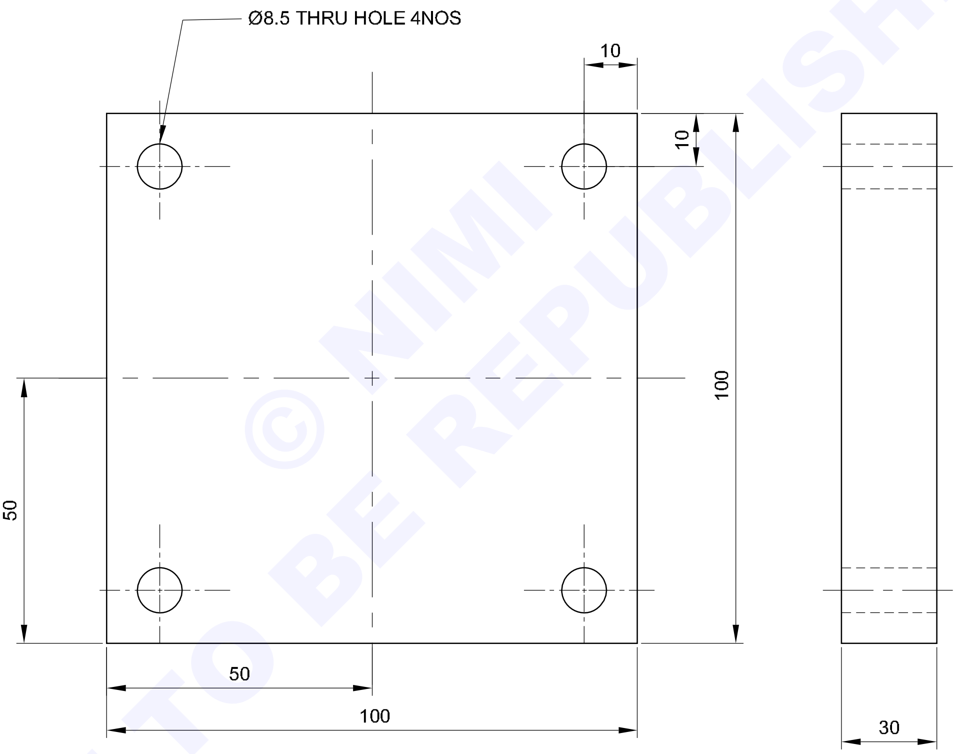
CYCLE 76 (10,-2,1,,5,100,100,0,0,0,0,0.5,0.1,0.1,100,0.1,0,5,12,6,3,-6,1100,10001,1)

Fig 6



Deep hole drilling

Objectives : At the end of this exercise you shall be able to
- **write the drilling program using G83 cycle**
- **verify the program**
- **run the program in auto mode to drill the workpiece.**



1	PRE MACHINED 100 X 100 X 30 mm		ALUMINIUM			2.4.206
NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.
SCALE 1:1	DEEP HOLE DRILLING				DEVIATIONS	TIME
					CODE NO. MA20N24206E1	

Job Sequence

Procedure

MSG (" FACE MILLING")

G17G71G90G40

TIM6; FACE MILL CUTTER DIA 50

D1;

M03S600;

G0G90G54X0Y0;

G0Z20;

M07;

CYCLE61 (10,0,5,-2,-50,-50,50,50,1,75,0, 200,31,0,1,11010);

G0Z20;

M9;

G75Z0;

G75X0Y0;

MSG ("PRE DRILLING HOLE DIA 10.5MM X 22 MM DEEP X 4 PLACES");

T3M6; DRILL DIA 10.5;

D1;

M03S1000;

G0G90G54X0Y0;

G0Z20;

M07;

MCALL CYCLE82 (10,2,1,-22,,0.6,0,10001,12);

CYCLE802 (111111111,111111111,25,0,0,25,-25,0,0,-25,,,,,0,0,1);

MCALL

G0Z20

M9

G75Z0

G75X0Y0

Face milling cycle

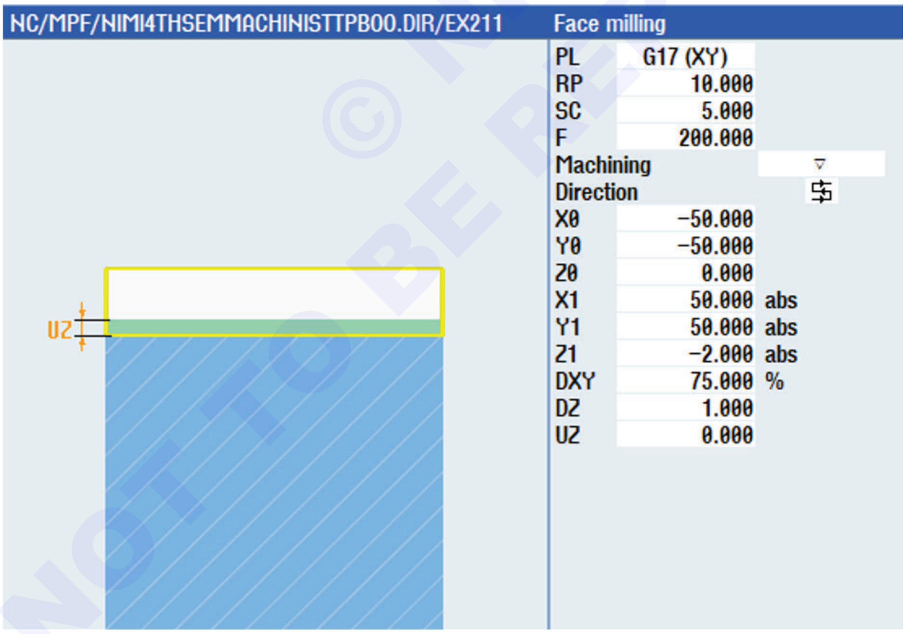
CYCLE61 (10,0,5,-2,-50,-50,50,50,1,75,0,200,31,0,1,11010);

1 CLICK ON THE "MILLING" OPTION

2 CLICK ON THE "FACE MILLING" OPTION

3 In the parameter window enter the following parameters as per the Fig 1.

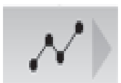
Fig 1



Drilling cycle with positions

MCALL CYCLE83 (10,0,5,-38,,10,,25,0.6,0.6,25,0,0,1.2,1.4,0.6,1.6,0,1,11221112)

CYCLE802 (111111111,111111111,40,40,-40,40,-40,-40,40,-40,,,,,0,0,1);

1 Click on the "Drilling Option". 

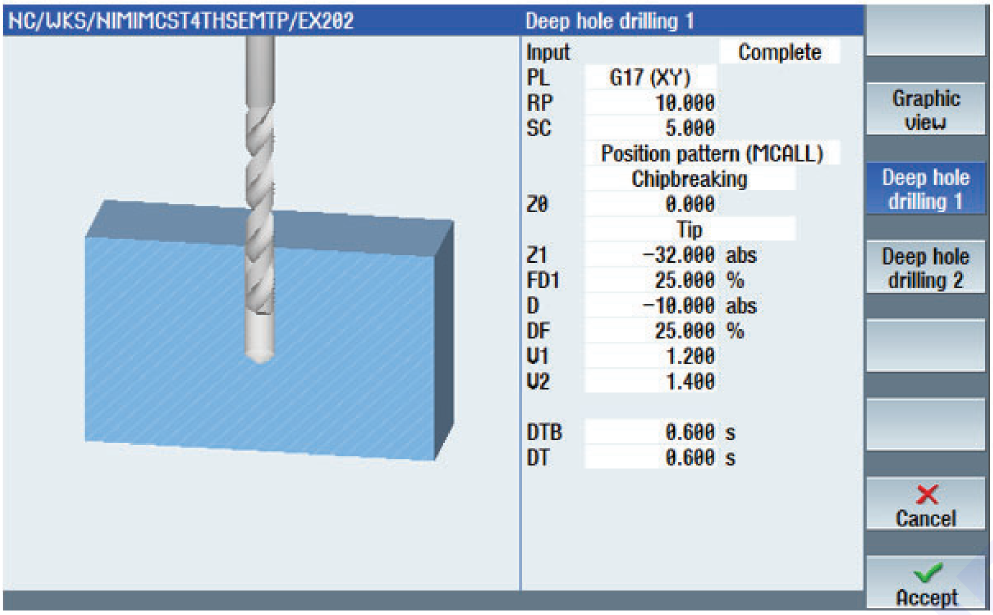
2 Click on the "Deep Hole Drilling" option

3 Click on the "Deep hole Drilling 1 "option & fill the parameters as shown in the Fig 2.

4 Then Press "Accept" the following programming line appears in the program

5 MCALL CYCLE82 (10,5,1,-32,,0.6,0,10001,12);

Fig 2



6 Click on the Drilling Option

7 Click on the Positions option

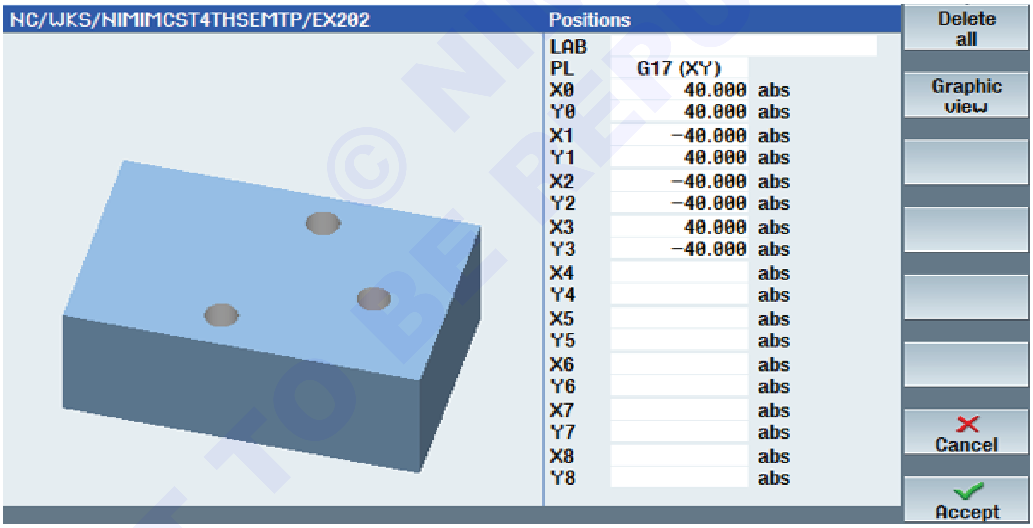
8 Click on the Option

9 In the position window fill the co-ordinates of the holes as shown in Fig 3

10 Then press "Accept" the following programming line appears in the program.

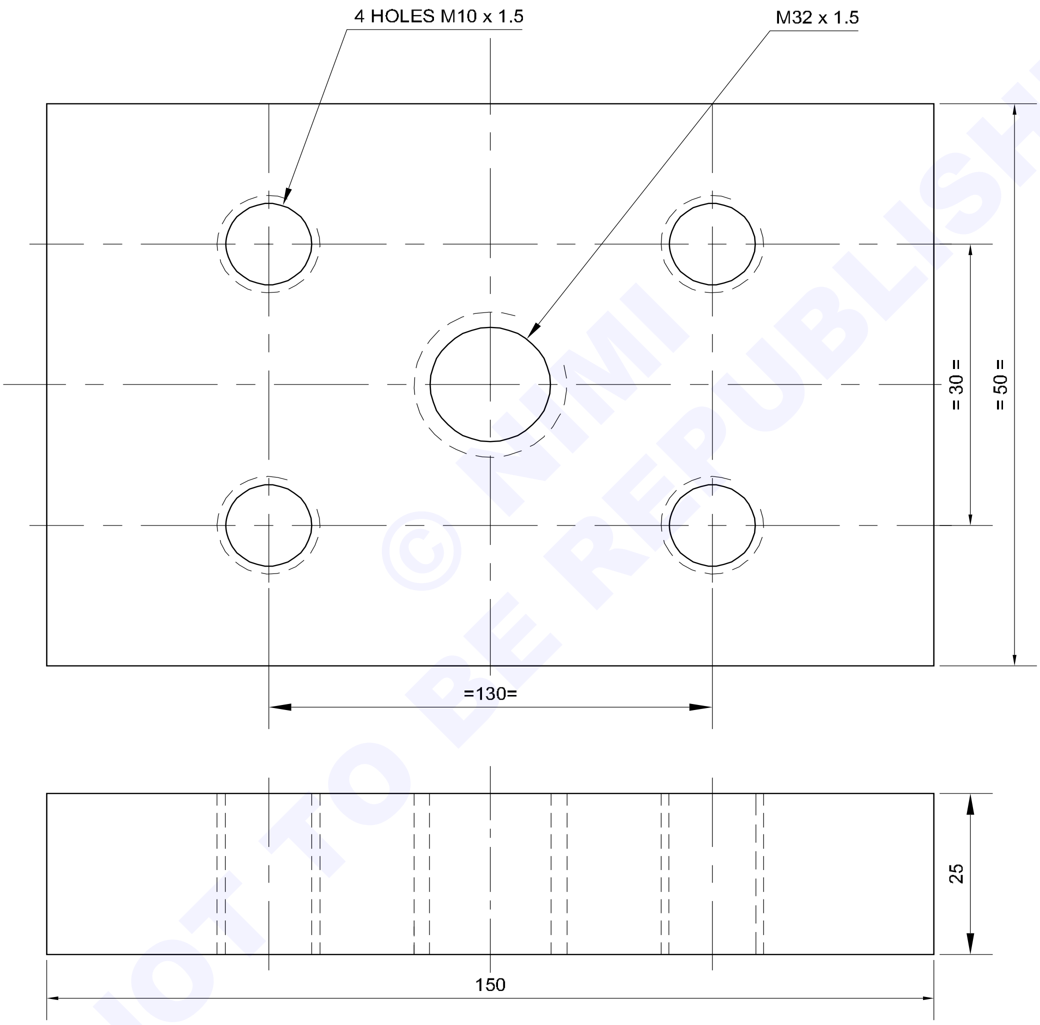
CYCLE802 (111111111,111111111,0,0,30,30,-30,30,-30,-30,30,-30,,,,,0,0,1)

Fig 3



Perform thread milling and tapping (G84) (Siemens)

Objectives : At the end of this exercise you shall be able to
- **prepare the program for centre drilling, drilling and tapping using build in cycles in Siemens -control**
- **prepare the program for circular pocket to core diameter of the thread**
- **prepare internal threading program using threading cycle**
- **execute the program.**

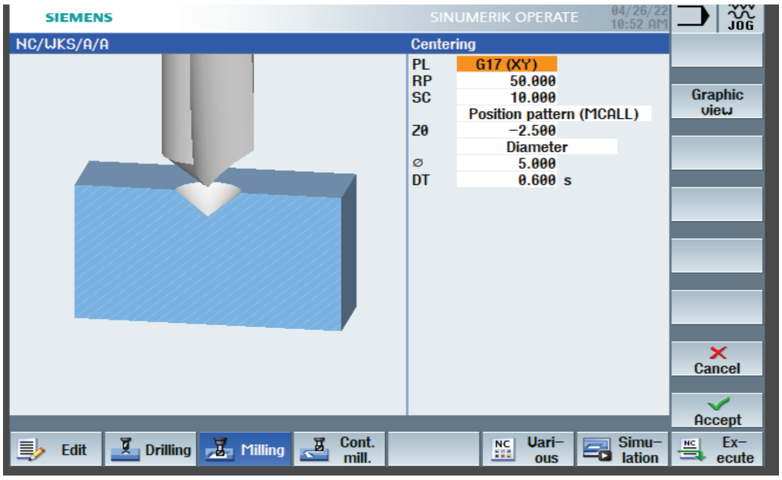


1	PRE MACHINED 150 x 50 x 25		ALUMINIUM			2.4.207
NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.
SCALE 1:1	**THREAD MILLING AND TAPPING**				DEVIATIONS	TIME
					CODE NO. MA20N24207E1	

Job Sequence

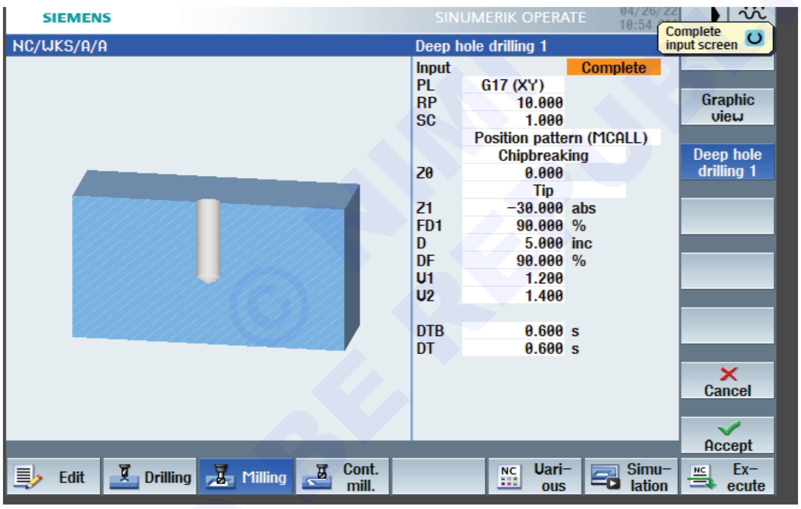
- Read the drawing
- List out the tools required to perform the practical
- Prepare the CNC program for
  - Center drilling use parameter as n Fig 1

Fig 1



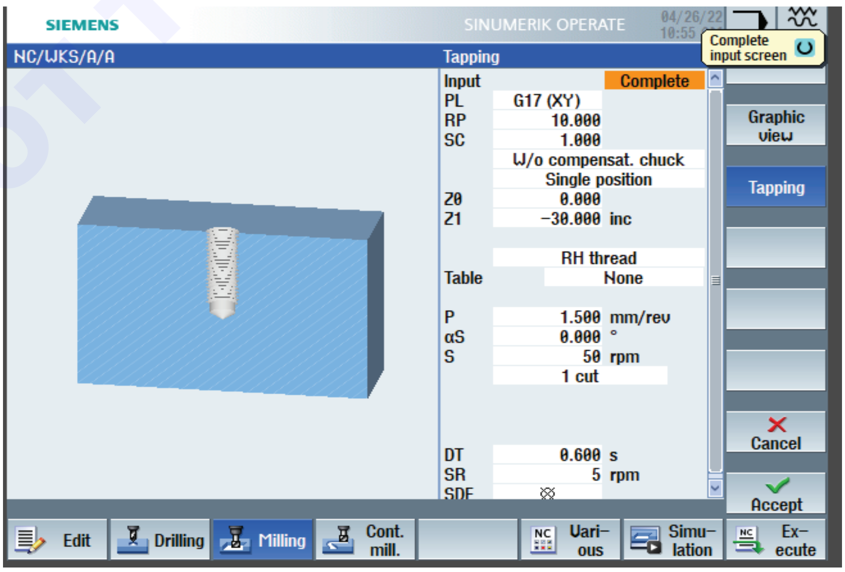
- Tap hole drilling use parameter as in Fig.2

Fig 2



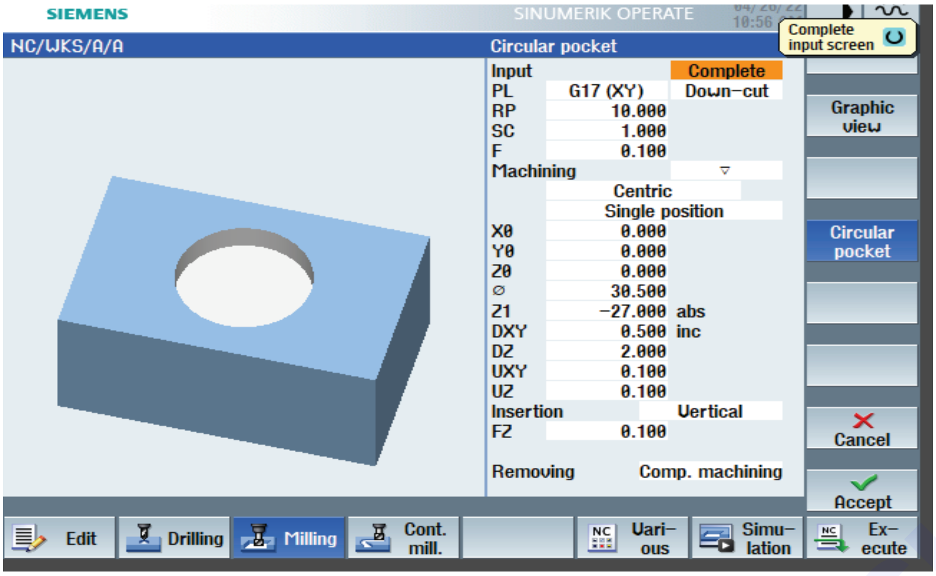
- Tapping M 10 x 1.5 use parameter as in Fig 3

Fig 3



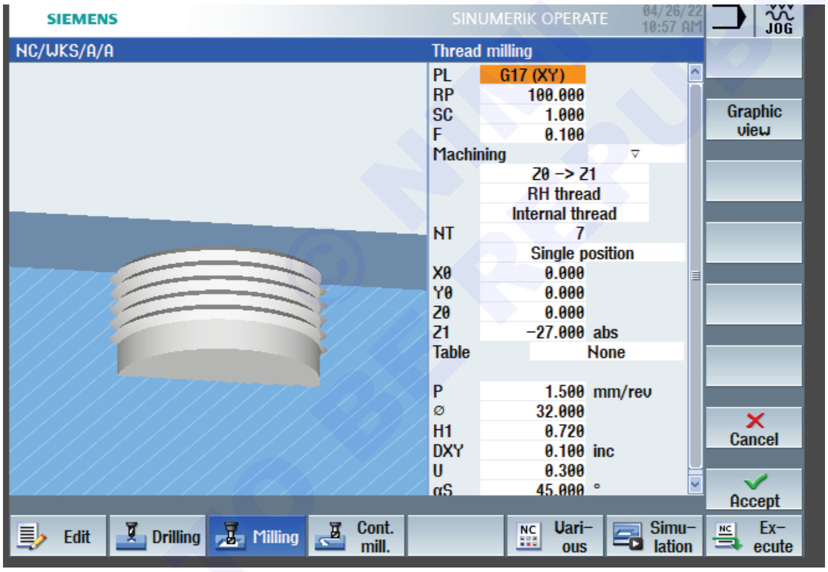
- Prepare the circular pocket program core diameter of thread M32 x 1.5 mm use the parameter as in Fig 4

Fig 4



- Prepare the RH internal thread milling program use the parameter as in Fig 5

Fig 5



- Verify the program
- Fix the offset sets and execute the program

Siemens Program

N10 T="CENTREDRILL 12"

N20 M06 M03 S700

N30 WORKPIECE(,"",,"RECTANG;E",0,0,-25,-80,150,50)

N40 G00 X50 Y50 Z100 F100

N50 MCALL CYCLE81 (50,-2.5,10,5,,0.6,10,1,11)

N60 X65 Y15

N70 X-65 Y15

N80 X-65 Y-15

N90 X65 Y-15

N100 MCALL

N110 G00 X100 Y100 Z100

N120 M05

N130 T="DRILL" (dia 8.5)

N140 M03 S500

N150 G00 X65 Y15 Z100 F100

N160 MCALL CYCLE83 (10,0,1,30,,,5,90,0.6,0.6,90,0,0,1.2,1.4,0.6,1.6,0,1,11211112)

N170 X65 Y15

N180 X-65 Y15

N190 X-65 Y-15

N200 X65 Y-15

N210 MCALL

N220 M05

N230 G00 X100 Y100 Z100

N240 T="TAP" (M10 tap)

N250 M6

N260 M03 S50

N270 G00 X65 Y15 Z100

N280 CYCLE (10,0,1,,-30,0.6,5,,1.5,0,50,5,0,1, 0,0,5,1,4,,,1001,1001001

N290 X-65 Y15

N300 X-65 Y-15

N310 X65 Y-15

N320 MCALL

N330 G00 X100 Y100 Z100

N340M05

N350 T="CUTTER 20" (end mill dia 20)

N360 M06

N370 M03 S400

N380 G00 X0Y0 Z10

N390 POCKET4 (10,0,1,-27,30.5,0,0,2,0. 1,0. 1,0.1,0.1,0,11,0.5,9,15,0,2,0,1,2,10100,111,100) (circular pocket)

N400 X0 Y0

N410 M05

N420 MCALL

N430 Z100

N440 T="THRREAD CUTTER"

N450 M06

N460 M03 S150

N470 G00 X0 Y0 Z100

N480 CYCLE70 (100,0,1,-27,32,0.72,0.3,1.5,7,0.1, 0.1,0,0,0,45,11,1,,,,1,0) (internal thread)

N490 MCALL

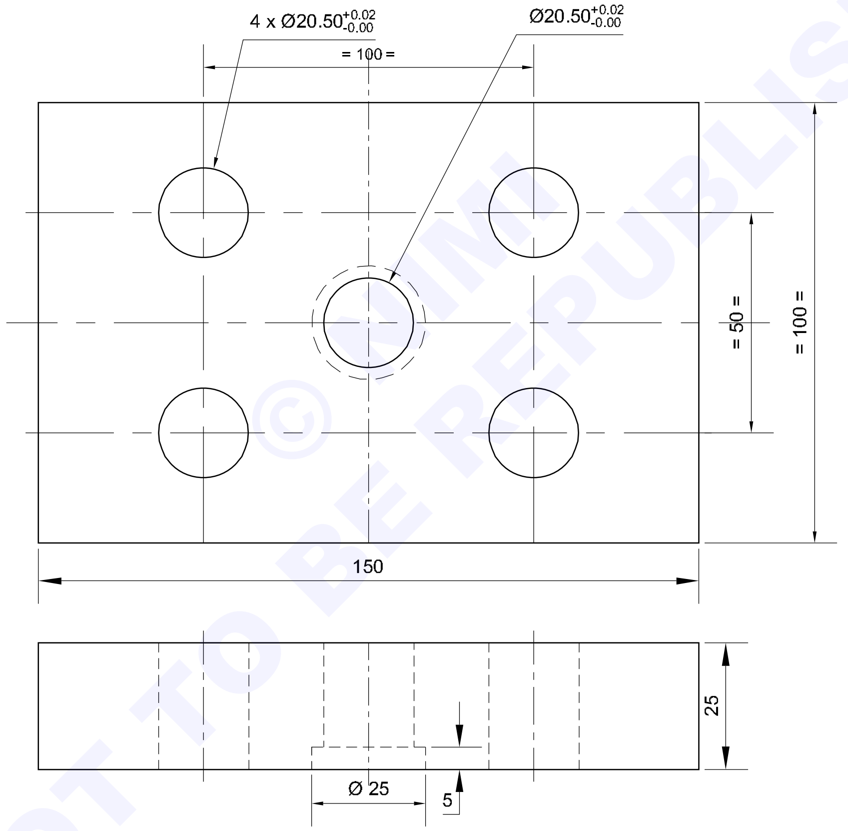
N500 M05

N510 G00 Z100

N520 M30

Carryout boring cycles G85-G89

Objectives : At the end of this exercise you shall be able to
- **drill hole using G83 cycle**
- **enlarge a drilled hole to the dimension using G85 cycle**
- **enlarge a drilled hole to the dimension using G86 cycle**
- **enlarge a drilled hole to the dimension using G87 cycle**
- **enlarge a drilled hole to the dimension using G88 cycle**
- **enlarge a drilled hole to the dimension using G89 cycle.**



1	PRE MACHINED 150 x 100 x 25		ALUMINIUM			2.4.208
NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.

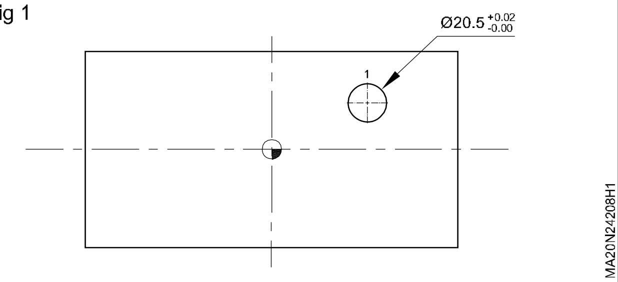
SCALE 1:1	**BORING CYCLES OPERATION G85 - G89**	DEVIATIONS	TIME
		CODE NO. MA20N24208E1	

Job Sequence

TASK 1: **(Fig 1)**

- Set the work piece on the CNC VMC
- Fix the work offset at the centre of the work piece

Fig 1



- Measure tool offset for 'U' drilling …20.00
- Set the boring head and the boring tool to the diameter of 20.5 $^{+0.02\ -0.00}$
- Measure the tool offset for the boring head and enter the tool offset in the tool offset page
- Write the program for 'U' drilling
- Write the program for boring using G85 cycle
- Verify the program and execute the program in automode with single block
- Measure the bore size, if required adjust the boring tool to the required dimension and re run the program
- Get it checked by the instructor

Example program

Fanuc G85 Boring Cycle format (Fig 2)

G85 X Y Z R F K

XY - Hole position

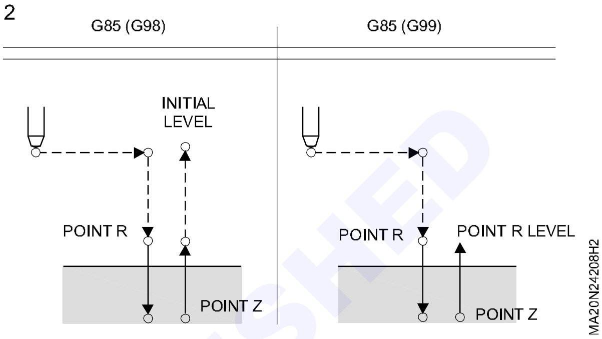
Z - Boring Depth (Absolute).

R - Tool starting position above the hole

F - Cutting feed rate

K - Number of repeats (if required)

Fig 2



Fanuc G85 Cycle example program

M 3 S100;

G00 90 G54 X50 Y40;

G43 H5Z100;

G99 G85 X50 Y50 Z-30-R5 F120;

G98 Z100;

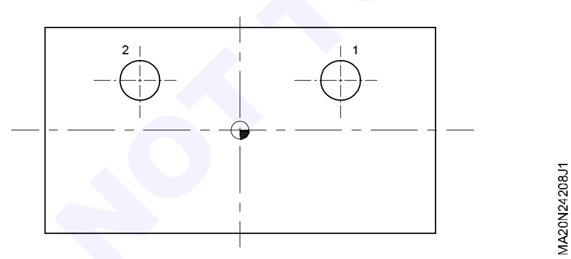
G80 G91 X0Y0Z0;

M5;

TASK 2: **(Fig 1)**

Follow the same steps except the boring cycle, that is used G86 for boring operation

Fig 1



G86 Boring Cycle

Fanuc G86 Boring cycle is used to bore the hole(s)

The tool travels to the bottom of the hole with feed and then retracts back out of the hole at rapid feed rate.

G86 Boring cycle format

G86 X Y Z R F K

XY - Hole position

Z - Boring Depth (Absolute).

R - Tool starting position above the hole

F - Cutting feed rate

K - Number of repeats (if required)

G86 Boring cycle operation

1. After positioning along the X and Y-axes, rapid traverse is performed to point R.
2. Drilling is performed from point R to Point Z.
3. When the spindle is stopped at the bottom of the hole, the tool is retracted in rapid traverse.

Toll Return Position

Return plane is dependant on G98, G99 G-Codes.

If G98 is specified with G86 boring cycle the tool return s to initial-level

If G99 is specified then tool will return to R level.

Fanuc G86 Cycle example program

M 3 S100;

G54 G90 X-50 Y50;

G43 H5 Z100;

G99 G99 G86 X-50 Y50 Z-30-R5 F120;

G99 Z100;

G80 G28 G91 X0Y0Z0;

M05;

— — — — — — — —

TASK 3:

Follow the same steps for drilling using G88 cycle for boring operation

G88 - Boring cycle

This one is more so practical for one-offs or really small production runs.

The reason for this is that the retract is manual. Here 's the order of events.

- Tool positions over hole
- Boring tool feeds down to programmed Z level
- Tool dwells for specified time
- Program stops
- Operator manually retracts the tool a safe location
- Spindle restarts and program continues.

G88 Cycle Format

G88 X- Y- Z- R- P- F- K-;

Parameters

X_ : Hole position data

Y_ : Hole position data

Z_ : The distance from point R to the bottom of the hole

R_ : The distance from the initial level to point R level

P_ : Dwell time at the bottom of the hole

F_ : Cutting feed rate

K_ : Number of repeats (if required)

G88 Cycle examples

G88 CNC Program example - 1

M3 S2000; (Cause the spindle to start rotating)

G90 G99 G88 X300. Y-250.Z-150.R-100.91000 F120.; (Position, drill hole 1 return to the point)

R then stop at the bottom of the hole for 1 second.

Y-550; (Position, drill hole 2, then return to Point R)

Y-750; (Position, drill hole 3, then return to Point R)

X1000; (Position, drill hole 4 then return to Point R)

y-550; (Position, drill hole 5, then return to Point R)

G98 Y-750; (Position, drill hole 6, then return to the initial level.)

G80 G28 G91 X0 Y0 Z0; (Return to the reference position)

M5; (Cause the spindle to stop rotating)

Note: Modify the program to perform Task 3

— — — — — — — —

TASK 4: **(Fig 1)**

Follow the same for drilling using G89 cycle for boring

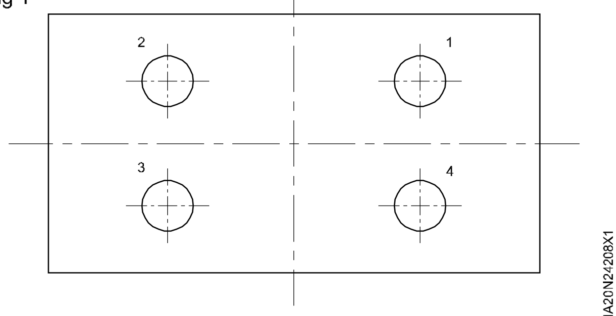
G89 Cycle introduction

G89 cycle is used to bore a hole. This cycle is almost the same as G 85. The difference is that this cycle performs a dwell at the bottom of the hole.

G89 Cycle format

G89 X_ Y_R_ P_ F_ K_;

Fig 1



Parameters

X_ : Hole position data

Y_ : Hole position data

Z_ : The distance from point R to the bottom of the hole

R_ : The distance from the initial level to point R level

P_ : Dwell time at the bottom of the hole

F_ : Cutting feed rate

K_ : Number of repeats (if required)

G89 CNC Program Example

M3 S1000; Cause the spindle to start rotating.

G90 G99 G88 X300. Y-250.Z-150.R-100.91000 F120.; Position, drill hole 1 return to the point R then stop at the bottom of the hole for 1 s

Y-550; Position, drill hole 2, then return to Point R

y-750; Position, drill hole 3, then return to Point R

X1000; Position, drill hole 4 then return to Point R

y-550; Position, drill hole 5, then return to Point R

G98 Y-750; Position, drill hole 6, then return to the initial level.

G80 G28 G91 X0 Y0 Z0; Return to the reference position

MS; Cause the spindle to stop rotating

Note: Modify the program to perform Task 4.

— — — — — — — —

TASK 5: **(Fig 1)**

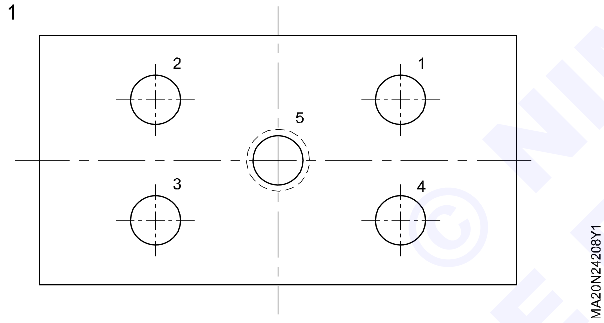
'U'Drill .. 20.00 mm at the centre position

Use G85 for boring operations

Set the tool for back boring in the diameter of 25.00

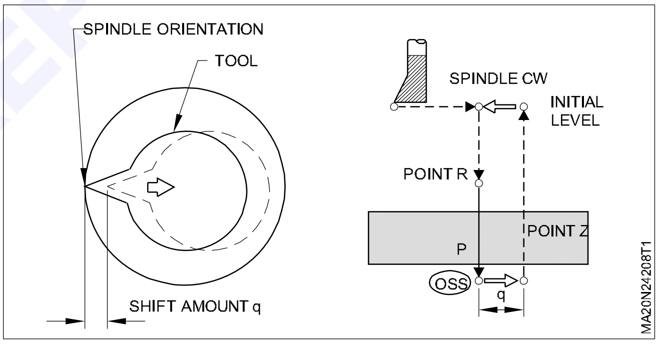
Perform the back boring using G87 cycles

Fig 1



G87- Back-Boring Cycle

- Boring tools are special tools that have a special tip and diameter to be machined is adjusted with the help of screw on the tool.
- This cycle basically works in the same way as the G86 boring process. The only difference between each other G86 from top to bottom, G87 works from bottom to top.
- With this cycle, boring tool goes to the coordinate specified in the program., the spindle stops in a fixed positon (usually in the orient position), the boring tool moves by the Q value in the opposite direction to the direction of the tool tip (nose), quickly approaches the depth Z axis indicated by R at the bottom of the hole, moves by the Q value in the tool tip (nose) direction, the spindle rotates in the specified direction (generally clockwise) up to the speed given, it moves to the height given by Z. with the feed rate specified by F and performs the hole expand and boring. Then, it goes up to the Z height (G98) while going back to the first hole center with rapid movement. If another coordinate is given after the cycle, it moves there and the cycle continues until G80 command is given.
- The cycle also works well for back-chamfering. Since this is a complicated cycle, it's a good idea to test it out a couple of times with a common Z workpiece shift above the part so you can make sure you got all the values right.
- Ultimately, though, this cycle can be really worth the effort. It can mean than you can possible finish a part in a single setup, which can make manufacturing considerably less expensive.



G87 Cycle format

G87 X.. Y.. Z.. Q.. P.. R.. K.. F..

G87: Boring cycle (From bottom to top)

: Hole position in X axis

Y: Hole position in Y axis

Z: Z axis end position = Z depth = Hole depth

Q: Tool sideway shift

P: Dwell time at the bottom of hole

R: Z axis start position = R level = Clearance

K: Number of cycle repetitions

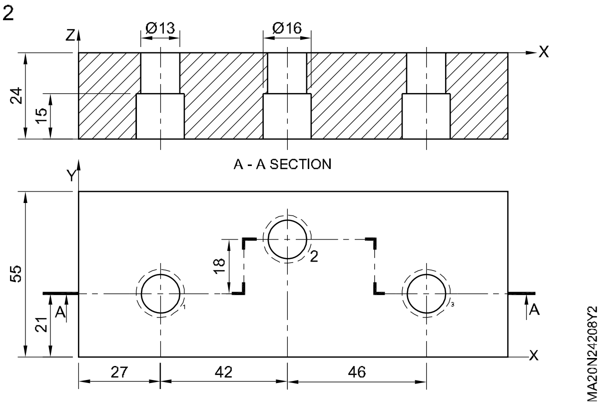
F: Feed rate

G87 Cycle Program example (Fig 2)

G87 Cycle program example

G28 G91 Z0;

Fig 2



T5 M5;
M03 S800;
G90 G54 G0 X0 Y0;
G43 H4 Z20 M08;
G90 X27 Y21;
G98 G87 Z-9 R-29 Q3 P1000 F120;
X69 Y39;
X115 Y21;
G80;
M30;

Note: Modify the program to suit the dimensions for Task 5.

Skill Sequence

Offset boring head

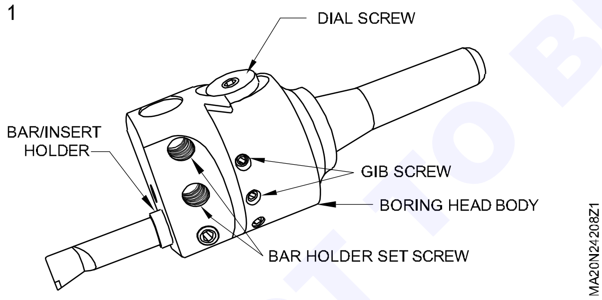
Objectives: This shall help you to
- **identify offset boring head**
- **Set the boring head to the required diameter.**

Offset Boring head

The offset boring is an attachment that fits the milling machine spindle and permit as most drilled holes to have a better finish and better diameter accuracy. Offset boring head are used to create large holes when tolerance do not allow for a drill bit or do not have a large enough drill or reamer. A offset boring head can be used to enlarge hole, or adjust hole centreline in certain instances.

Fig 1 shows an offset boring head. Note that the boring bar can be adjusted at a right angle axis. This feature makes it possible to position the boring cutter accurately to bore holes of varying diameters.

Fig 1



This adjustment is more convenient than adjusting the cutter in the boring bar holder or changing the boring bar. Another advantage of the offset boring head is the fact that graduated micrometer collar allows the tool to be moved accurately a specified amount usually in increments of (0.1/0.001) without the use of dial indicator or other measuring device.

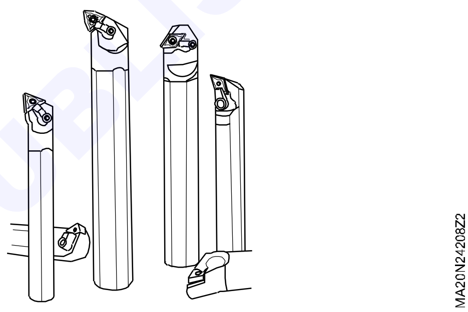
Figure 2 shows the single point boring tool and figure 3 shows the assemble of boring tool with boring head.

Setting of boring head

Select the appropriate boring head for boring operation.

Select the suitable boring tool depending upon the bore diameter and depth of the bore.

Fig 2



Insert the boring tool in boring head and tighten it.

Loosen the locking screw

Set the boring tool point for required diameter

(Turn the dial screw clockwise to increase the diameter and counter clockwise to decrease the diameter).

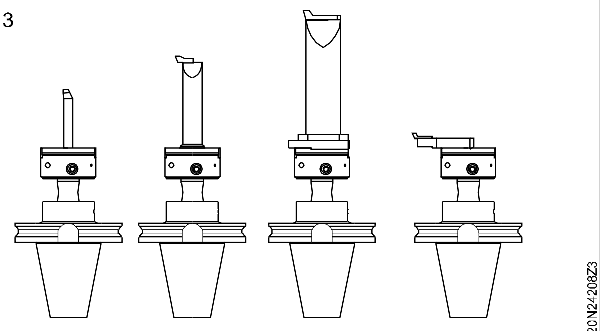
Tighten the locking screw.

Set the tool in tool magazine with corresponding tool number in the program.

Set the tool off set in x and z direction enter it in the offset page

Execute the boring operation and check the diameter if require re adjust the diameter of the bore.

Fig 3

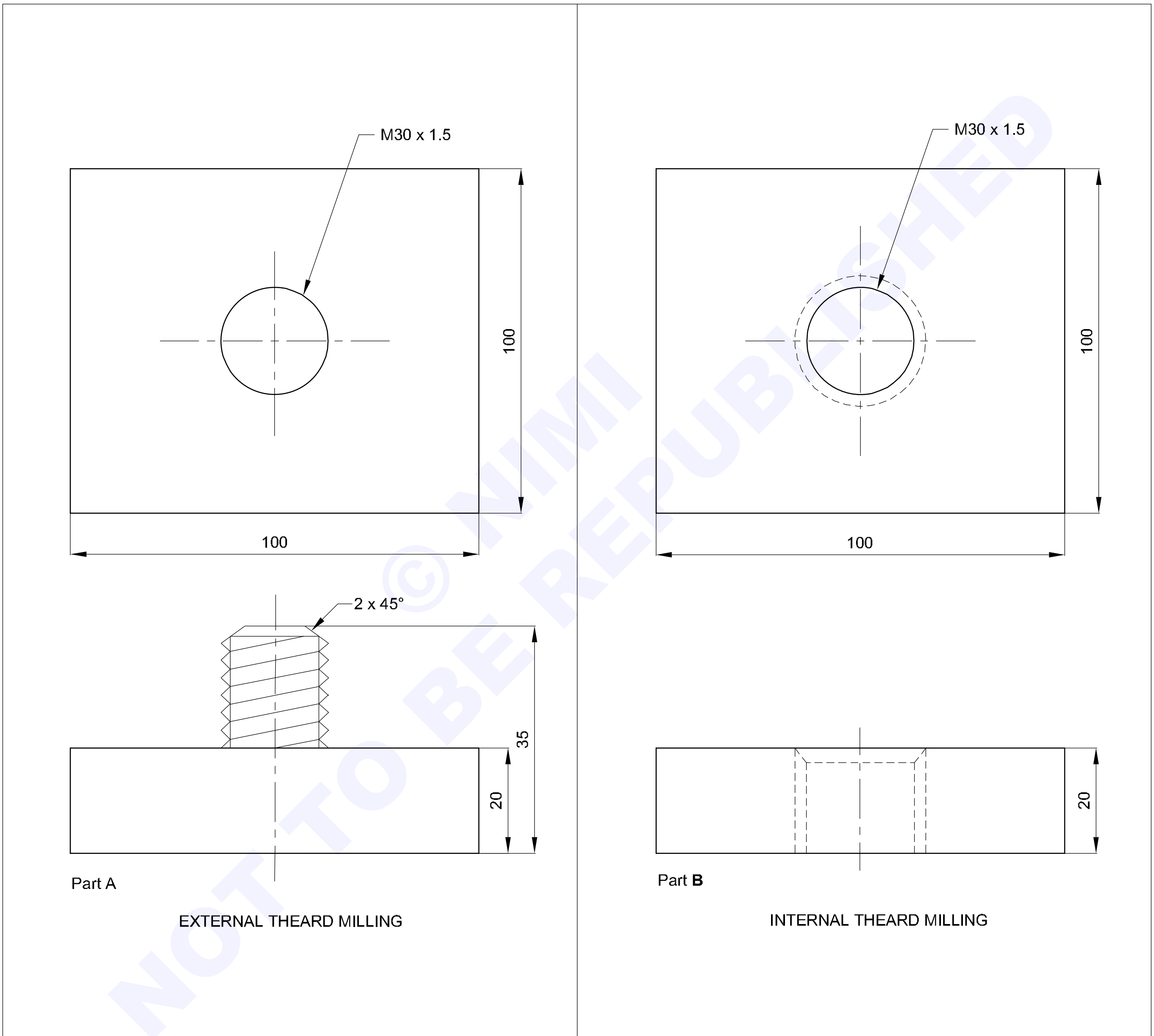


BORING TOOL

Part program for thread milling

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

- **mill round projection on the given work piece**
- **mill external thread using multi point threading tool**
- **bore to the required dimensions using solid drill**
- **mill internal thread using multi point threading tool.**



1	PRE MACHINED 100 X100X 20 mm	–	ALUMINIUM-LM25	PART-B	–	2.4.209
1	PRE MACHINED 100 X 100 X 35 mm	–	ALUMINIUM-LM25	PART-A	–	2.4.209
NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.

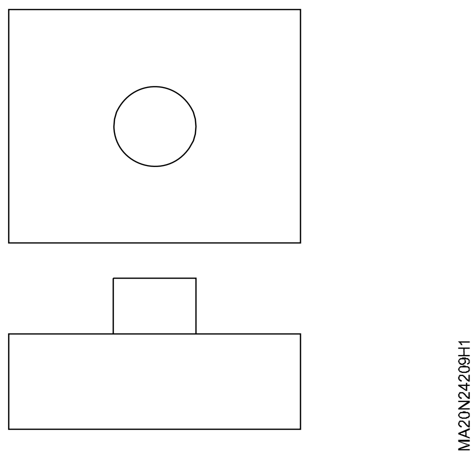
SCALE 1 : 1	**THREAD MILLING**	TOLERANCE	TIME
		CODE NO. MA20N24209E1	

Job Sequence

Part A

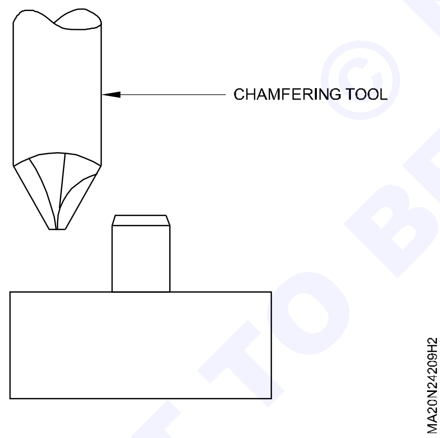
- Set the work piece on the machine
- Fix the work offset at the centre of the workpiece
- Measure tool offset, and enter in offset page
- Prepare a program to mill ∅ 30 mm x 15 mm deep as shown in Fig 1.

Fig 1



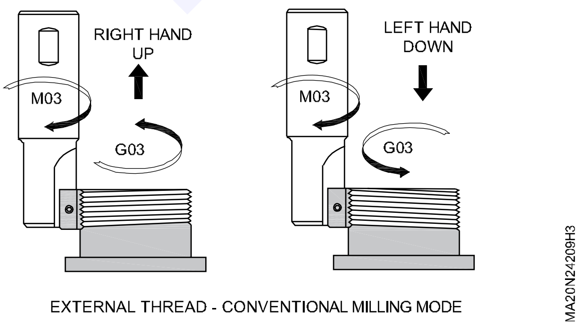
- Write the program to remove the material around the round projection
- Write the program to chafer 2 x 45° as shown in Fig 2.

Fig 2



- Write the program for external thread milling on M30 x 1.5 by helical interpolation method. (Fig 3)
- Verify the program and execute the program

Fig 3



Part A

O4001 (Program for round boss milling) (Fig 4)

N1 G80 G40 G49 G69;

N2 G90 G21 G94 G17;

N3T01 M06; (Solid end mill ∅25);

N4 S700 M03;

N5 G00 G55 G43 H01 X-65 Y0 Z50;

N6 G00 Z-5;

N6 G01 G41 D101 X-15 Y15 F100;

N7 G02 X-15 Y15 I15 J0;

N8 G01 Z-10;

N9 G02 X-15 Y15 J15 J0;

N10 G01 Z-15;

N11 G02 X-15 Y15 J15 J0;

N12 G01 X-35 Y0;

N13 G02 X-35 Y35 I35 J0;

N14 G01 x -55;

N15 G01 X-55 Y55 I55 J0;

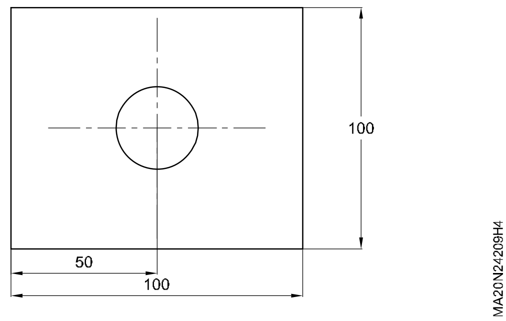
N16 G00 Z100;

N17 M05;

N18 G91 G28 X0 Y0 Z0;

M19 M30;

Fig 4



Program for Part A (External threading)

O4002 (program external thread milling)

N1 G80 G40 G49 69;

N2 G90 G21 G94 G17

N3 T02 M06; (multi point threading tool);

N4 S700 M03;

N5 G00 G55 G45 H02 X-65 Y0 S50; (Fig 5);

N6 G01 Z-14 F100;

N7 G01 G41 D102 X-29.02 Y0; (Fig 6);

Fig 5

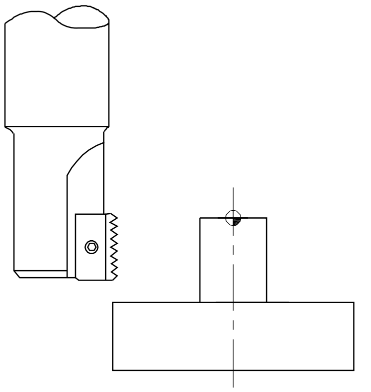
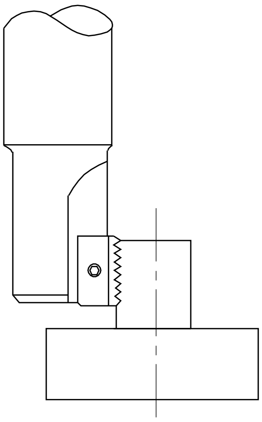


Fig 6



N8 G03 X-29.02 Y0 I29.02 J0 Z-12.5; (Fig 7)

N9 G01 G40 X-65 Y0;

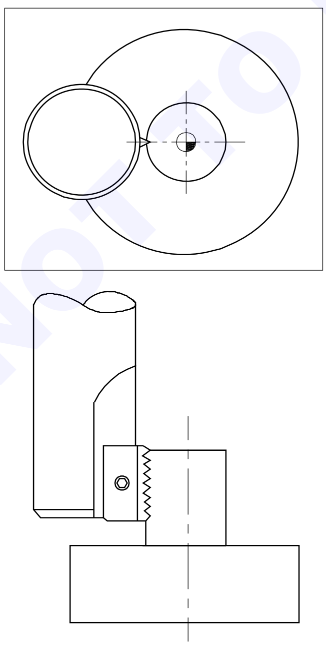
N10 G00 Z100;

N11 M05;

N12 G91 G28 X0 Y0 Z0;

N13 M30;

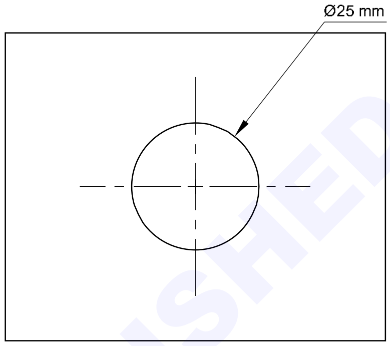
Fig 7



Part B

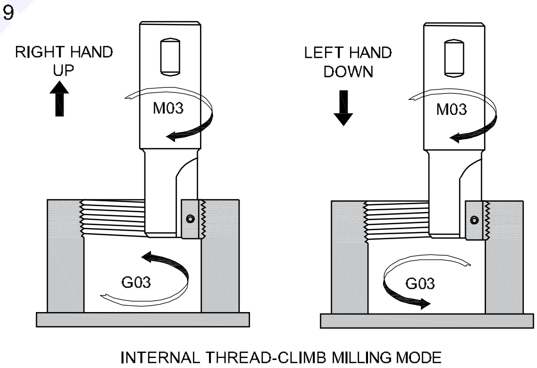
- Set the work piece on CNC machine table with suitable work holding device
- Fix the work offset at the center of the work piece
- Measure the tool offset (solid end mill/end cutting end mill) ∅25 mm
- Enter the offset at relevant offset pages.
- Prepare the program to drill hole at the centre of the work piece using drilling cycle (Fig 8)

Fig 8



- Enlarge the hole to minor diameter of thread (M30 x 1.5) by circular interpolation using the end mill
- Write the program for internal thread milling using multi point treading tool for M 30 x 1.5 shown in Fig 9
- Verify and execute the program

Fig 9



Program for circular pocket milling by circular interpolation

O4003

N1 G80 G40 G49 G69;

N2G90 G21 G90 G1;

N3T03 M06 (Solid carbide end cutting tool ∅25mm);

N4 S700 M03;

N5G00 G55 G43 H03 X0 Y0 Z5;

N6 G01 Z-30;

N7G01 G42 X28.5 Y0, (minor dia of the thread);

N8 G03 X 28.5 Y0 I-28.5 J0;

N9 G40 X0 Y0;

N10 G00 Z100;

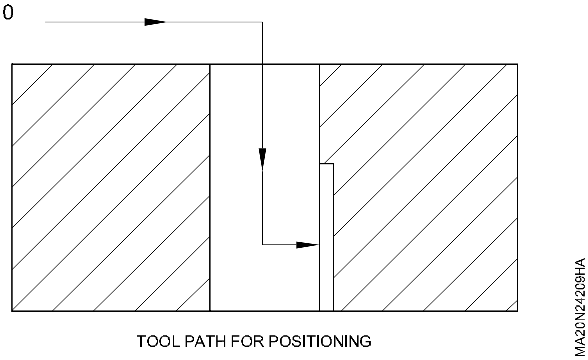
```
N11 M05;
N12 G28 G91 X0 Y0 Z0;
N13 M30;
```

Program for female thread

Part B

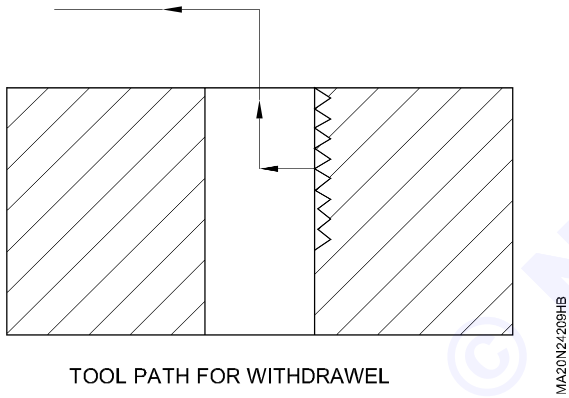
O4004 (Female thread milling (Fig 10&11)

Fig 10



TOOL PATH FOR POSITIONING

Fig 11



TOOL PATH FOR WITHDRAWEL

```
N1 G80 G40 G49 G69;
N2 G90 G21 G94 G17;
N3T02 M06 (Multi point threading tool);
N4S700 M02;
N5G00 G55 G43 H02 X-30 Y0 Z5;
N6 G01 G42 X0 Y0 F100;
N7 G01 Z-15;
N8 G01 X 28.5 Y0 (Fig 1);
N9 G03 X28.5 Y0 I-28.5 J0 Z-13.5;
N10 G01 X 0.0 Y0.0;
N11G01 Z10;
N12G01 G40 X-65 Y0 (Fig);
N13 M 05;
N14 G28 91 X0 Y0 Z0;
N15 M30;
```


Skill Sequence

Thread milling on VMC

Objectives: This shall help you to
- **perform external thread milling by multi point threading tool**
- **perform internal thread milling by multi point threading tool.**

With today's CNC machining centres, thread-milling has entered a whole new era. Indexable inserts now allow tough precision threads to be quickly and efficiently milled on almost any material, in holes from 9.5mm diameter upto virtually any size. What has given rise to this possibility? The development of Helical interpolation function of modern CNC machines is the reason.

What is helical interpolation?

The prerequisite for generating threads with a thread milling insert is the CNC function for guiding a tool movement along a helical path Fig 1 illustrates an orbital movement of the tool in the X-Y plane with a simultaneous linear movement along Z axis

Fig 1 Helical interpolation creates tool movement in a helical path from point A to point B on the surface of an imaginary cylinder. This helical path involves simultaneous movements in the X, Y and Z axes. In thread-milling, the circular movement in the X-Y plane creates the thread diameter while the simultaneous linear change in Z area creates the pitch of the thread.

Thread Milling Tool (Fig.2)

Thread milling cutters are available in at least two varieties some are made of solid carbide, some use carbide interchangeable inserts. In either design, the threading tool pitch must match the pitch of a thread required by the drawing. Tool has to be small enough to fit into the available internal space and large enough to guarantee suitable rigidity while cutting externally. For internal thread milling, cutters are available for thread milling in holes as small as 0.250 inch (6.35 mm)

Fig 1

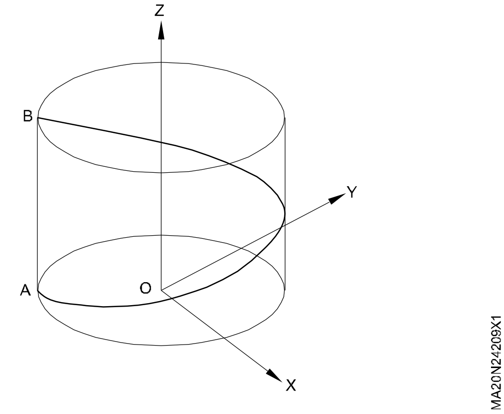
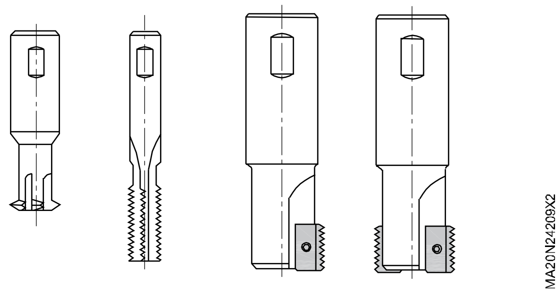


Fig 2



Premachining Requirements

A hole for a top cannot have the same diameter as the tap itself. It has to be smaller to accommodate the depth of the thread. The same rule applies to helical milling;

- If a thread is milled on the inside diameter of the part (internally), the premachined diameter must be smaller that the normal size.
- If a thread is milled on the outside diameter of the part (externally), the premachined diameter must be equal to the nominal size.

Either diameter (internal or external) may vary slightly larger or slightly smaller than the 'normal' size, but this deviation is decided by the required 'fit' of the thread.

Starting position

All required data have been collected and properly calculated, another step can be made, this time to calculate the thread start position.

This is easy for X and Y axes-the center of thread diameter is as good start as any -better, in fac. In this example and for simplicity, this XY position is also equivalent to the part origin X0Y0 position.

Start position of thread cutter measured along the Z-axis is much more important in helical milling than in any other type of milling. Z-axis start position must always be synchronized with the thread pitch, as the cutting will proceed in three axes simultaneously Z-axis zero (Z0) will be at the top of the part.

Z-Axis start position is determined by several factors- size of the thread mill (in this case a tool with an indexable insert), pitch of the thread, direction of Z-axis motion (Up or down) and the method of infeed along XY axes.

When a thread is cut using the helical interpolation feature, all the three axes used must be considered equally. Just like defining the approach are for circular interpolation, the approach are for a helical interpolation can be defined the same- the procedure is exactly the same

Motion Rotation and Direction Fig 3, 4 & 5)

In helical interpolation it is important to coordinate, to synchronize, the following three program items:

- Spindle rotation
- Circular cutting direction
- Z-axis motion direction

Why are these three items so important? Why do they have to be coordinated at all? Evaluate them one by one

Spindle Rotation

Spindle rotation can be either M03 (clock wise) or M04 (counter clock wise)

Circular cutting direction

Circular direction follows the rules of circular interpolation- G02 is clockwise, G03 is counter clockwise direction

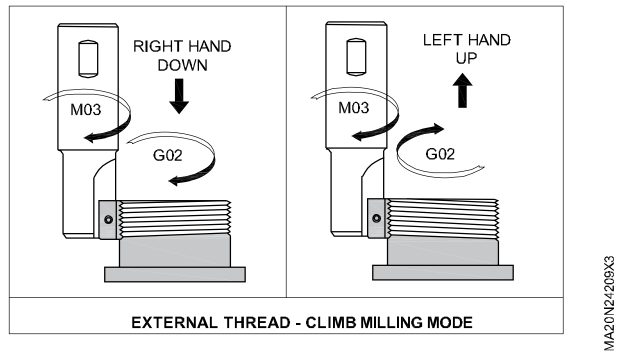
Z-Axis motion direction

For vertical machining, Z axis cutting direction may be along two directions/

- Up or positive
- Down or negative

EXTERNAL thread milling using the climb milling mode- right and left hand thread, spindle rotation and cutter motions as shown in FIg 3

Fig 3

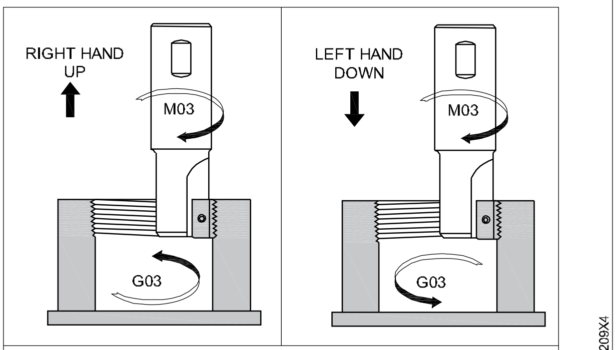


Helical milling

INTERNAL thread milling using the climb milling mode- right and left hand threads, spindle rotation and cutter motions as shown in Fig 4.

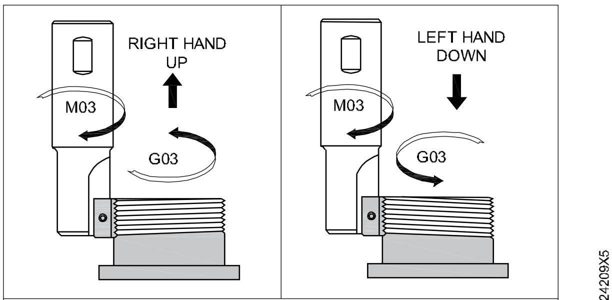
EXTERNAL thread milling using the conventional milling mode-right and left hand threads, spindle rotation and cutter rotation as shown in Fig 5.

Fig 4



INTERNAL THREAD-CLIMB MILLING MODE

Fig 5

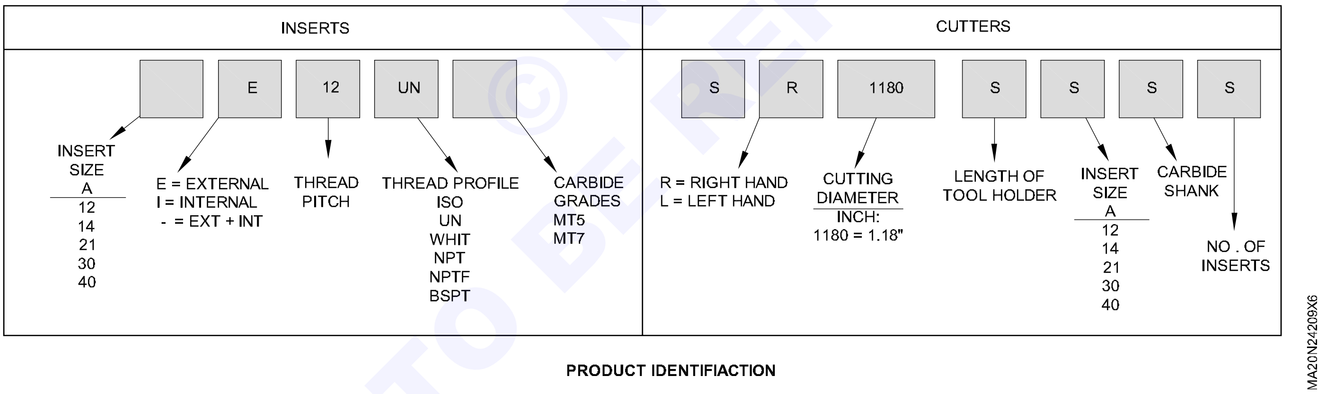


EXTERNAL THREAD - CONVENTIONAL MILLING MODE

Advantages

- Thread is produced in one tool pass
- Same tool holder and inset can produce both right hand and left hand threads
- A single insert and tool holder can produce a given thread on many diameters (External and internal)
- Prismatic shape ensures exact and reliable clamping in the tool holder
- Most inserts are double sided, having two cutting edges.
- Longer tool life due to a special multi-layer coating process
- Capable of producing tapered threads
- Improved productivity due to increased cutting speeds and multitooh type carbide inserts
- Threading to within one pitch of the bottom in a blind hole
- Considerably less expensive than using taps and dies, lowering tooling costs.
- Since lower machine power is required, a smaller machine can produce larger threads in a single operation with less idle time and tool changes.

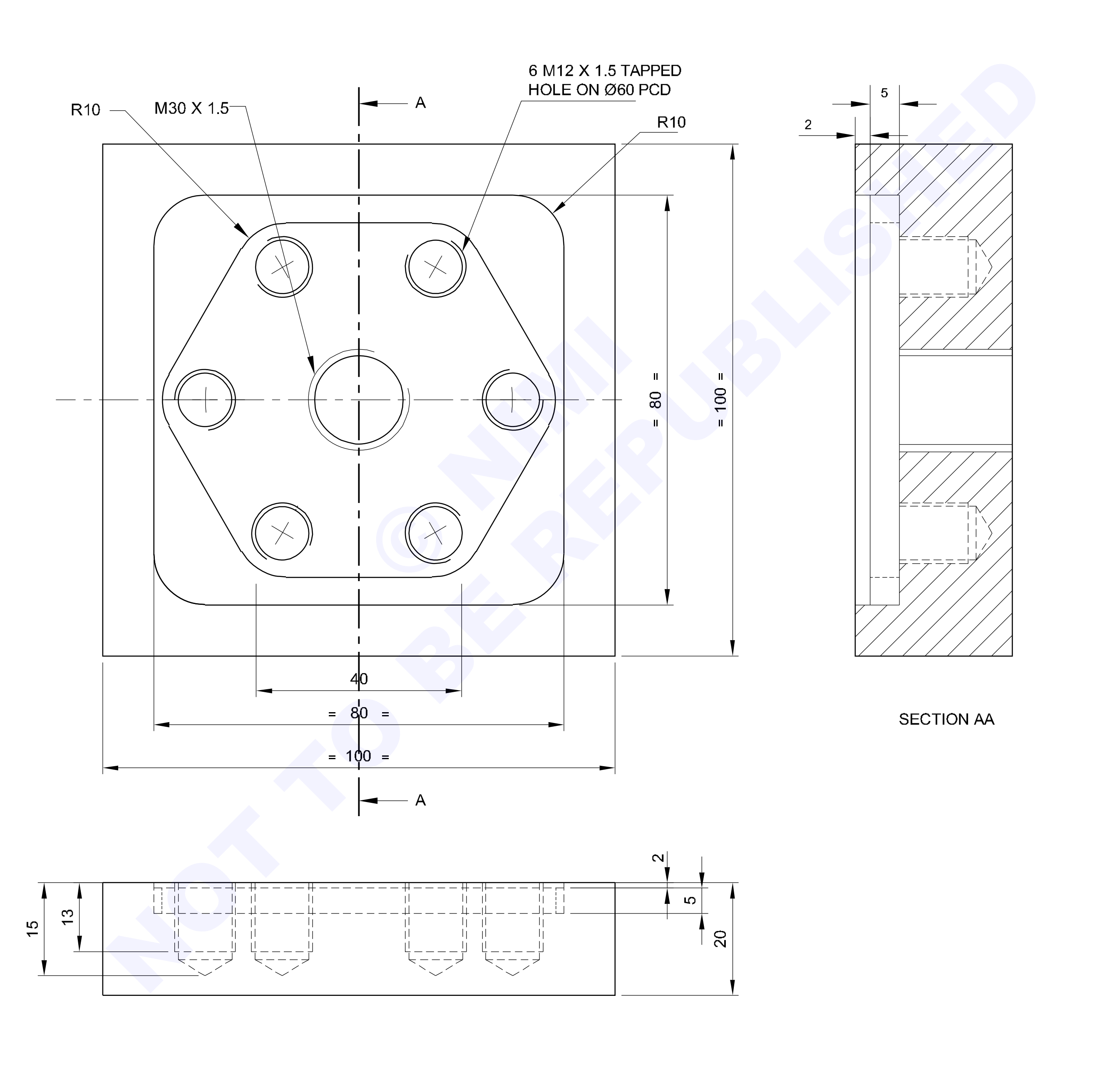
Fig 6



PRODUCT IDENTIFIACTION

Drilling, milling and threading patterns (Siemens Control)

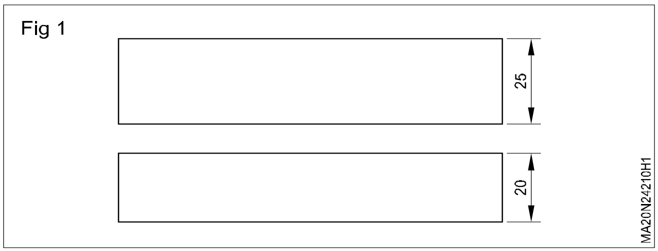
Objectives : At the end of this exercise you shall be able to
- **prepare the program for face milling and verify it.**
- **prepare the program for rectangular pocket milling and verify it.**
- **prepare the hexagon pocket milling using polar co-ordinates and verify it**
- **prepare the program for drilling and tapping using polar co-ordinates and verify it**
- **prepare the program for thread milling usng multi point-threading tool and verify it.**



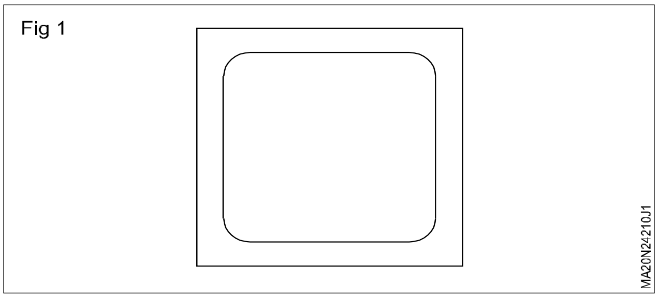
1	PRE MACHINED 150 x 100 x 25		ALUMINIUM			2.4.210
NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.
SCALE 1:1	DRILLING AND MILLING PATTERNS				DEVIATIONS	TIME
					CODE NO. MA20N24210E1	

Job Sequence

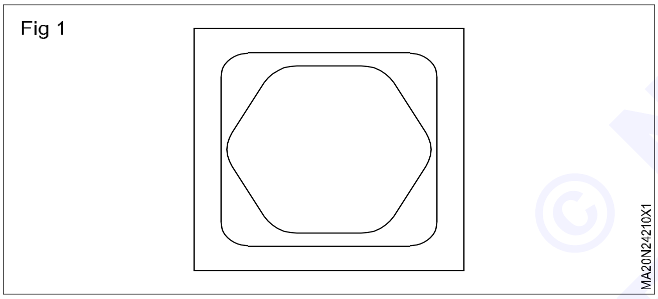
TASK 1: **Prepare face milling program to reduce the thickness if workpiece 25 mm to 20 mm (Fig 1)**



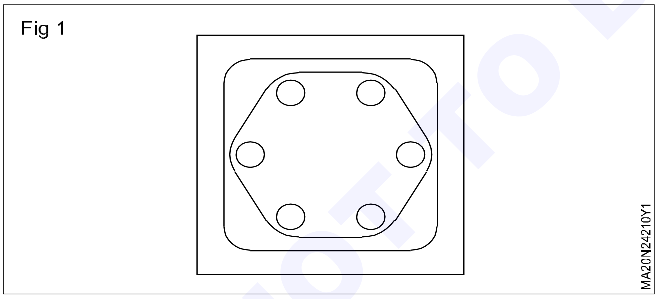
TASK 2: **Prepare rectangle pocket milling program to a depth of 2 mm (Fig 1)**



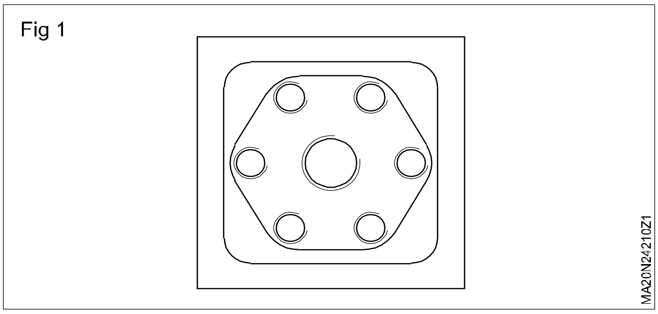
TASK 3: **Prepare program for hexagonal pocket milling using polar co-ordinates to a depth of 1 mm from top surface (Fig 1)**



TASK 4: **Prepare program for polar co-ordinate drilling/tapping as per drawing (Fig 1)**

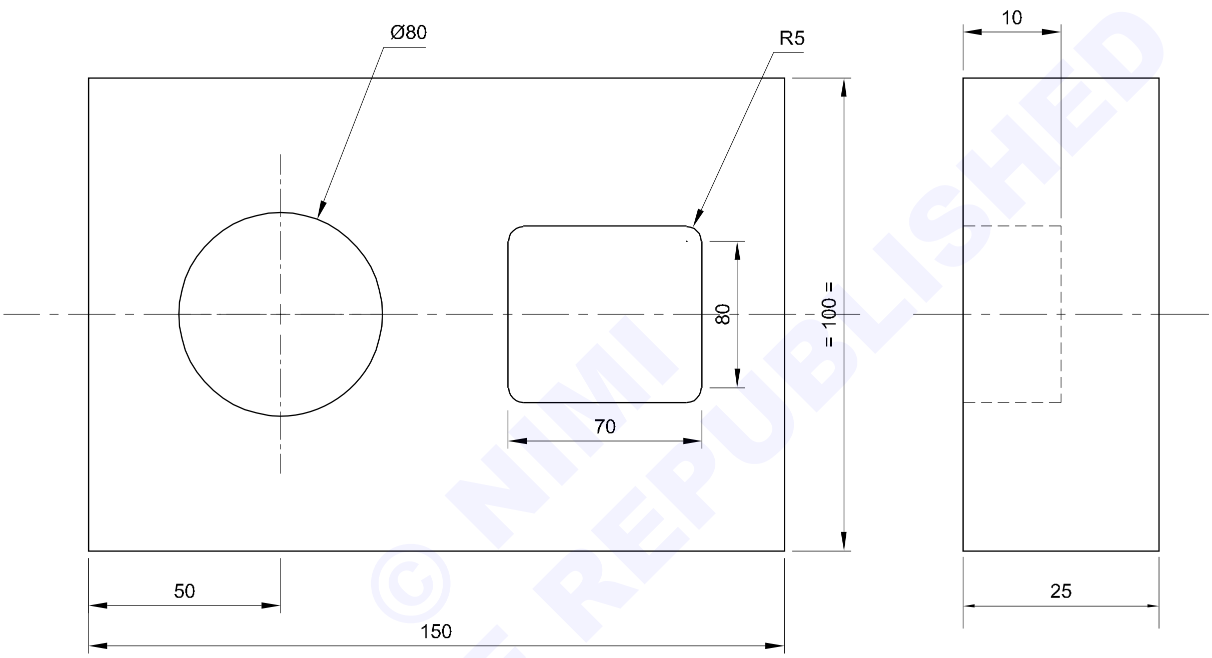


TASK 5: **Prepare thread milling program by mill thread of M 30 x 1.5 mm using multi point threading tool (Fig 5)**



Circular and rectangular pocket machining

Objectives : At the end of this exercise you shall be able to
- **prepare the part program for circular pocket milling in sinumeric control using cycle**
- **prepare the part program for rectangular pocket milling in sinumeric control using cycles.**



Job Sequence

- Execute the program in automode and maintain the dimensions.
- Trainee shall prepare part program and verify the program
- Get it checked by the instructor
- Execute the program in automode.
- Verify the dimensions, if needed correct the wear offset.

1	PRE MACHINED 150 x 100 x 25		ALUMINIUM			2.4.211
NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.
SCALE 1:1	CIRCULAR AND RECTANGULAR POCKET MILLING				DEVIATIONS : ±0.02mm	TIME
					CODE NO. MA20N24210E1	

Calculation of machine hour rate for CNC machines

Objective: At the end of this exercise you shall be able to
• **calculate the machine hour rate for any machine.**

Job Sequence

Machine hour rate obtained by dividing the total running expenses of a machine during a particular period by the number of hours the machine is estimated to work during that period.

Machine Hours Rate

- The machine hour rate is similar to the labour hour rate method and is used where the work is performed primarily on machines.

Formula:

The formula used in computing the rate is:

Factory overhead/Machine hours

If factory overhead is Rs.3,00,000 and total machine hours 1,500, the machine hour rates is Rs.200 per machine hour (Rs.3,00,000/ 1500 hours)

Problem 1:

In a machine shop, the machine hour rate is worked out at the beginning of a year on the basis of 13 week period which is equal to three calendar months.

The Following estimates for operating a machine are relevant:

Total working hours available per week	48 hours
Maintenance time included in the above	2 hours
Setting up time included in the above	2 hours
Cost details:	
Operators wages (per month)	Rs.650
Supervisory Salary (per month)	
(Common supervisor for 3 machines)	Rs. 1,800
W.D.V of machine (Depreciation at 10% plus 2% on average for extra shift allowance)	Rs.1,80,000
Repairs and maintenance (per annum)	Rs.16,000
Consumable stores (per annum)	Rs.30,000
Rent, rates and taxes (for the quarter apportioned)	Rs.5,000

Power consume is @15 units per hour "@40 paise per unit. Power is required for productive hours only, setting-up time is part of productive time but no power is required for setting up jobs.

The operator and supervisor are permanent. Repairs and maintenance and consumables are variable.

You are required to:

a Work out the machine hour rate.

b Work out the rate for quoting to the outside party for utilising the ideal capacity in the machine shop assuming a profit of 20% above the variable cost.

Solution:

a Computation of Machine Hour Rate

Standing charges per quater

Rent, Rates and Taxes	Rs. 5,000.00
Supervision	Rs. 5,400.00
Operator's wages	Rs. 1,950.00
Total standing charges	Rs.12,350.00

Effective hours (46 x 13) = 598

Fixed cost per hour (12,350/598 = Rs.20.65

Variable costs per hour

Power: $\frac{15 \times 44 \times 0.40}{46}$	= 5.74
Repairs and maintenance: 4,000/598	= 6.69
Consumable stores 7,500/598	= 12.54
Depreciation:	= 10.3
	Rs.35.00

Machine hour rate = Standard charges + Variable charge

= 20.65 + 35.00

= Rs. 55.65

Problem 2

An engineering company engaged in the manufacturing of various heavy engineering products, has installed one Numerical Control Horizontal Borer for specified manufacturing operations.

Calculate the machine hour rate on the basis of the following particulars:

i F.O.B cost of the machine Rs.24 lakhs

ii Custom duty, insurance, freight etc. Rs.11 lakhs iii Installation expenses RS.3 lakh

iv Cost of tools adequate for 2 years only Rs.4 lakhs

v Cost of the machine room Rs.3 lakhs

vi Cost of air conditioning for machine room Rs.2 lakhs

vii Rate of interest on term loan to finance the above capital expenditure @12% per annum

viii Salaries, etc for operators and supervisory staff of RS.2 lakhs per year

ix Cost of electricity Rs.11 per hour

x Consumption of Stores RS.5000 per month

xi Other expenses Rs.5 lakh per annum

xii Assume rate of depreciation as 10% per annum on fixed assets

xiii Total working hours in the machine room is 200 hours in a month

xiv Loading and unloading time is 10% of machine time

xv You can make suitable assumptions if necessary, for the purpose of your computation.

Solution:

Computation of Machine Hour rate

		Per hour
Machine Expenses:		
Cost of Machine	24.00.000	
Customs duty, insurance, freight etc.	11,00,000	
Installation expenses	3,00,000	
Cost of Machine room	3,00,000	
Cost of air conditioning of the machine room	2,00,000	
Total Capital Cost	43,00,000	
i Depreciation @ 10% pa.	4,30,000	Rs.199.07
ii Cost of electricity		11.00
iii Cost of tools	2,00,000	92.13
Standing charges:		
i Salaries	2,00,000	
ii Interest @12% p.a on total capital Investment including cost of tools, i.e on RS.47,00,000	5,64,000	
Consumption of stores	60,000	
Other expenses	5,00,000	
Total standing charges	13,24,000	
Cost per hour of standing charges		
(13,24,000/2,60 hrs.)		613.00
Machine hour rate		Rs.915.20

Notes

1 Total working hours per annum are 200 x 12=2,400

2 Loading and unloading time is 10% of machine time.

3 Effective working hours are 2,160 hours (2,400-10% of 2,400).

4 It is assumed that there is no consumption of electricity during loading and unloading time

5 Interest has been treated as an element of cost. It could be ignored also.

Exercises

Calculate machine hour rate for machine

Consumable stores	600
Repairs	800
Lighting and Heating	360
Rent	1260
Insurane of Building	4800
Insurance of machines	800
Depreciation of machines	700
Room service	60
General Charges	90

Working hours 1000 Hrs. Broke value 12,000.

(**Hint:** Insurance and depreciation of machines should be apportioned on the basis book values of machines and all other expenses on the basis of floor area covered.

Estimation of cycle time for milling - operation

Objectives : At the end of this exercise you shall be able to
- **calculate the machining time for face milling**
- **calculate the machining time for side milling**
- **calculate the time for drilling**
- **calculate the time for tapping.**

Job Sequence

TASK 1: **Estimation of cycle time for face milling operation**

Step

1 Determine the RPM (Spindle speed)

2 Determine the feed rate

3 Determine the approach distance

4 Calculate the machining time

Formula to calculate RPM

Rotational Speed (RPM's)

$$N = \frac{V \times 1000}{\pi D}$$

N= Rotational speed (RPM's)

V= Cutting Speed (M/minul)

D= Cutter diameter in millimetre

Feed rate: f_r (Dist/min)

$fr = N\, n_t\, f$

f_r = Feed rate (Dist/Min)

N = Rotational speed

n_t = Number of Teeth on the cutter

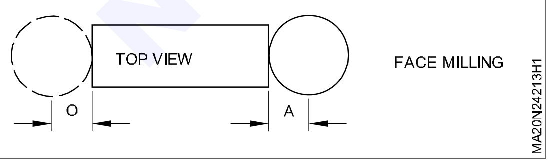
F = feed (mm/tooth)

Formula to determine Approach Distances

A= Approach Distance

O= Cutter Run out (Face milling)

Fig 1



Approach Distance: Face Milling:

$$A = O = \frac{D}{2}$$

A= Approach Distance

O=Cutter Run out distance

D= Cutter Diameter

Formula for Machining Time: Face Milling

$$T_m = \frac{L + A + O}{f_r}$$

T_m=Machining Time (Min)

L = Length of cut

A = Approach distance

O= Cutter Run out Distance

F_r= Feed rate (Dist/min)

Face Milling example (Fig 2)

Data D = 100; n_t = 6;

f = 0.04 m/tooth

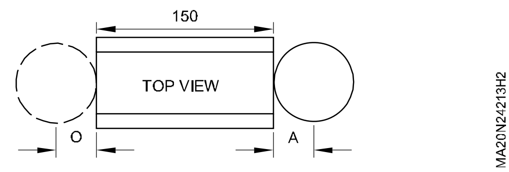
D = 100 mm

n_t = 6

f = 0.04 mm/tooth

V = 300 meter/minute

Fig 2



Approach & over travel distance

$$A = O = \frac{D}{2}$$

$$\frac{100}{2} = 50\text{mm}$$

Face milling example

Spindle rotation

$$N = \frac{V}{\pi D}$$

$$N = \frac{300 \times 1000}{3.4 \times 100}$$

N = 955.41

Feed rate

$f_r = N\, n_t\, f$

F_r = 955.41 x 6 x 0.04

= 229.3mm/minute

Face miiling example

Machining time

$$T_m = \frac{L + A + O}{f_r}$$

$$T_m = \frac{150 + 50 + 50}{229.3}$$

= 1.09 minute

— — — — — — — —

TASK 2: **Estimation of cycle time for side milling**

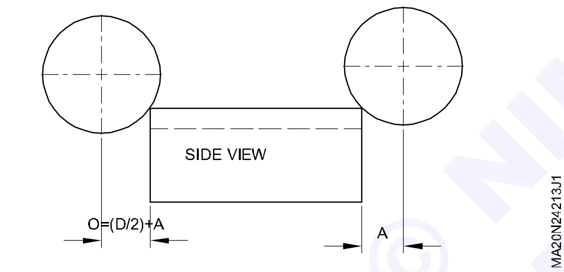
Formula to calculate

RPM (N)

Feed rate formula is same as face milling

Example (Fig 1)

Fig 1



$$\frac{100}{2} = 50mm$$

D = 100 min

d = 6 min

L = 100 mm

f = 0.012 mm/teeth

V = 300m/minute

n_t = 20 teeth.

Side milling example

Approach distance

$$A = \sqrt{d(D - d)}$$

A = Approach length

D = Diameter of cutter

d = Depth of cut

O = Cutter run out

$$= \sqrt{6(100 - 6)}$$

= 9.69 mm

Spindle rotation

$$N = \frac{V \times 1000}{\pi \times D}$$

$$= \frac{300 \times 1000}{3.14 \times 100}$$

= 955.14

Feed rate

$f_r = N\, n_t\, f$

= 955.4 x 20 x 0.012

= 229.2 mm/min

Machining time

$$T_m = \frac{L \times A}{f_r}$$

$$= \frac{100 + 9.69 + 50}{229.2}$$

$$= \frac{159.69}{229.2}$$

= 0.69 minute

— — — — — — — —

TASK 3: **Estimation of cycle time for drilling**

The drilling through hole (Fig 1)

The machining time, T_c is estimated from,

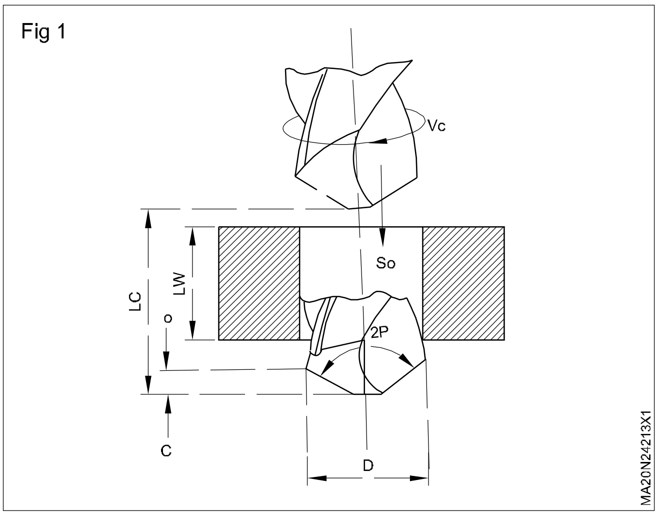


Fig 1

$$T_c = \frac{L_c}{NS_o} \qquad (4.9.8)$$

Where, $L_C' = L_h + A + O + C$

A, O = approach and over run

and $C = \frac{D}{2}\cot P$

D = Diameter of the hole i.e., drill

P = Half of the drill point angle.

Speed, N and Feed S_o are selected in the same was as it is done in case of turning

Therefore, the drilling time can be determined from,

$$T_c = \frac{\pi D(L_h + A + O + C)}{1000 V_C S_o} \qquad (4.9.9)$$

In the same the Tc is determined or estimated in boring also. Only the portion 'C' is not included.

For blind hole, only over run, 'O' is excluded.

Example

For D = 25 mm, p = 60°. V_c = 44 m/mm

L = 60 mm. S_o = 0.25 mm/rev

A=O=2 mm

Tc = π x 25 {60+2+2+(25/2)cot60°} / (1000 x44x0.25)

0.5 min

— — — — — — — —

TASK 4: **Estimation of cycle time for tapping**

RPM = (SFM X 3.82) divided by D

D = Diameter of Tap in inches

SFM = (3.14 x D x RPM) divided by 12

Metric Taps

MM/Min = RPM x Metric Pitch

Distance = Rate x Time

$$\text{Time} = \frac{\text{Distance}}{\text{Rate}}$$

Example : The machine feed rate (Vf) of a single start M12 x 1.75 thread ran at 186 revolutions per minute (RPM) is calculated as follows Vf= 186 RPM x 1.75 mm Thread pitch = 325.5 mm per minute feed rate millimetres per minute (mmpm) 325.5/1,000 = 0325 feed rate in Meters Per Minute (MPM)

Between 150 amd 250 rpm

For best results, the speed of the spindle should be between 150 and 250 rpm.

Note: To get the total cycle time, the idle return time and work-handiling time should be added to the time for actual cutting. In estimating the time to return the table to the starting position. A rapid traverse time of 2500 mm per minute may be used. Work-handling time varies with each job and must be determined by method- analysis studies.

— — — — — — — —

Documentation - 1

Objectives : At the end of this exercise you shall be able to
- **prepare and fill batch processing record format**
- **prepare and fill up bill of materials (BOM)**
- **prepare and fill up production cycle time format**
- **prepare and up daily production report format**
- **prepare and fill up manufacturing stage inspection report format.**

- **Trainer may arrange for a industrial visit near by your institute, collect inputs and fill up format as required.**
- **Trainees will be guided by the concerned trainer.**
- **Collect necessary information, forms and instruct the trainees to reproduce the forms and guide them to fill it up.**

Job Sequence

- Study the different types of documentation provided (format).
- Visit to industry and collect the input/ information from industry and fill it up of all the format.
- Prepare the required format with the knowledge gained during the industrial visit.
- Record relevant information in the format.
- Get it checked by your trainer.

BATCH PROCESSING RECORD - FORMAT - 1

Batch Processing Record

Description of job		Batch no. :
Part no. :		Batch quantity :
Name of part :		Batch record no. :
		Purchase order no. :
Description of process :		
Manufacturing Organisation :		
Period of manufacture (Year - Qtr):	Start date of manufacture:	End date of manufacture:

Number of pages according to batch:	Inserted pages:	Manufacturing facilities:
Total number of pages		
1. Operator / Technician	Date	Name and signature
2. Production in-charge:	Date	Name and signature
3. Section manager	Date	Name and signature
4. Plant in-charge:	Date	Name and signature
5. Production in-charge:	Date	Name and signature
Remarks (if any)		

BILLS OF MATERIALS (BOM) FORMAT - FORMAT - 2

BOM Level	Part Number	Part Name	Code	Quantity	Unit	Date of Procurement	Designators	Remarks

Date

Place

Incharge

PRODUCTION CYCLE TIME - FORMAT-3

Organisation Name: Department / Section:	Process:					Line Incharge:				Date/Time:	
Operator:										Machine Cycle Time	Notes
Operations Sequence	Observed Times								Lowest Repeatable		

DAILY PRODUCTION REPORT - FORMAT-4

	Daily Production Report							**Organisation Name:**				
Date:	**Department:** **Section:**											
	Process - I		Process-II		Process-III		Process-IV		Quality Control		Packing	
	Planned	Completed	Planned	Completed	Planned	Completed	Planned	Completed	Planned	Completed	Planned	Completed
Job Order No. Quantity Material & Size												
Job Order No. Quantity Material & Size												
Job Order No. Quantity Material & Size												
Job Order No. Quantity Material & Size												
Job Order No. Quantity Material & Size												

Signature of section Incharge

MANUFACTURING STAGE INSPECTION REPORT - FORMAT-5

Organisation Name:

Department / Section:

Status: From Date ..../...../...... To Date ..../...../......

Date	Product ID/ Code	Customer	P.O. No. & Date	Job Order No.	J.O. Date	Process	Qty	Accepted	Rejected	Inspection Record No. & Date	Inspection/ Test conducted by

Documentation-2

Objectives : At the end of this exercise you shall be able to
- **prepare and fill job card format**
- **prepare and fill activity log format**
- **prepare and fill batch production record**
- **prepare and fill estimation sheet**
- **prepare and fill maintenance log format**
- **prepare and fill the history sheet of machinery and equipment format**
- **prepare and fill maintenance record.**

- **Trainer may arrange for a industrial visit near by your institute, collect inputs and fill up format as required.**
- **Trainees will be guided by the concerned trainer.**
- **Collect necessary information forms and instruct the trainees to reproduce the forms and guide them to fill it up.**

Job Sequence

- Study the different types of documentation provided (format).
- Visit to industry and collect the input/ information from industry and fill it up of all the format.
- Prepare the required format with the knowledge gained during the industrial visit.
- Record relevant information in the format.
- Get it checked by your trainer.

JOB CARD - FORMAT-1

Job Card	Doc No.	
	Rev No.	
	Date	
Order Starting Date		
Customer		
Work Order No.		

Details

S.No.	Date	Production Line Description	Time (Minutes)			Location Time	Remarks
			Start Time	End Time	Total Time		

WORK ACTIVITY LOG - FORMAT-2

Organisation Name:

Department:

Section:

Employee Name:

Supervisor Name:

Date:

Start / Stop	Operations performed	Equipment / Machinery/ Instruments used	Remarks
8.00 to 9.00 a.m.			
9.00 to 10.00 a.m.			
10.00 to 11.00 a.m.			
11.00 to 12.00 noon			
12.00 to 1.00 p.m.			
1.00 to 2.00 p.m.			
2.00 to 3.00 p.m.			
3.00 to 4.00 p.m.			
4.00 to 5.00 p.m.			
5.00 to 6.00 p.m.			

BATCH PRODUCTION RECORD - FORMAT-3

<table>
<tr><td colspan="4">Batch Production Record in accordance with batch processing record

Manufacturing Organisation Name: ________________________

Description of job: ________________________

Name of part: ____________________________

Batch No.: _____________________________

The following deviations have appeared (continued)</td></tr>
<tr><th>No. process step</th><th>Name of processing step</th><th>Documented page no.</th><th>Short description of deviation</th></tr>
<tr><td>1</td><td>Raw material preparation:
Operation 1: ______________
Operation 2: ______________
Operation 3: ______________</td><td></td><td>1. ______________
2. ______________
3. ______________
4. ______________</td></tr>
<tr><td>2</td><td>Sizing of material:
Operation 1: ______________
Operation 2: ______________</td><td></td><td>1. ______________
2. ______________
3. ______________</td></tr>
</table>

ESTIMATION SHEET - FORMAT-4

<table>
<tr><td colspan="3">Part Name: ____________________

Assembly: __________________

Assembly No.: _______________</td><td colspan="2">Part No.: ______________

Material: _______________

Stock size: ______________</td><td>Part Drawing</td></tr>
<tr><th>Operation No.</th><th>Operation description</th><th>Machine</th><th>Estimated time</th><th>Rate / piece per hr.</th><th>Tools</th></tr>
<tr><td></td><td></td><td></td><td></td><td></td><td></td></tr>
<tr><td></td><td></td><td></td><td></td><td></td><td></td></tr>
<tr><td></td><td></td><td></td><td></td><td></td><td></td></tr>
<tr><td></td><td></td><td></td><td></td><td></td><td></td></tr>
<tr><td></td><td></td><td></td><td></td><td></td><td></td></tr>
<tr><td></td><td></td><td></td><td></td><td></td><td></td></tr>
</table>

Prepared by: _________ Date: __________

Approved by: _________

MAINTENANCE LOG - FORMAT-5

Organisation Name :

Department :

Section :

Name of the machine :

S. No.	Date	Nature of fault	Details of rectification done	Signature of in-charge

MACHINERY AND EQUIPMENT RECORD - FORMAT-6

Organisation Name :				
Department :				
Section :				
History sheet of machinery & Equipment				
Description of equipment				
Manufacturer's address				
Supplier's address				
Order No. and date				
Date on which received				
Date on which installed and placed				
Date of commissioning				
Size: Length x Width x Height				
Weight				
Cost				
Motor particulars	Watts/H.P./	r.p.m:	Phase:	Volts:
Bearings/ spares/ record				
Belt specification				
Lubrication details				
Major repairs and overhauls carried out with dates				

PREVENTIVE MAINTENANCE RECORD - FORMAT-7

Organisation Name :	
Department :	
Section :	
Name of the Machine :	Location of the machine :
Machine Number :	
Model No. & Make :	

Check list for machine inspection

Inspect the following items and tick in the appropriate column and list the remedial measures for the defective items.

Items to be checked	Good working/ satisfactory	Defective	Remedial measures
Level of the machine			
Belt/chain and its tension			
Bearing condition (Look, feel, listen noise)			
Driving clutch and brake			
Exposed gears			
Working in all the speeds			
Working in all feeds			
Lubrication and its system			
Coolant and its system			
Carriage & its travel			
Cross-slide & its movement			
Compound slide & its travel			
Tailstock's parallel movement			
Electrical controls			
Safety guards			

Inspected by

Signature

Name:

Date: Signature of in-charge

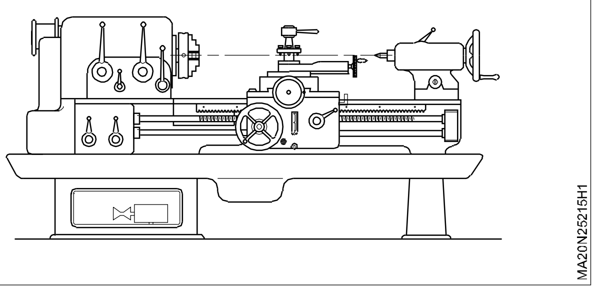
Perform periodic lubrication system on machines

Objectives: At the end of this exercise you shall be able to
- **clean and apply lubrication as recommended by machine builder**
- **maintain the centralized lubricating system**
- **service the power pack system.**

Job Sequence

TASK 1: **Lubrication of centre lathe (Fig 1)**

Fig 1



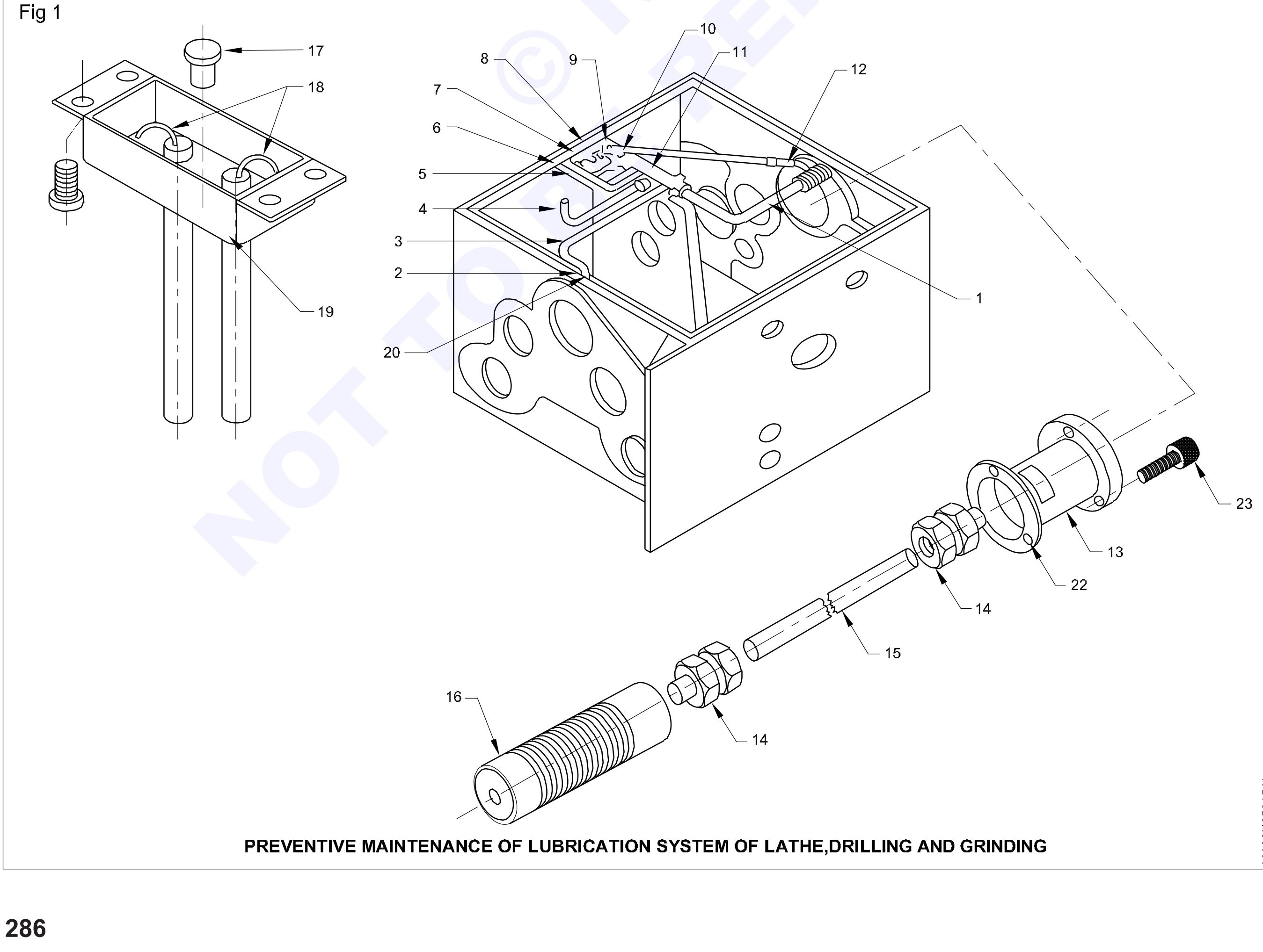
- Clear the machine throughly
- Apply oil with oil cane on sliding surfaces
- Clean, move clean apply oil and move (Carriage, tail stock, compound slide, cross slide and etc).
- Apply all the oil point with oil gun
- Apply grease point with greace gun
- Check the oil level on head stock.
- Check the condition of oil.
- Oil level is low top oil up to the mark.
- Oil condition is not good change the oil.

Use only the machine builder recommended oil or equavalant of that.

— — — — — — — —

TASK 2: **Maintanance of lubricating system (Fig 1)**

Fig 1



PREVENTIVE MAINTENANCE OF LUBRICATION SYSTEM OF LATHE,DRILLING AND GRINDING

- Check the lubrication oil level in the oil tray (part no 19).
- In case of any lubrication problem then remove the socket head cap screw (part no 9,10,23).
- Check the copper tubes through which oil is passing (part nos 1,3,4,5,6,7,15)
- Check the straight connector's condition (part no 2,8,14).
- Remove the suction plug (part no 13).
- Remove the gasket (part no 22).
- Check the strainer and clean it (part no 16).
- Remove the drain screw (part no 21) there is any oil drain is required (Part No 17,18).
- Remove and clean oiled (part no 17) felt wick (part no 18).
- Remove the swivell banjo (part no 12) and manifold (part no 11) clean it and fix it properly.
- Assembly all the parts of lubrication system.
- Refill the recommended grade of lubricant and ensure the oil level.
- Switch on the machine and check the performance of lubrication system.

Lubrication assembly

Sl. No.	Part Name
1	Copper tube
2	Straight connector 1/8" SSp
3	Copper tube
4	Copper tube
5	Copper tube
6	Copper tube
7	Copper tube
8	Straight connector F-SC-4T M8x1
9	Plug M10x1P
10	Soc. Head cAp-screw
11	Manifold M-5
12	Swivel banjo FB-4 1/8" BSP x 4 OD
13	Suction block
14	Straight connector 61/4 x 6mm
15	Copper tubes 6
16	Strainger
17	Oiler 5/16" with cap
18	Felt wick 1/8" x 50 L
19	Oil tray
20	Elbow 1/8 BSP
21	CH HD Screw M6 x 10
22	Gasket
23	Soc. Cap-screw M6 x 20

Ensure the proper oil pumbing by watching through oil glass provided at the gear box of the machine

— — — — — — — — —

TASK 3 **Servecing of power pack system**

- Locate the power pack of the hydraulic system.
- Ensure the system is in 'off' condition.
- Remove the top cover of the power pack after unscrewing the fastening bolts. (Fig 1)
- Pipes are provided below the top cover; place it carefully.
- Place the top cover upside - down with the various elements mounted on it carefully on the work bench. (Fig 2)

Fig 1

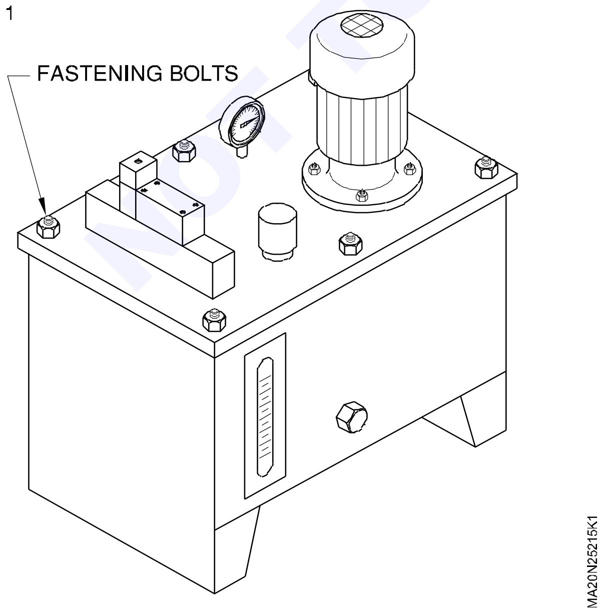
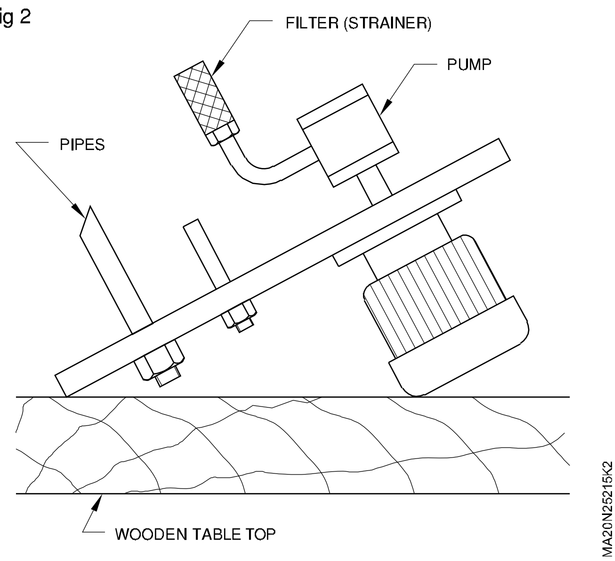


Fig 2



- Identify the various elements, their names and function. Also observe the order of connection.
- Keep the reservoir closed with a plastic cover to avoid contamination. Remove, clean and assemble the inlet fitter. Prepare the power pack for operation. Set the pressure of relief valve.

— — — — — — — —

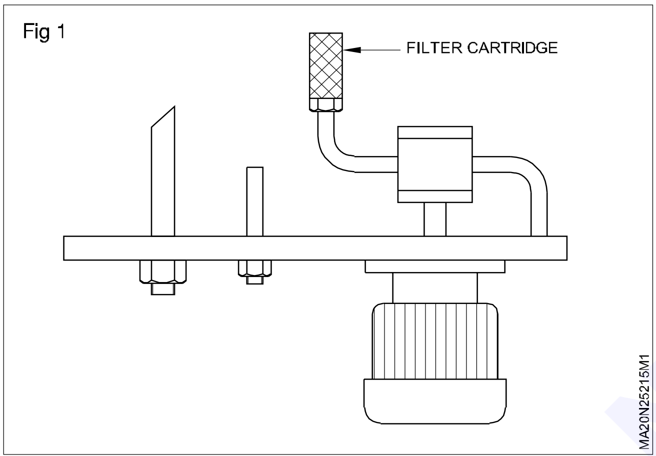
Skill Sequence

Removing, cleaning and assembling of inlet filter (for a closed type of reservoir with removable top cover)

Objective: This shall help you to
- **dismantle, clean and assemble inlet tilter.**

Inlet filter is normally called as suction strainer. Unscrew the inlet cartridge (Fig 1) wipe of the excess sludge collected on the filter.

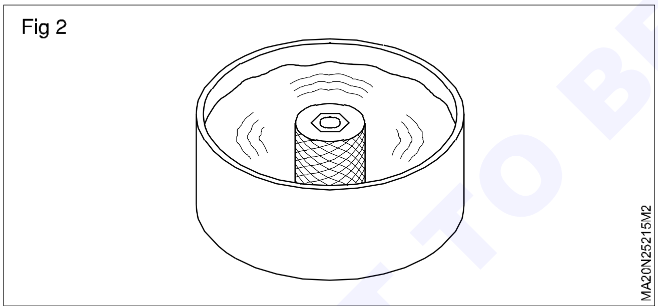
Fig 1



Soak it in kerosence and remove the sludge.

Flush the strainer with clean kerosene (Fig 2)

Fig 2



Blow compressed air on the mesh area.

Clean the mounting area of the strainer.

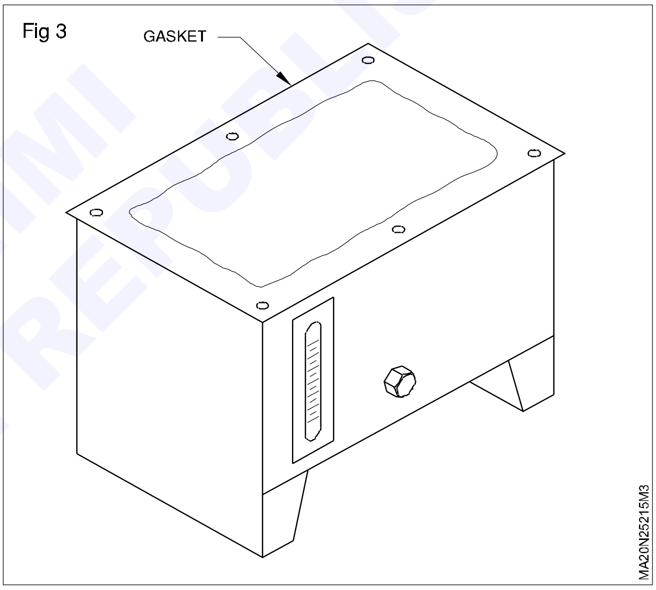
Screw the strainer back in its location.

Strainer/Filters should be clean periodically as per recommendation

Replace with new strainer, care shoud be taken to select the correct strainer.

Checking the gasket of the top, cover of the reservoir for proper seating. (Fig 3)

Fig 3



Place the top cover of the reservoir in its place.

Mount the cover by screwing the fastening screws.

Now inspect the cover for proper seating all over.

Preparing the power pack for an operation

Objective: This shall help you to
- **prepare the power pack for an operation.**

A power pack can perform well only if it is an ideal condition. So before putting on a hydraulic system, the power pack should be checked for it preparedness.

Check the proper mounting of all units.

Check the coupling between motor and pump for freeness, before mounting the top plate.

Check and confirm oil level. (Fig 1)

If oil level is less than the mark, fill the correct grade of oil.

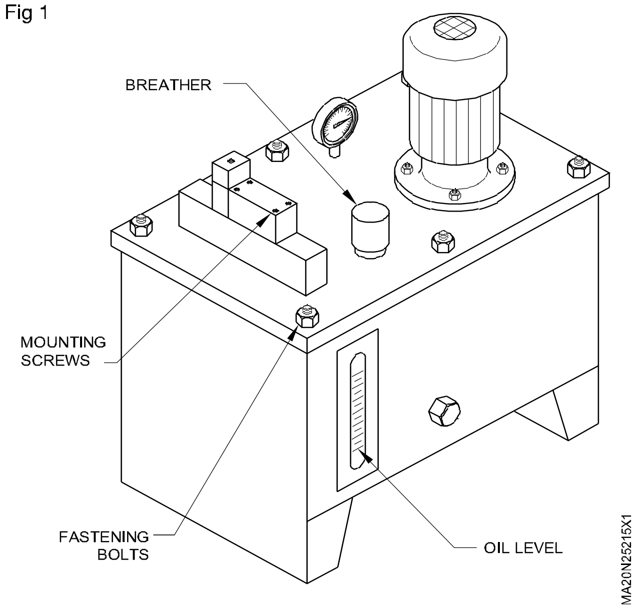
Keep the reservoir clean and clear all unnecessary things around and underneath the resevoir.

Check for proper tightening of all connecting hoses.

Check whether the breather is placed properly.

Oil drain hole is plugged and no oil leakage.

Fig 1



Starting and setting the pressure in a power pack

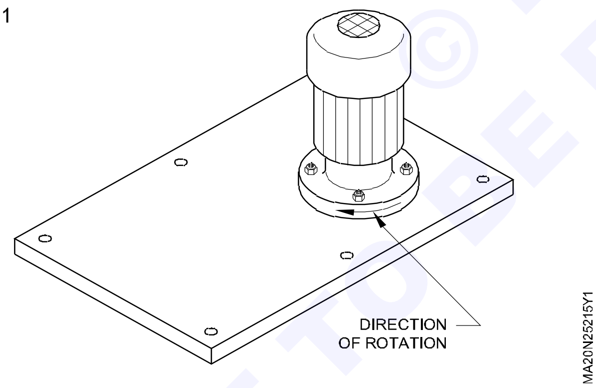
Objective: This shall help you to
- **starting and setting the pressure in a power pack.**

Switch on the electric motor of the power pack.

Confirm no loose ends of pipes exist before switch on the motor.

Observe and confirm the direction of rotaion of the motor's indicated in the motor body. (Fig 1)

Fig 1



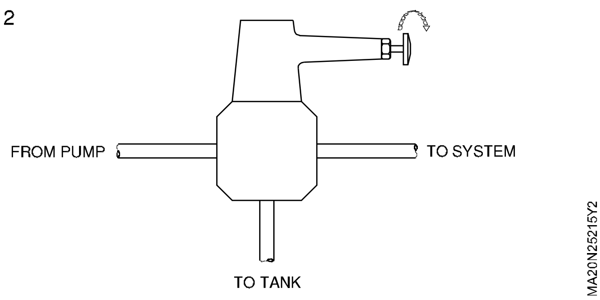
Put off the notor if it rotates inthe opposite direction and call electrician.

Observe the pressure in the pressure gauge.

Now get the required pressure on the pressure relief valve. (Fig 2).

Rotate clockwise to increase pressure and vice versa.

Fig 2



Removal of an inlet filter

Objective: This shall help you to
- **removal of an inlet filter.**

The procedure for removing the inlet filter depends on the construction of the power pack. The suction strainer is usually placed immersed in the oil and locating it needs some experience.

Open type of reservior (Fig 1)

In a open type reservoir, the steps to be followed are

Put off hydraulic system.

Remove the top cover plate.

Keep your hand clean.

Insert your hands inside the oil and locate the suction strainer.

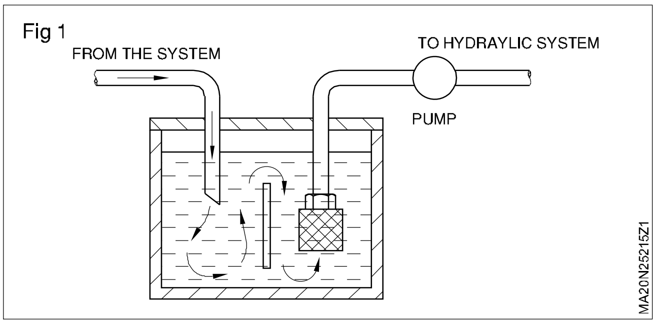
Use a suitable spanner and loose the suction strainer

Clean the strainer using kerosene and blow with compressed air.

Check for damages, if any replace with new filter.

Screw on the clean filter back into position.

Fig 1

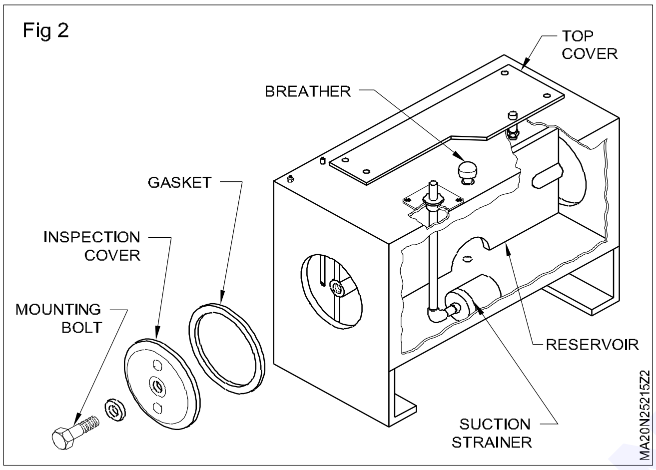


Removal of filter in a closed type of reservoir

Already the procedure for the filter of a closed type of reservoir with removable top cover has been explained. Other type of reservoir is explained below.

All sides welded reservoir (Fig 2)

Fig 2



Put off the hydraulic system.

Drain the oil from the reservoir

Remove the inspection cover after unscrewing mounting bolt.

Locate and unscrew the suction strainer.

Clean, strainer with kerosene and blow it with compressed air.

Clean inside of the reservoir thoroughly.

Screw the suction strainer after inspecting it for damages.

Replace inspection cover and gasket, tighten mounting screws.

Refill the oil in the reservoir after filtering the oil using mesh.

Check for oil leakage through inspection cover. Confirm no leakage of oil.

Check for oil level.

Now the system is ready for use.

Externally mounted suction strainer (Fig 3).

To dismantle this type of suction strainer the steps are as follows

Put off the hydraulic system.

Unscrew both the cap nuts of the lines coming to the filter unit and going out of it.

Hold the filter unit in the benchvice and unscrew the mounting bolt (Fig 4).

Remove the filter insert clean/replace filter insert.

Clean the casing thoroughly.

Fig 3

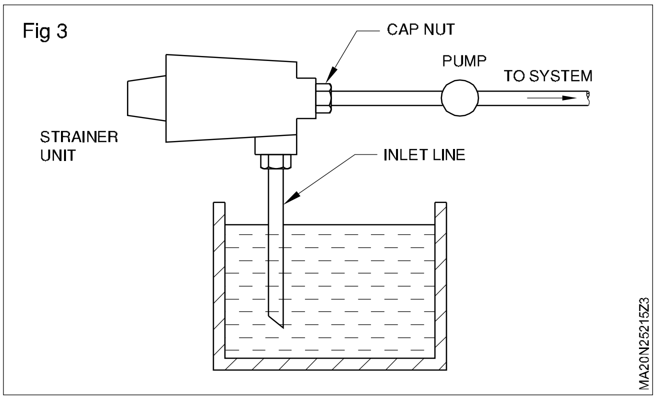
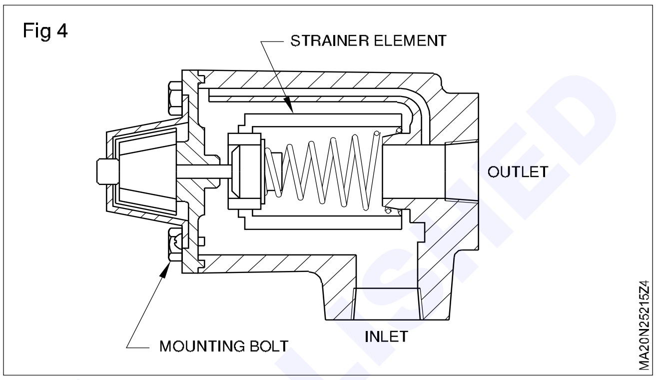


Fig 4



Place the insert and screw the mounting bolt.

Mount the filter unit back in this position.

Confirm proper tightening of connectors.

Checklist for preventive maintenance of centralized lubricating system

Daily requirements

1 Manually activate system and abserve cycle completion, warning devices and system operating pressures.

2 Visually check level of lubricant in reservoir and record. Note the brand in use.

3 Check condition of lubricant visually.

4 Check settings on time clock.

5 Check control cabinet for cleanliness and damage. Make sure the doors are closed and latched.

6 Fill the air line lubricator on pneumatic supply line (pneumatic pumps only) using recommended lubricant.

Weekly or monthly requirements

1 Check and clean filters, strainers or screens as required.

2 Check piping and hoses on primary and secondary distribution system for broken lines, leaks and blockage or kinks.

3 Visually check metering devices for proper operation.

4 Check distribution lines between metering devices and lubrication points for leaks or damage.

Quarterly or Annual Requirements

1 Clean interior of electrical panels and controls.

2 Check for loose electrical connections.

3 Check the pressure gauges and calibrate as needed.

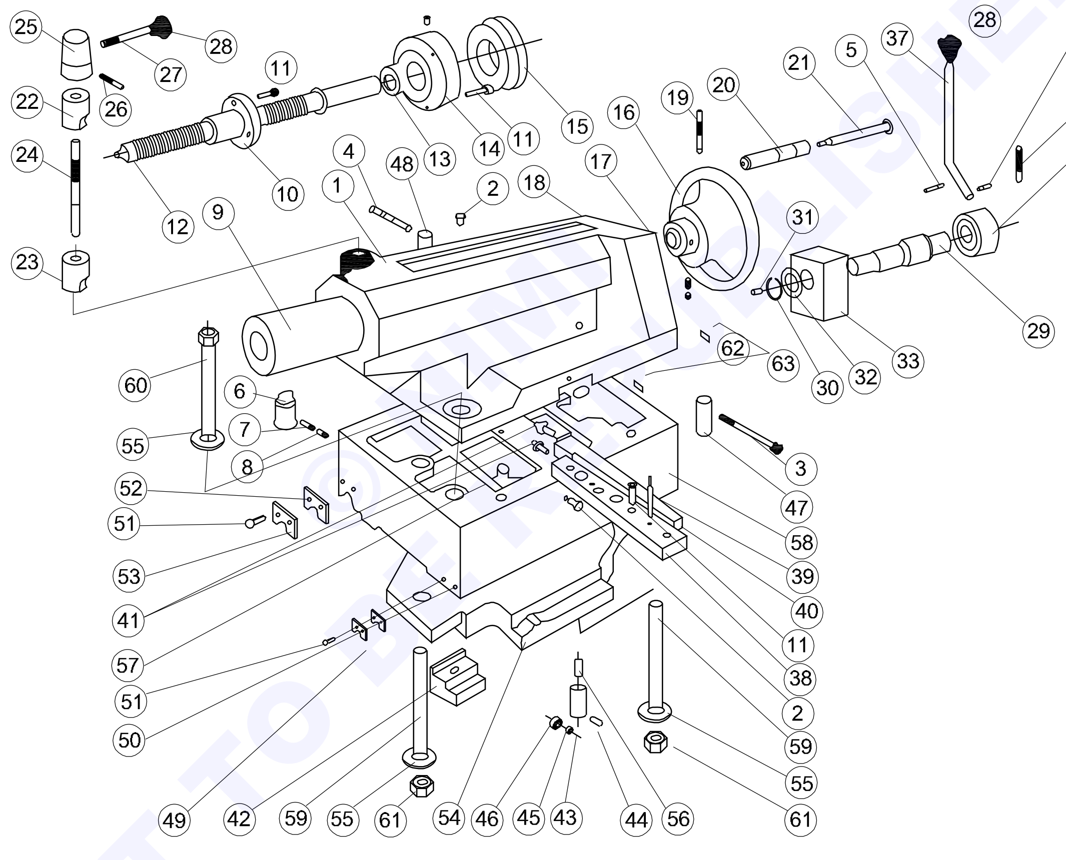
4 Check the operation of warning and safety devices.

Perform repair work

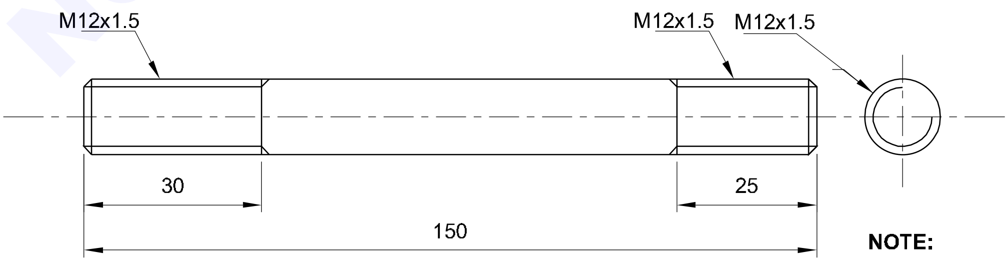
Objectives: At the end of this exercise you shall be able to
- **identify the defects in tail stock assembly**
- **dismantle the tail stock assembly**
- **identify the defective/worn out parts**
- **prepare the defective parts**
- **assemble in the tail stock**
- **check the tail stock performance.**

Fig 1

TASK-1 DISMANTLE THE TAIL STOCK AND KEEP THE PARTS IN A TRAY SEPARATELY AND IDENTIFY THE DAMAGED WORNOUT PARTS



TASK-2 PREPARE A NEW SCREW ROD INSTEAD OF WORNOUT SCREW ROD



NOTE:

STOCK SIZE : Ø16 X 155mm
MATERIAL : Fe310

ALL DIMENSION ARE IN mm

MA20N25216H1

Job Sequence

Identification of defects in a tail stock

- Identify defect in a tail stock.
- Rotate the tail stock hand wheel for moving the spindle.
- Lock the spindle using the locking lever.
- Rotate the tail stock hand wheel and check the spindle movements and locking position. If the spindle is not locked properly it will move.
- Hence, it is known as screw rod spindle lock is not working properly.
- Dismantle the spindle locking unit from the tail stock.
- Prepare the new screw rod instead of defective screw rod.
- Assemble the prepared screw rod instead of wornout screw rod.
- Check the tail stock performance and lock the spindle in the proper position.

Tailstock

Group Assembly Drawing

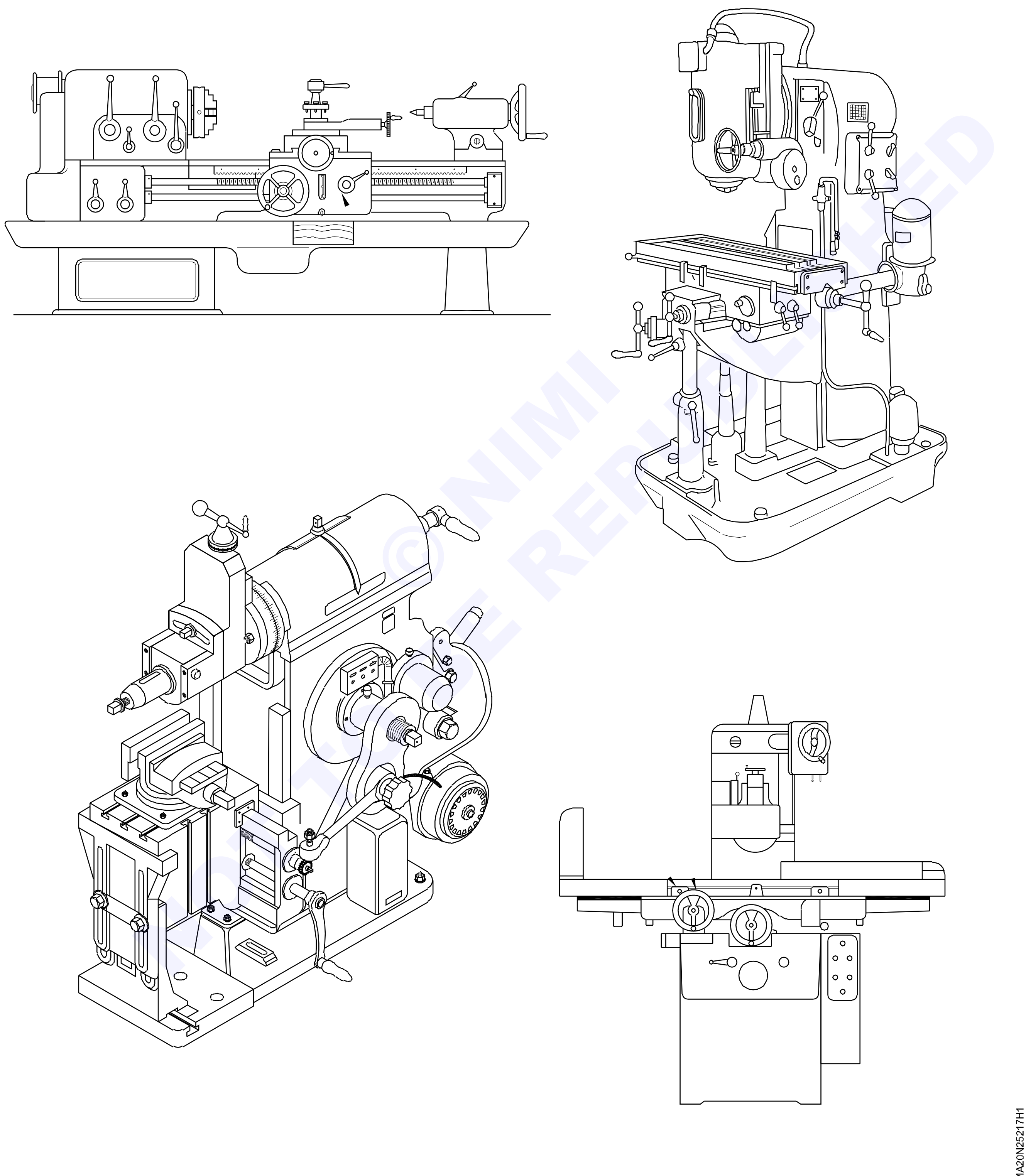
No. On DRG	Qty / Group	Description	Size
1	1	Tailstock	
2	6	Oil nipple	C8
3	1	Hex. Soc. hd. cap. screw	M8 x 100
4	1	Hex. Soc. hd. cap. screw	M8 x 60
5	1	Cyl.pin	10 x 50
6	1	Key	
7	1	Grub Scr. ‘G’	M8 x 16
8	1	Grub Scr. ‘A’	M8 x 10
9	1	Sleeve	
	1	Sleeve (with tenon slot)	
10	1	Nut	
11	10	Hex. Soc. hd. cap. screw	M8 x 25
12	1	Screw	
13	1	Th. ball bearing (51205)	25/47 x 15
14	1	Flange	
15	1	Graduated collar	
16	1	Hand wheel	
17	3	Compression spring	
18	3	Steel Ball Class V	5/16” class V
19	1	Taper pin	6 x 60
20	1	Handle	
21	1	Handle rod	
22	1	Clamp piece	
23	1	Clamp piece	
24	1	Screw rod	
25	1	Cap	
26	1	Taper pin	6 x 50
27	1	Handle rod	
28	2	Knob	
29	1	Eccentric shaft	
30	1	External circlip	A 30
31	1	Cyl. plug	6
32	1	Spacer	
33	1	Clamp nut	

No. DRG	Qty / Group	Description	Size On
34	1	Cap	
35	1	Taper pin	6 x 80
36	1	Taper pin	4 x 30
37	1	Handle rod	
38	1	Tenon	
39	2	Int. Thrd. taper pin	8 x 50
40	1	Gib	
41	2	Spec. screw	
42	1	Clamp piece	
43	3	Bearing holder	
	3	Hex. soc. grub screw	M6 x 10
44	3	Spec. pin	
45	3	Needle roller bearing DL-810	8/14 x 10
46	3	Bearing bush	
47	1	Shaft	
48	1	Shaft	
49	2	Wiper	
50	2	Plate	
51	8	Slotted ch. hd. scr. 'A'	M6 x 18
52	2	Wiper	
53	2	Plate	
54	1	Clamp plate	
55	3	Spec. washer	
56	3	Compression spring	
57	3	Spec. grub screw	
58	1	Tailstock base (For NH22)	
	1	Tailstock base (For NH26)	
	1	Tailstock base (For NH32)	M20 x 130
59	2	Stud 'B' (For NH22)	
	2	Stud 'B' (For NH26)	M20 x 170
	2	Spec stud (For NH 32)	
60	1	Hex. bolt (For NH22)	M20 x 140
	1	Hex. bolt (For NH26)	M20 x 180
	1	Hex. bolt (For NH32)	M20 x 220
61	2	Sef locking nut	

Perform the routine maintenance with check list

Objective: At the end of this exercise you shall be able to
- **perform the routine maintenance of general machine shop machines with check list.**

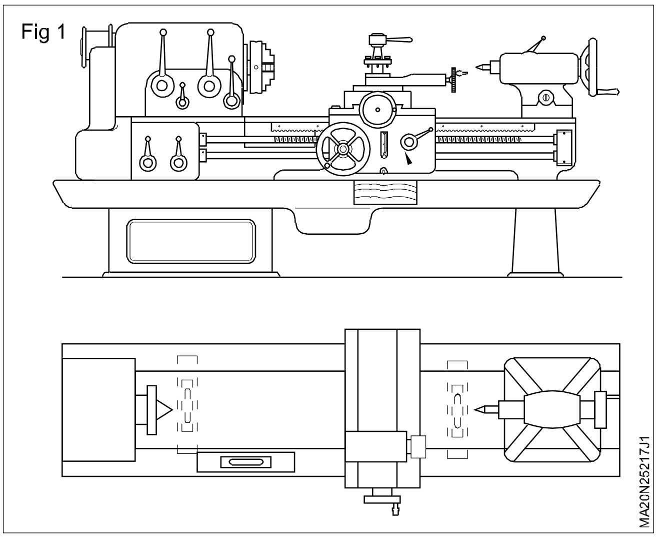
Fig 1



Job Sequence

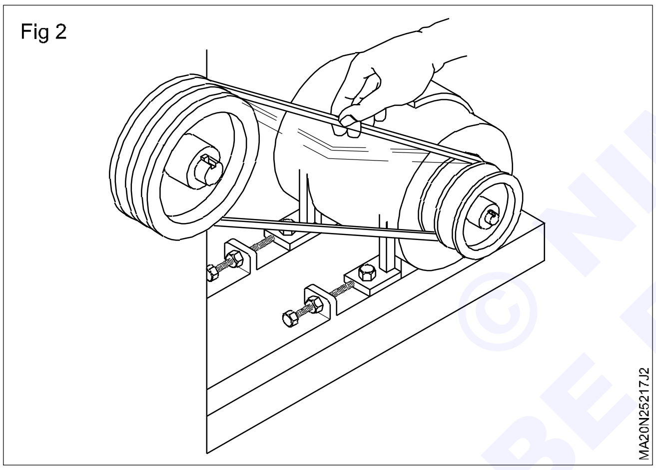
- Check the level of the lathe with a spirit level and adjust using wedges. (Fig 1)

Fig 1



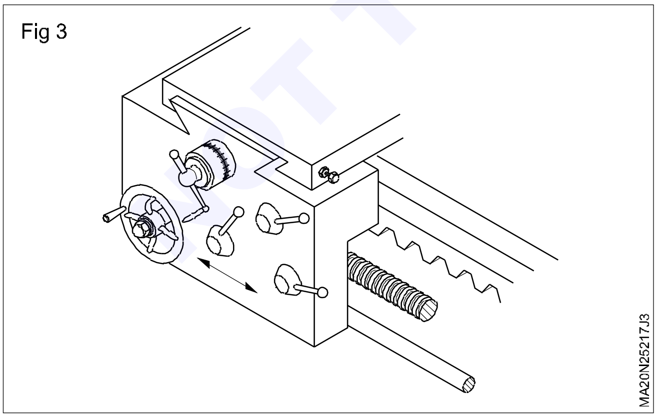
- Check the tension of the belt and adjust. (Fig 2)

Fig 2



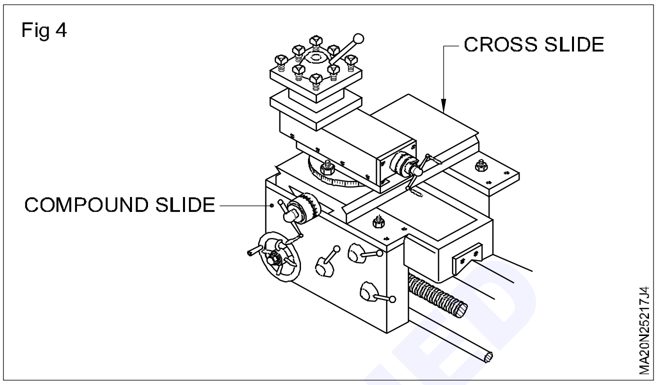
- Check the free movement of tailstock over the bed.
- Check the movement of the carriage of the lathe. (Fig 3)

Fig 3



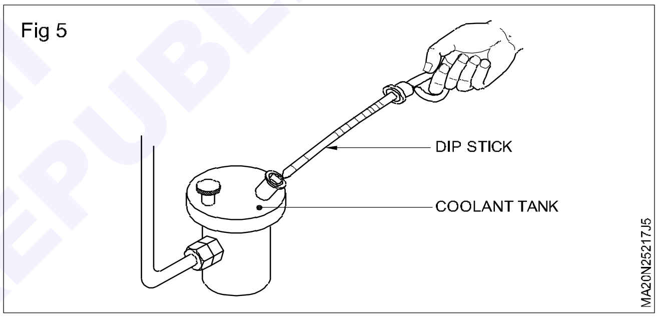
- Run the machine on different spindle speeds and check the speed.
- Engage the power feed and check the longitudinal and transverse feed movements.
- Check the function of clutches by operating the clutch lever.
- Check the movement of the cross-slide and the compound slide. (Fig 4)

Fig 4



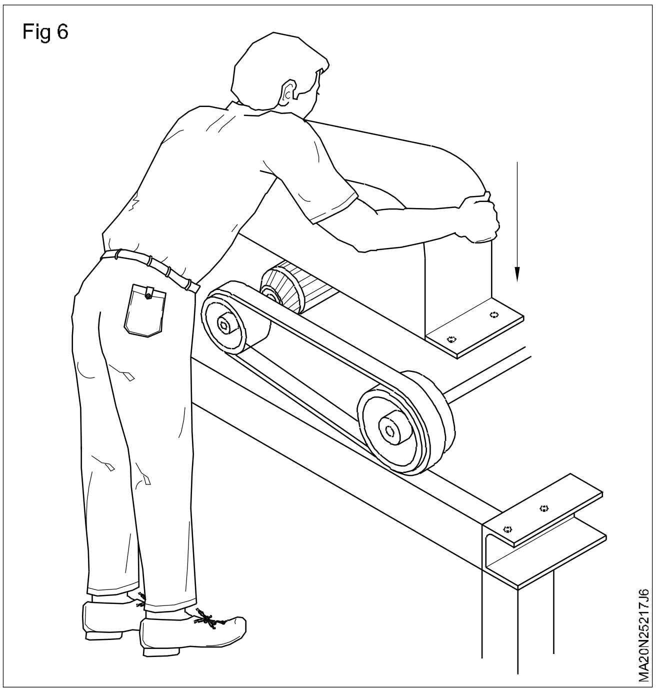
- Check the oil level and the functioning of the lubricating pump.
- Check the coolant level and the functioning of the coolant pump. (Fig 5)

Fig 5



- Check the safety guards. (Fig 6) and ensure they are in postion.
- Inspect the machines as per check list and record it accordingly.

Fig 6



Inspect the following items and tick in the appropriate column and list the remedial measures for the defective items.

Items to be checked	Good working/Satisfactory	Defective	Remedial measures carried out
Level of the machine			
Belt and its tension			
Bearing sound			
Driving clutch and brake			
Exposed gears			
Working in all the speeds			
Working in all feeds			
Lubrication system			
Coolant system			
Carriage & its travel			
Cross-slide & its movement			
Compound slide & its travel			
Tailstock's parrallel movement			
Electrical controls			
Safety gaurds			

Preventive Maintenance Programme **Annexure I - B**

Name of the Machine : Location of the machine:

Machine Number :

Model No & Make :

CHECK - LIST FOR MACHINE INSPECTION (Milling machine)

Inspect the following items and tick in the appropriate column and list the remedial measures for the defective items.

Items to be checked	Good working / satisfactory	Defective	Remedial measures
Level of the machine			
Belt and its tension			
Bearing sound			
Driving clutch			
Working in all the speeds			
Working in all feeds			
Lubrication system			
Coolant system			
Table travel			
Cross - slide & its movement			
Saddle & its travels			
Knee up & down movement			
Electrical controls			
Safety gaurds			

Inspection by

Signature

Name :

Date : Signature of in - charge

Preventive Maintenance Programme **Annexure I - C**

Name of the Machine : Location of the machine:

Machine Number :

Model No & Make :

CHECK - LIST FOR MACHINE INSPECTION (Shaping)

Inspect the following items and tick in the appropriate column and list the remedial measures for the defective items.

Items to be checked	Good working / satisfactory	Defective	Remedial measures
Level of the machine			
Belt and its tension			
Bearing sound			
Table elevation			
Exposed gears			
Working in all the speeds			
Working in all feeds			
Lubrication system			
Ram and its travel			
Saddle & its movement			
Tool head angle rotation			
Stroke length adjustment			
Position of the stroke			
Safety guards			

Inspection by

Signature

Name :

Date :

Signature of in - charge

Preventive Maintenance Programme **Annexure I - D**

Name of the Machine : Location of the machine:

Machine Number :

Model No & Make :

CHECK - LIST FOR MACHINE INSPECTION (Surface grinding)

Inspect the following items and tick in the appropriate column and list the remedial measures for the defective items.

Items to be checked	Good working / satisfactory	Defective	Remedial measures
Level of the machine			
Belt and its tension			
Bearing sound			
All feed movements			
Condition of bellows			
Dust collecting system			
Lubrication system			
Coolant system			
Condition of magnetic chuck			
Electrical controls			
Safety guards			

Inspection by

Signature

Name :

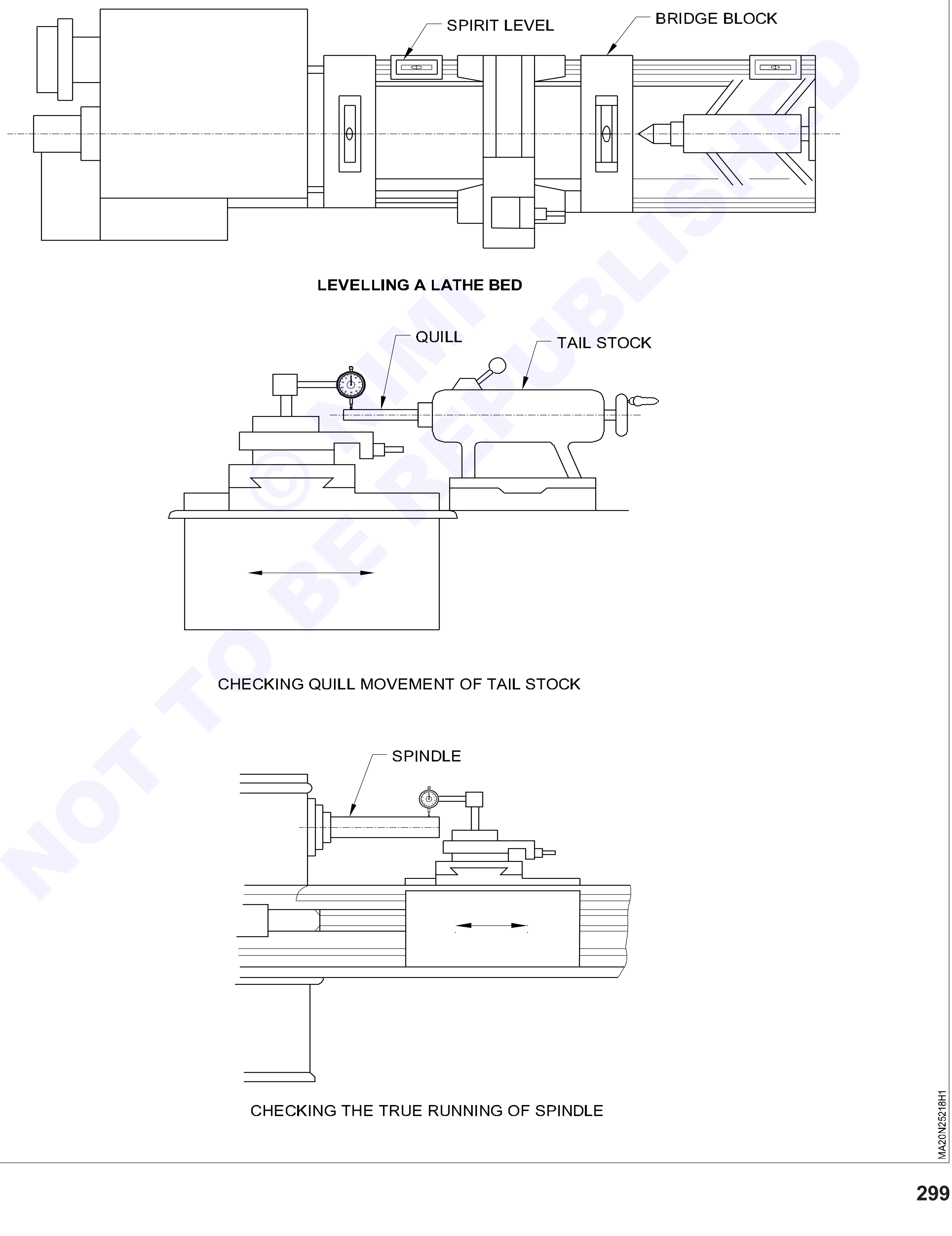
Date :

Signature of in - charge

Inspection of machine tool such as alignment and levelling

Objectives: At the end of this exercise you shall be able to
- **check the level of a centre lathe**
- **check the true running of a lathe spindle**
- **check the alignment of the main spindle and the tailstock spindle of a lathe**
- **check the parallelism of the tailstock sleeve with respect to bedways.**
- **perform practical test on turned component.**

Fig 1



Skill Sequence

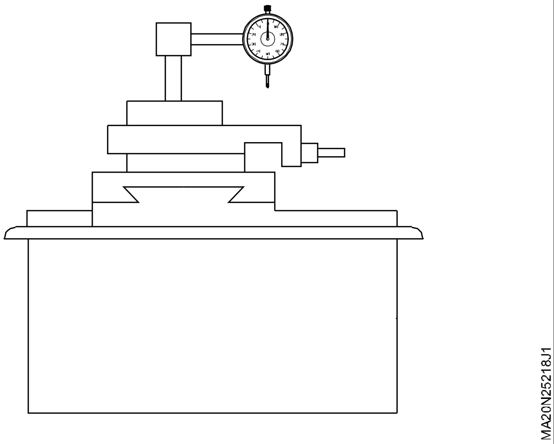
Checking quill movement of tailstock

Objective: This shall help you to

- **test the tailstock sleeve movement relative to the carriage guideways.**

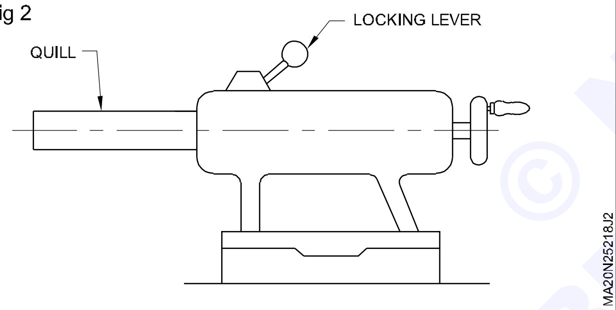
Fix the dial gauge on the carriage. (Fig 1)

Fig 1



Project the quill of the tailstock to the maximum extent possible and lock it. (Fig 2) Check the quill in the vertical and horizontal positions by a dial test indicator.

Fig 2

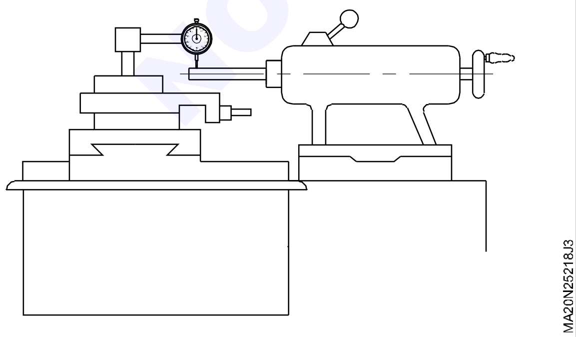


Clamp the quill during each measurement. If it is not clamped it will affect the measurement.

Place the dial plunger to contact over the free end of the quill in the vertical plane. (Fig 3)

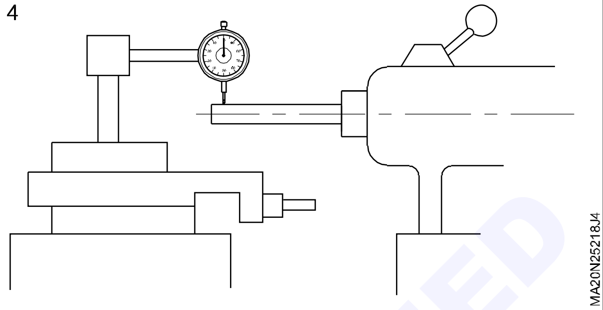
Ensure that the dial is set at the topmost point of the quill.

Fig 3



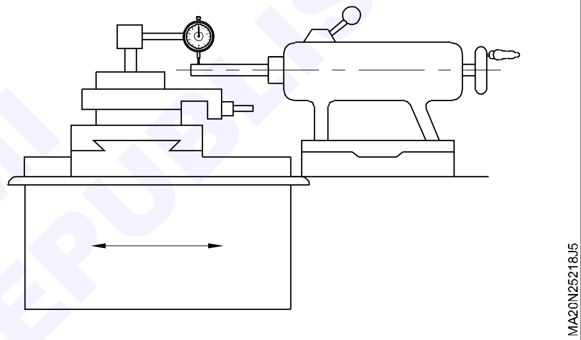
Set the dial at the zero position. (Fig 4)

Fig 4



Move the carriage slowly towards the entire length of the quill. (Fig 5)

Fig 5



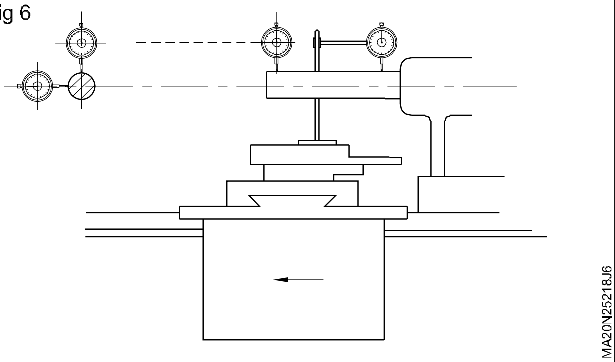
Note the dial reading at the extreme end of the quill.

Verify the deflection of the dial reading and compare the value with the test chart supplied. (IS: 6040)

For checking in the horizontal plane, set the dial horizontally and repeat the above procedure. (Fig 6)

Fix the test mandrel into the tailstock spindle. Repeat the same procedure to test the accuracy of the tailstock spindle bore in the vertical and horizontal positions as shown in the Fig 6.

Fig 6



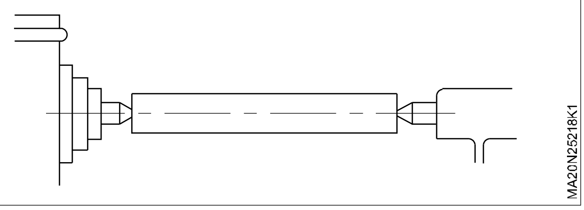
Checking the alignment of the main spindle and the tailstock spindle of a lathe

Objective: This shall help you to
- **to check the alingment of the spindle and the**
- **tailstock sleeve.**

Insert a hollow test mandrel (300 to 500 mm long) in between the centres. (Fig 1)

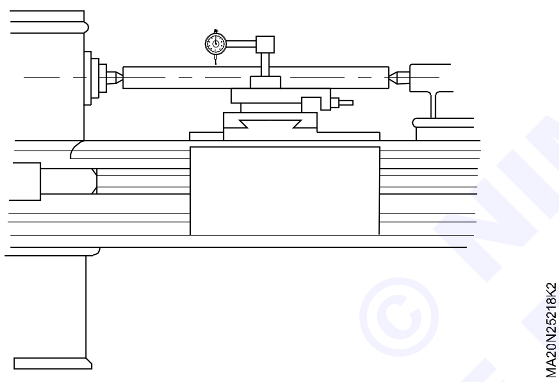
Ensure that the spindle bearing is at its working temperature.

Fig 1



Fix the dial gauge on the saddle, the plunger touching a position of the mandrel and set it to zero.(Fig 2)

Fig 2

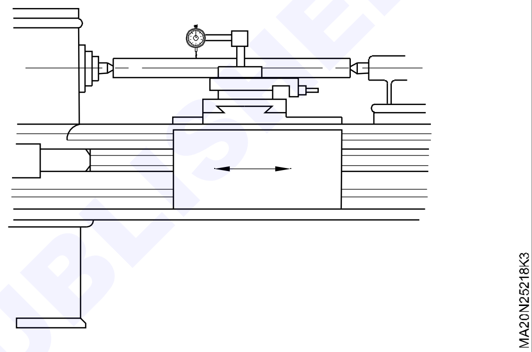


Move the carriage from one end to the other end of the mandrel to check the mandrel is in correct alignment in the horizontal position.

Rest the dial plunger at right angles (radially) to the surfaces to be tested.

Set the dial plunger at the top of the mandrel and move the saddle along the bed slowly to the entire length of the mandrel. (Fig 3)

Fig 3



Observe the reading of the dial as the saddle moves along the beds and note for variation, if any.

The tailstock centre must be higher than the spindle centre within the permissible limit.

Verify the deflection of the dial gauge reading and compare the value with the test chart. (IS: 6040)

Checking the true running of a spindle

Objective: This shall help you to
- **test the true running of a lathe spindle with a test mandrel.**

Locate the taper shank of the test mandrel in the spindle taper.

Hold a dial gauge, stationary in the carriage, its plunger contacting the mandrel near its free end (Fig 1) and set it to '0' position.

Rest the dial gauge plunger at right angles (radially) to the surface to be tested.

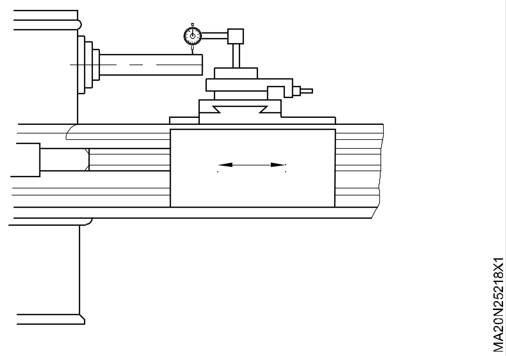
Rotate the spindle along with the mandrel slowly by hand.

Observe and note the reading of the dial gauge.

Move the dial gauge near the spindle nose. Rotate the spindle along with the mandrel slowly by hand and note the reading.

Take readings of the dial gauge while the spindle is slowly rotated. Verify the deflection of the dial reading and compare the value with the test chart. (IS: 6040)

Fig 1

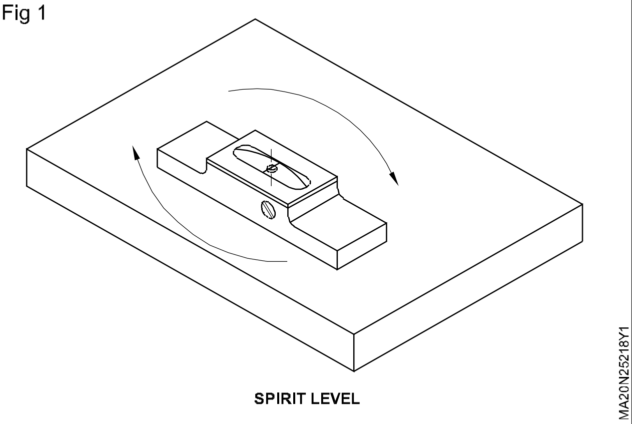


Adjustment of the spirit level with the plane surface

Objective: This shall help you to
- **adjust the spirit level with the plane surface.**

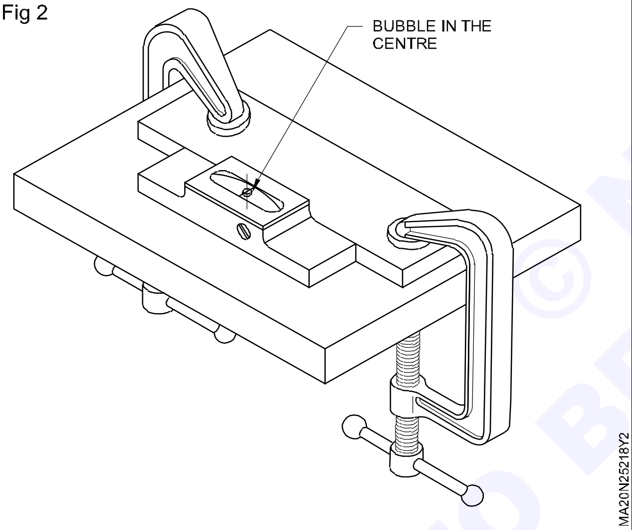
Move the spirit level on the plane surface until the bubble is in the centre of the scale.(Fig 1)

Fig 1



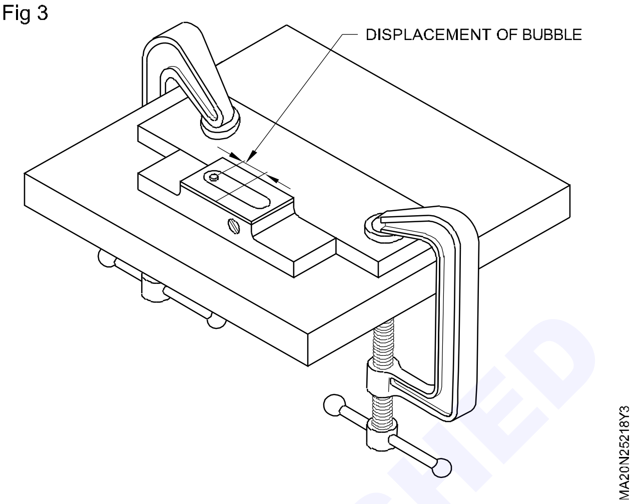
Place a straight edge against the level and clamp to the plate. (Fig 2)

Fig 2



Turn the level through 180° (end for end) and place against the straight edge and note the displacement of the bubble. (Fig 3)

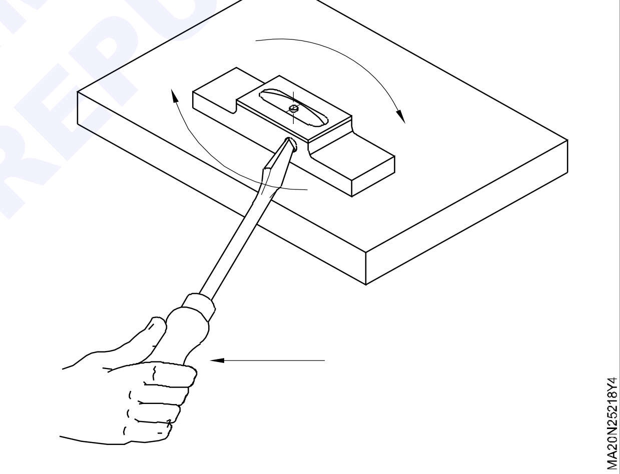
Fig 3



Adjust the vial to half of the total displacement of the bubble. (Fig 4)

Repeat the above sequence until the level is turned end for end without displacement of the bubble.

Fig 4



Level the lathe bed

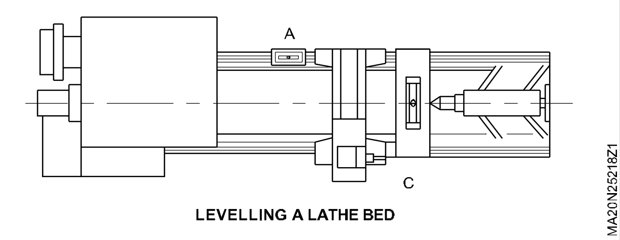
Objective: This shall help you to
- **level the lathe horizontally with the help of a spirit level.**

Position the carriage in the middle of the bed.

Keep the spirit level on the rear slideway (i.e. the slideway opposite the operator's side) longitudinally at the position 'A'.(Fig 1)

Keep the second spirit level transversally at the position 'C'. (Fig 1)

Fig 1

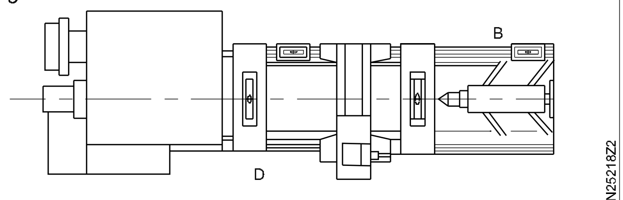


Take the readings of both the spirit levels.

Adjust the level of the bed till both the spirit levels show the same readings.

Keep the spirit levels longitudinally and transversally at positions 'B' and 'D'. (Fig 2)

Fig 2

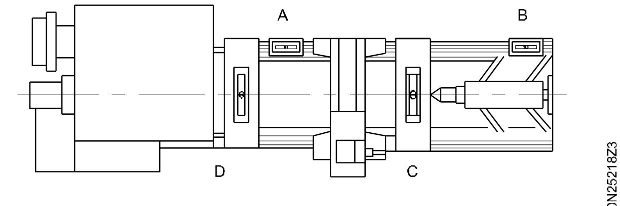


LEVELLING A LATHE BED

Adjust the bed till both the spirit levels show the same readings.

Repeat the sequence of operation till both the spirit levels show the same reading in all the positions A, B, C & D. (Fig 3)

Fig 3

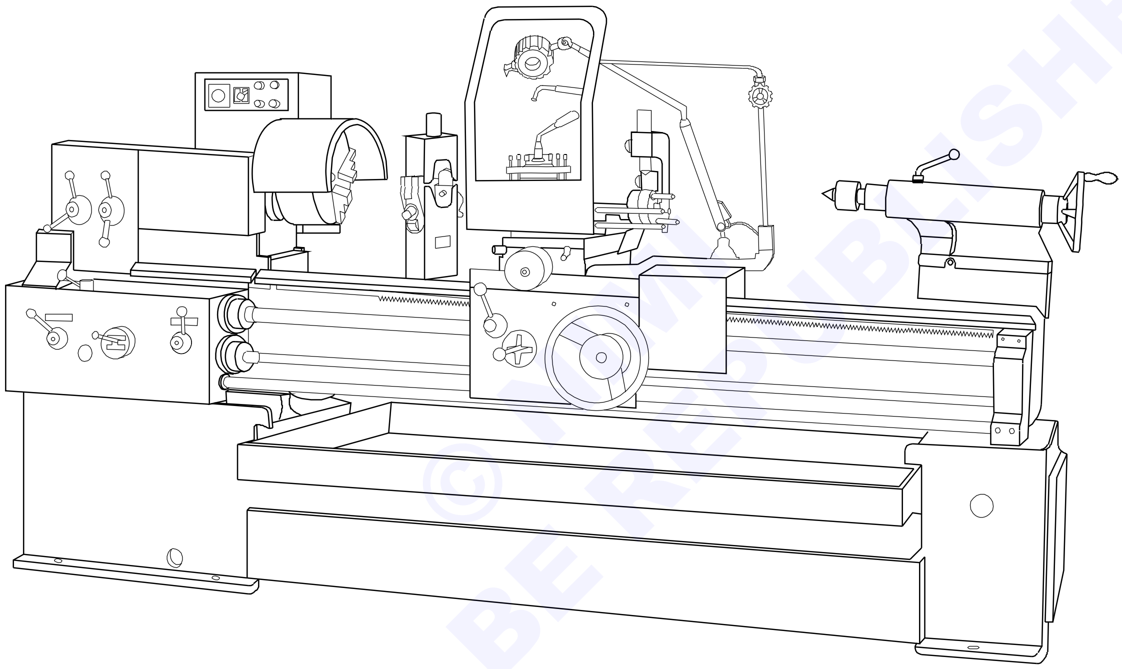


LEVELLING A LATHE BED

Accuracy testing of machine tools such as geometrical parameters

Objectives: At the end of this exercise you shall be able to
- **check the level of a centre lathe**
- **check the true running of a lathe spindle**
- **check the alignment of the main spindle and the tailstock spindle of a lathe**
- **check the parallelism of the tailstock sleeve with respect to bedways.**
- **perform practical test on turned component.**

Fig 1



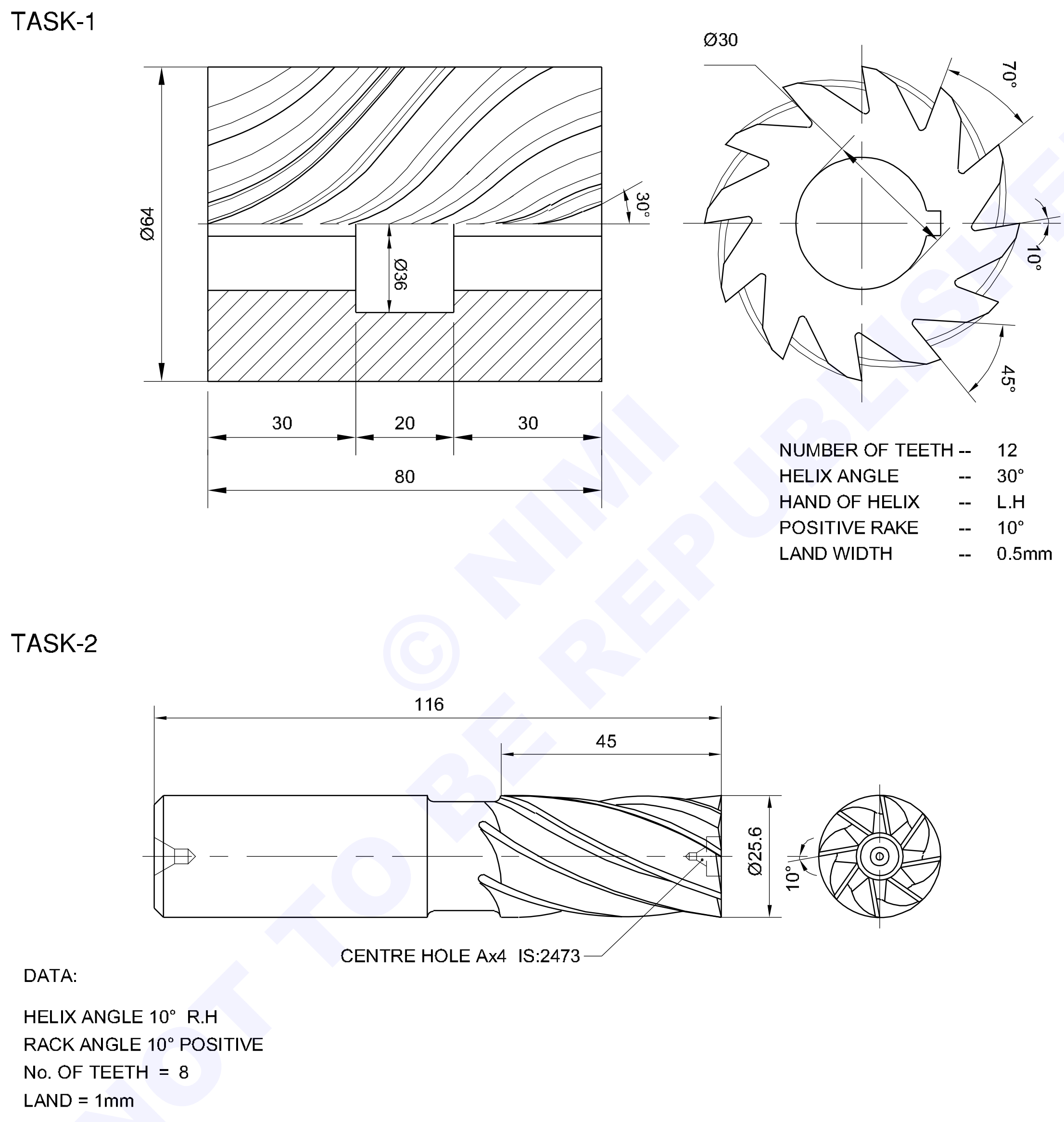
Job Sequence

Trainees may be asked to write the job sequence and record the tested values for

1 Levelling
2 True running of spindle
3 Alignment of mainspindle and the tailstock spindle
4 Parallelism of tailstock sleeve with respect to bedways
5 Test the machine by producing component.

Milling helical slab cutter and end mill cutter

Objectives: At the end of this exercise you shall be able to
- **mill helical grooves, to the full length**
- **mill secondry relief or clearance to the full length.**



PREMACHINED BLANK AS PER DRAWING

ALL DIMENSIONS ARE IN mm

NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.
1	Ø 65 x 85	-	Fe310	-	2	2.6.220
1	Ø 36 x 120	-	Fe310	-	1	2.6.220

SCALE 1:1	**MILLING HELICAL SLAB & END MILL CUTTER**	DEVIATIONS MEDIUM IS:2102	TIME
		CODE NO. MA20N26220E1	

Job sequence

TASK 1: **Mill slab milling cutter 12T 30° LH**

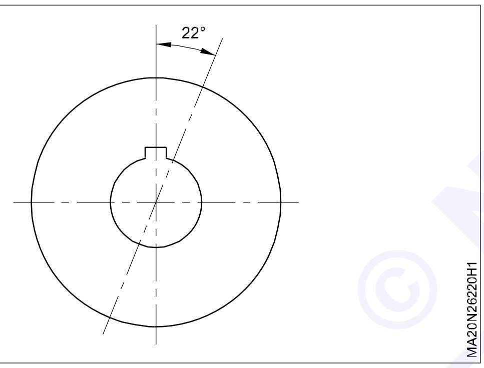
Milling helical flutes

- Clean the surface of the universal milling machine table.
- Mount the dividing head on the table and align its axis in parallel with the table.
- Connect the dividing head table screw using gear trains for required lead.
- Arrange the indexing movement for indexing 12 divisions.
- Swivel the table to 30° pushing the left side in.
- Mount the job blank between centres with a mandrel.

Ensure that the blank will not loosen during milling

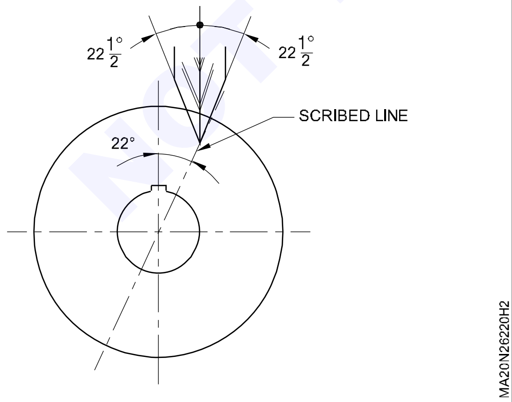
- Apply blue at the face of the blank at the tailstock end and scribe the radial line.
- Rotate and bring crank radial line through 22° with vertical axis as shown in the sketch i.e. 12° + 10° for positive rake. (Fig 1)

Fig 1



- Mount double equal angle cutter 45° on the arbor.
- Manipulate the position of the job with reference to the cutter at the tailstock end of the job.
- Start the machine and raise the table to cut a groove such that one side of the cutter coincide with the scribed line. (Fig 2)

Fig 2

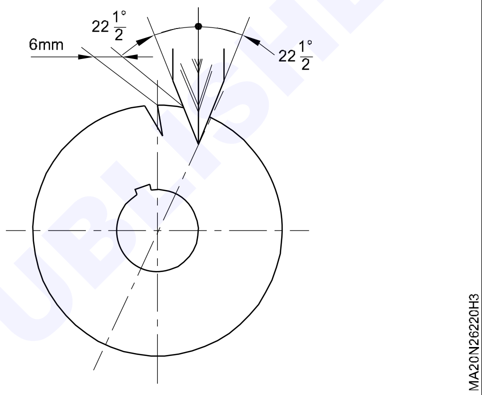


- Mill the groove to a depth of 10mm and to full length by feeding the blank through the crank of the dividing head and set the vertical feed dial to '0'.

Ensure that the back stopper pin is disengaged from the index plate.

- Lower the table by 1mm and bring back the blank to starting position and index for the next tooth.
- Raise the job to '0' on vertical feed dial.
- Mill the 2nd tooth groove and measure the land width is 6 mm on the 1st tooth. (Fig 3)
- Repeat the above steps and complete the remaining teeth.

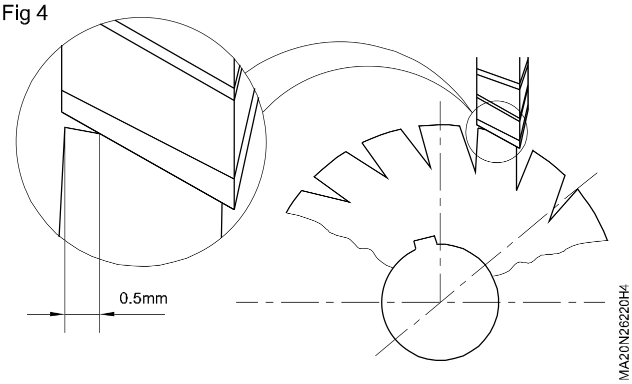
Fig 3



Milling secondary clearance

- Mount unequal angle cutter 70°(58° +12°)on the arbor.
- Index further 25° in the same direction and adjust the sector arm to this new position.
- Manipulate the blank with reference to the 58° side of the cutter and mill the secondary clearance to the extent that the land width is maintained at 0.5 mm. (Fig 4)
- Set vertical feed dial to '0'.
- Index for the next tooth and mill the secondary clearance for all the 12 teeth.
- Remove the job and deburr.

Fig 4



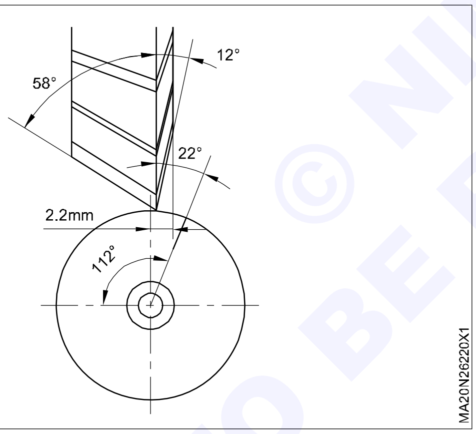
TASK 2: **Milling end mill cutter**

Milling flutes to form peripheral cutting edges

- Select a universal milling machine.
- Clean the table.
- Mount the dividing head and foot stock and align the spindle axis with reference to the machine table and lock it.
- Swivel the table 10° degree anticlockwise. (Push the R.H. end)
- Mount a double unequal angle cutter 58° +12° in the middle of the arbor.
- Hold the job between centres.
- Determine the required gear train and connect it with the index-head auxiliary spindle to lead screw of table.
- Position the 12° side cutting edge of the cutter in line with the vertical axis of the job at the tailstock end.
- Offset the job to a distance of 2.20 mm from the centre line.

  Rule: offset = Rake angle x cutter dia x 0.0087
- Mark reference line at centre using a scribing block and index through 12°. (Fig 1)

Fig 1

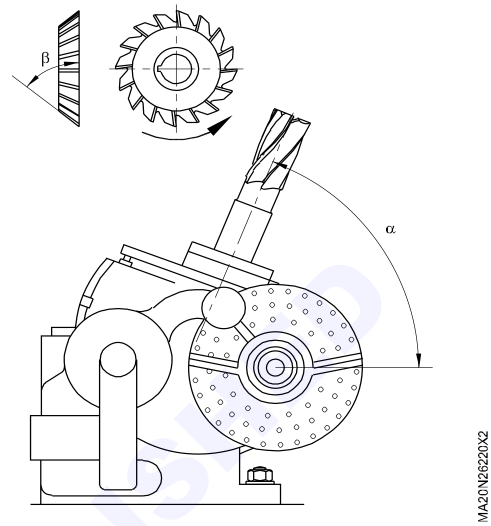


- Start the machine and raise the table for partial depth (2 mm) and manipulate vertical feed and cross feed and mill the flute to the entire length of the body.
- Bring back the job to the starting position.
- Crank for 5 revolutions to index the next tooth.
- Mill the 2nd flute to full depth and maintain the land width to 1 mm.
- Bring back the job to starting position and index for 3rd flute.
- Mill the 3rd flute to full length.
- Repeat the above steps and complete the 4th, 5th, 6th, 7th, 8th and 1st flutes.

Milling flutes to form end cutting edges

- Mount the chuck on the dividing head, spindle and hold the job
- Tilt the index head to 90° and set the face of one tooth straight and parallel to the teeth of the single angle cutter.
- Mount single angle cutter 70° R.H. on the arbor.
- Tilt the index head to angle (68°.40) and lock it.(Fig 2)

Fig 2

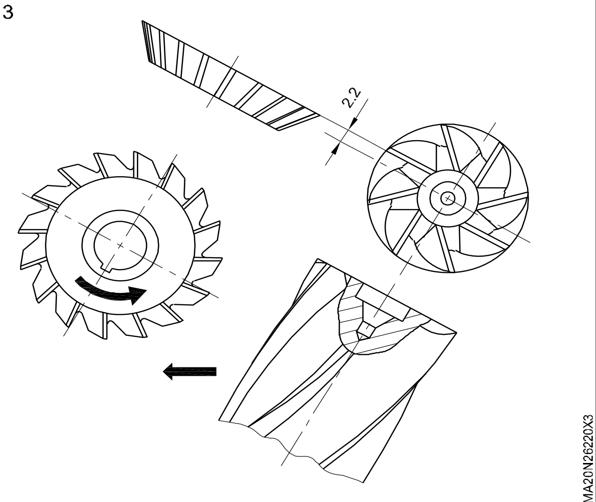


$$\text{Cos } \alpha = \text{Tan } \frac{360^\circ}{N} \text{ x Cot of cutter angle } (\beta)$$

where N = No. of teeth on the job.

- Bring the job near the cutter and adjust such that the straight side of the cutter coincides with the peripheral cutting edge of the end mill already milled. (Fig 3)
- Start the machine and raise the job to 1 mm and mill the lip of the job. (Fig 3)
- Bring back the job to starting position.
- Index for 2nd tooth. (5 revolutions)
- Raise the job gradually and mill the groove to a depth till you get the land width of 1 mm. (Fig 3)
- Repeat the above steps to mill the grooves 3rd, 4th, 5th, 6th, 7th, 8th and 1st.
- Remove the job and deburr.

Fig 3



Skill sequence

Cutting teeth on slab milling cutter

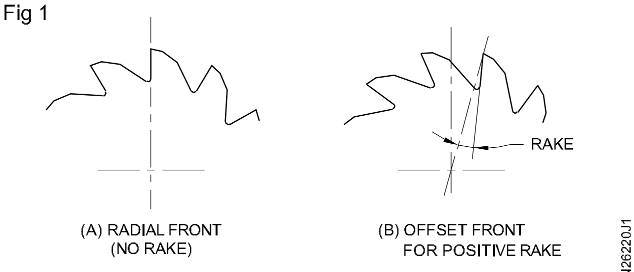
Objective: This shall help you to
- **cut helical flutes on slab milling cutters,end mills, and counter bores.**

Helical milling operation is very common in most machine shops. This offers excellent practice to the learner in cutting the flute on slab milling cutters, end mills and counterbores. This operation involves less mathematics, and the some knowledge of helical milling with more skill in the set up.

The common form of flute of a slab milling cutter is shown in Fig 1. 1(A) Radial front (no rake)

1(B) offset front for positive rake)

Fig 1

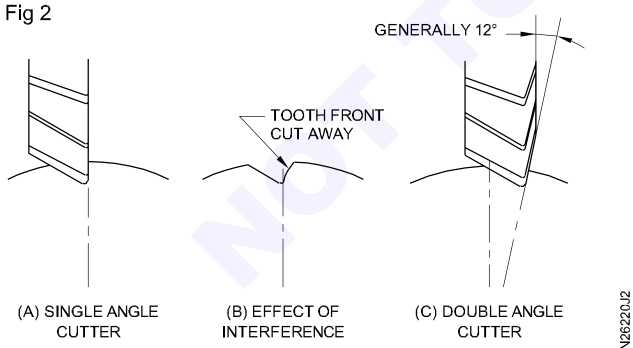


FLUTED CUTTER TOOTH

Select a double, unequal angle cutter to suit the given form of flute.

> **Refer to job drawing for angle and shape of flute and hand of helix i.e. RH or LH.**
>
> **If a single angle cutter is used to produce the form of flute on the cutter blank, then the cutting edges formed will be cut away due to interference occurred during milling. (Fig 2) To avoid this defect a double unequal angle cutter is always selected for fluting operations.**

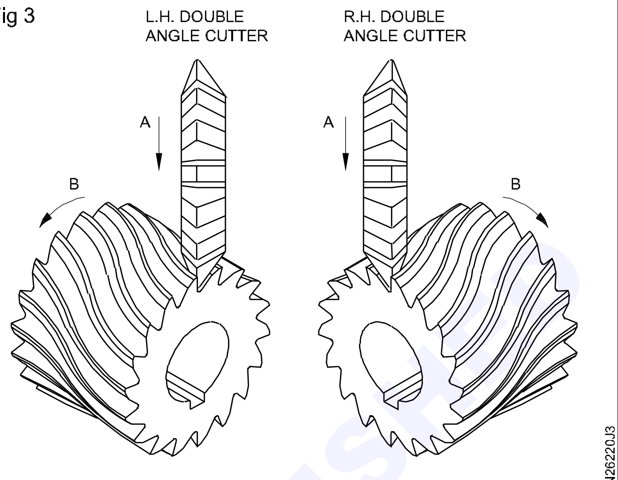
Fig 2



CUTTING FLUTES

Fig 3 shows the difference in the setting of the table and the shape of the cutter used when milling RH & LH helical flutes. Both views are shown as seen from the footstock end of the table. Arrow A denotes the direction of the cutter and arrow B the direction of the rotation of the blank as the helical groove is being milled.

Fig 3



Having selected the right type of cutter for the fluting operation, proceed as follows:

Lubricate the milling machine, dividing head and feed mechanism.

Mount the work (cutter blank) on the mandrel, tighten the nut securely.

Wipe the bore of the spindle and the shank of the machine arbor and mount securely with a drawn in bolt.

Set the index head and sector arms for the indexing movement required.

> **Disengage the stop pin at the back of the index plate.**

Calculate the lead of helix, arrange change gears and test by using hand feed to be sure that the helical mechanism operates freely. (Lead = Circumference of the job X Cotangent of helix angle)

> **Check the movement and direction of rotation of the work (cutter blank) for the given hand of helix ie. RH or LH. (Fig 3) If necessary add one idler to change the direction of rotation.**

Swing the table to the helix angle, tighten lightly in this position (temporarily), check for a minimum. 12 mm clearance between table and column.

Mount the cutter blank between the index centres.

Locate and mount the double unequal angle cutter on the machine arbor, and swing the table to zero (straight) again.

Adjust the cutter to the centre of the blank. (Fig 4A)

Apply prussian blue on the end of the blank where lines are to be scribed and scribe centre (radial) line.

Index for one tooth space and scribe another line. (Fig 4B)

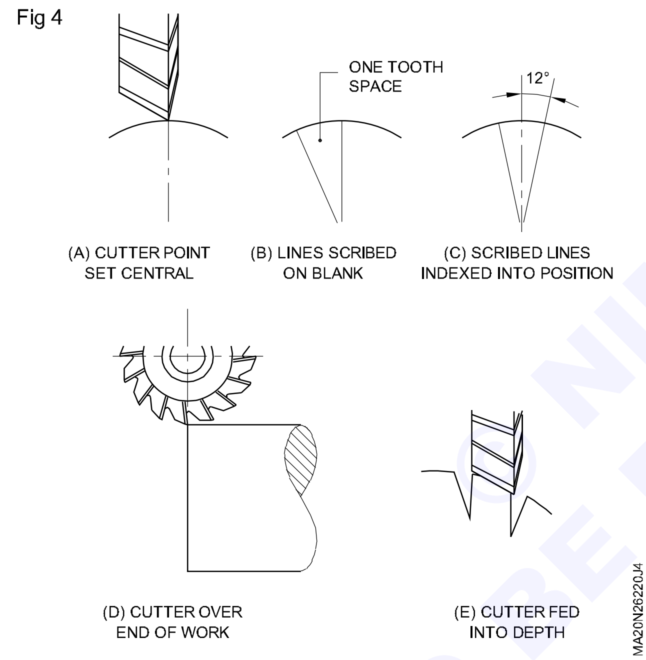
Index the second line to a position 12° from the vertical. (Fig 4C)

Swivel the table to the required helix angle and clamp tightly.

Adjust the knee so that the periphery of the cutter just touches the blank and adjust the table longitudinally to bring the axis of the cutter level with the end of the blank. (Fig 4D)

Start the machine, raise the knee so that the cutter commences to sink into the blank. Operate the cross- slide to bring the 12° edge of the cutter coincident with the line set at 12°, and at the same time raise the knee until the correct depth of flute is being cut leaving 1 mm land. (Fig 4E)

Fig 4



Do this without touching the index crank and longitudinal table feed.

Set the graduated dials of the cross feed and vertical feed at 'zero'.

Depending upon the condition of the machine and the depth of the flute the operation can be completed either by two cuts (ie. roughing and finishing) or one cut.

Be sure to lower the table at the end of each cut before returning it for the next cut. Raise the table watching '0' reading on the knee, before starting the new cut.

Repeat indexing and cutting and complete all the flutes.

Total precautions

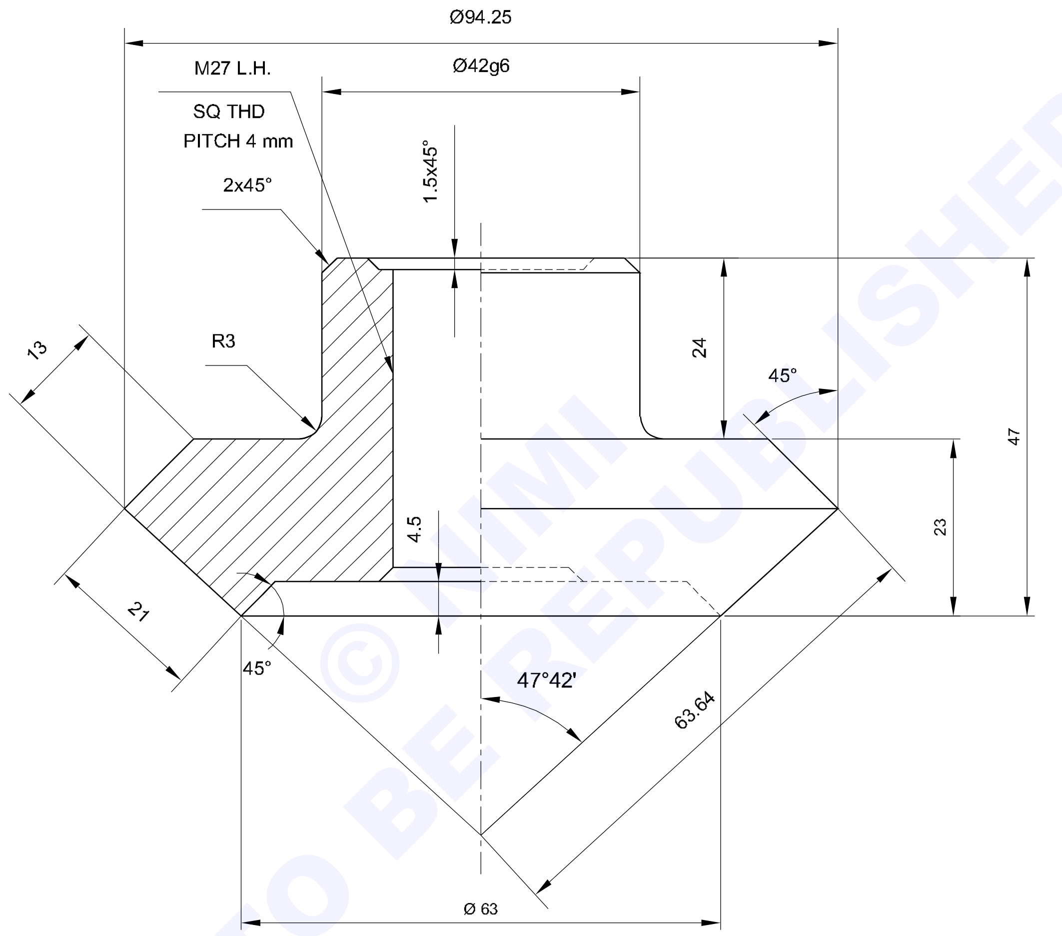
Make sure that the

- work is regidly held on mandrel
- cutter arbor is tight in the spindle
- dog is securely fastened
- dog has plenty of clearance with the cutter when rotating.

Remove the mandrel from the index centres and deburr the edges.

Milling bevel gear

Objectives: At the end of this exercise you shall be able to
- **calculate the essential data for milling teeth on bevel gear using different formulae**
- **set, correct bevel gear cutter for milling bevel gear**
- **set dividing head according to the requirement**
- **cut the bevel gear teeth step by step.**



MODULE	- 3	OUTSIDE DIAMETER	- 94.25
PRESSURE ANGLE	- 20°	CONE DISTANCE	- 63.64
NUMBER OF TEETH	- 30	FACE WIDTH	- 21.00
ANGLE OF SHAFT	- 90°	FACE ANGLE	- 47°42'
PITCH CONE ANGLE	- 45°	ROOT ANGLE	- 41°38'
PITCH DIAMETER	- 90	CUTTER NUMBER	- 3

42g6 - $42^{-0.010}_{-0.025}$

PREMACHINED BLANK AS PER DRAWING

ALL DIMENSIONS ARE IN mm

NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.
1	Ø 100 X 55		-	-	-	2.6.221

SCALE 1:1	**MILLING BEVEL GEAR ON MILLING MACHINE (STRAIGHT TOOTH)**	DEVIATIONS ±0.05	TIME
		CODE NO. MA20N26221E1	

Job sequence

Mill teeth on bevel gear (St.tooth)

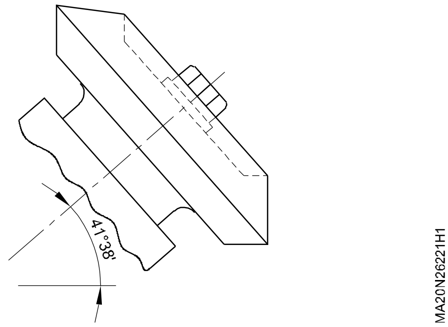
Module 3

Number of teeth = 30

Pitch cone angle = 45°

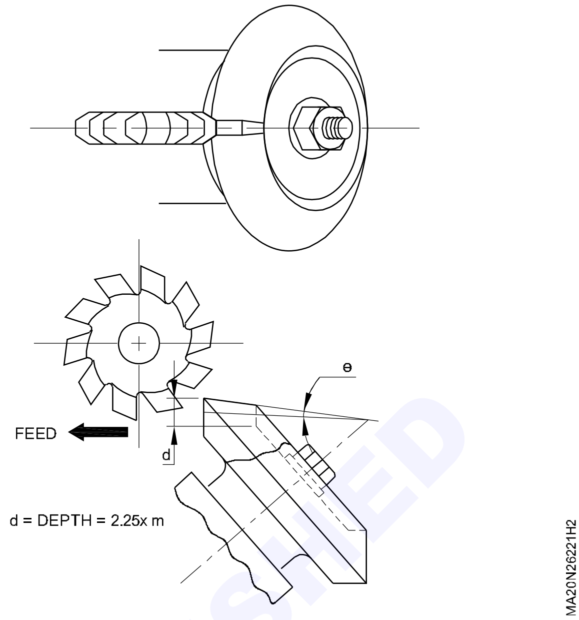
- Mount the universal dividing head on a universal/plain milling machine.
- Tilt the spindle of the dividing head to 41°38. (Fig 1)
- Set the crank pin in the 33 hole circle. Set 11 holes in between sector arms.

Fig 1



- Check the size of the bevel-gear blank. Outside dia = 94.25 mm. Face angle = 47° 42'.
- Fix the blank in a M.T. Shank mandrel and the mandrel into the spindle of the index head.
- Select and set the bevel gear cutter 3, module No.3 and mount it on the arbor.
- Centralise the job with respect to the cutter. Lock the cross feed and set graduated collar at '0' when the cutter touches at the larger end of bevel gear.
- Apply a depth of cut 4 mm and rough out all the teeth.
- Set the depth of cut at the large end to 6.75 mm. Lock the vertical feed.
- Mill the 1st tooth space. (Fig 2)
- Index 1 full turn and 11 spaces in the 33 hole circle.
- Mill the second tooth space.
- Check the form and measure thickness at both ends of the tooth by gear tooth Vernier caliper.
- Determine the excess material at larger end to be trimmed by off-set method.
- Repeat the steps and mill all the teeth.
- Unlock the cross-feed.

Fig 2



- Offset the job by 0.04 mm for 1st offset (trimming). Compare one half of the trimming stock already measured.
- Rotate the job by 3° (11 spaces in the 33 hole circle, for the 1st angular movement.)
- Correct and mill all the 30 teeth.

Ensure that no material is being cut at smaller end of tooth by the cutter after first offset and first angular movement.

- Check the thickness of the tooth.
- Offset the job by 0.07 mm. (In the opposite direction for the 2nd offset.)

Eliminate back-lash before attempting to offset

- Rotate the job by 6°. (22 spaces in the 33 hole circle for the 2nd angular movement)
- Correct and mill all the 30 teeth in the opposite flank.
- Check for correct thickness at the larger end by a gear tooth vernier caliper.
- Form the smaller end by filing.

Skill sequence

Cutting bevel gear (straight tooth) in a milling machine

Objectives: This shall help you to
- **tilt the dividing head to the root angle of the bevel gear**
- **offset the job for 1st and 2nd trimmings.**

It is impossible to cut an accurate bevel gear in a milling machine, since the pitch and involute curve are different at the larger end and at the smaller end. These two different pitches and curves can not be formed by a single gear cutter. It often happens, however, that a bevel gear may be wanted in a hurry, or that an extremely accurate gear is not required. Then it is convenient to know how to mill a bevel gear in a milling machine. Proceed as follows:

Before starting for cutting a bevel gear, the following essential data should be kept ready by calculation.

i Number of teeth

ii Pitch (DP/module)

iii The root angle

iv Chordal thickness at larger end

v Chordal thickness at smaller end

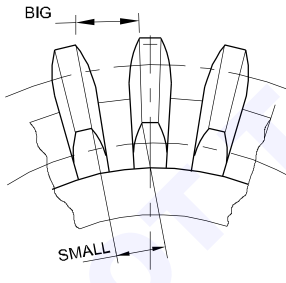
vi Addendum at larger end and smaller end

vii Whole depth at larger end

Selecting the gear cutter

Bevel gear cutters are made in sets similar to spur gear cutters with the same range of number of teeth to each cutter. Bevel gear cutters have a curve of cutting edge that is right for the larger end of the tooth, but they are thinner than spur gear cutters because they must pass through the space at the smaller end of the tooth. (Fig 1)

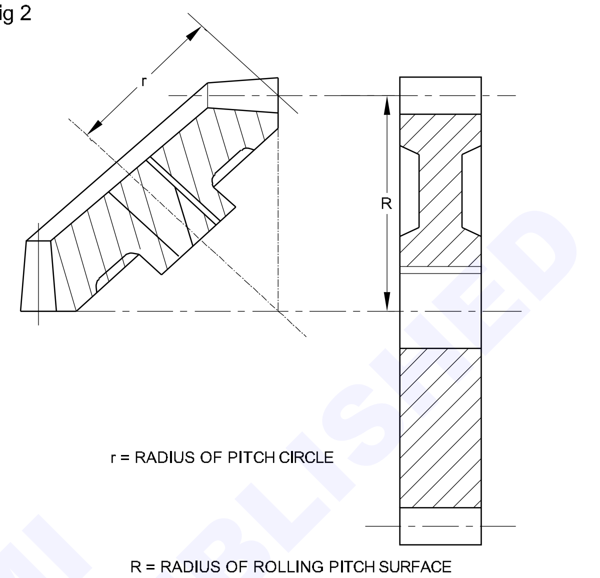
Fig 1



The bevel gear cutter is not selected for the number of teeth in the bevel gear itself but for a spur gear having a pitch radius equal to the back-cone radius of the bevel gear. (Fig 2)

The involute curve depends upon the radius of the rolling pitch surface and the radius of the rolling pitch surface of the bevel gear is longer than the radius of the pitch circle of that bevel gear.

Fig 2



Rule

Number of teeth for which to select the cutter for bevel gear

$$= \frac{\text{Number of tooth on bevel gear}}{\text{Cosine of pitch angle}}$$

This number will be always more than the actual number of teeth on the bevel gear to be cut.

E.g. Number of teeth on bevel gear 20, pitch angle is 45°. Then the number of teeth on "virtual spur gear", will be

$$\frac{N}{\text{Cos Pitch cone angle } \delta'} = \frac{20}{0.707} = 28$$

Select cutter number 4 of required module/DP based on 28 teeth.

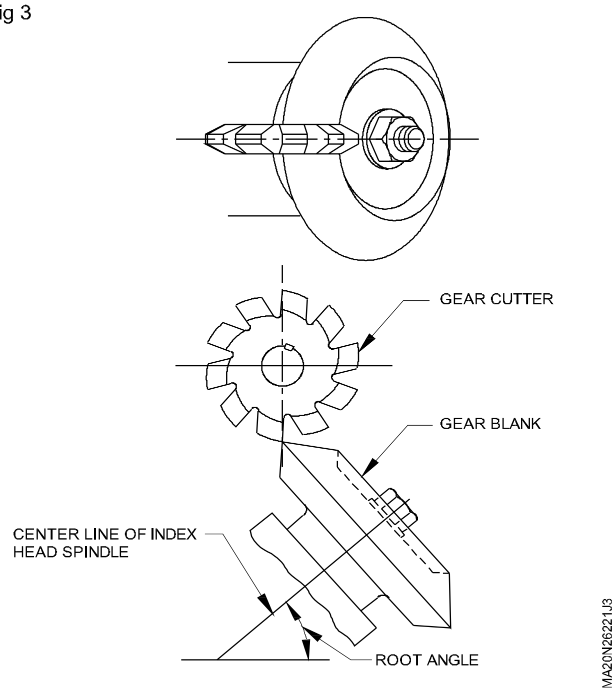
Check the size of the blank, especially the outside diameter and the face angle.

Set the index head for indexing given number of teeth.

Always select the longer circle possible as it permits finer adjustment.

Mount the bevel gear blank with the spindle of the dividing head. For smaller blank, a straight mandrel with threaded nut is sufficient. In case of larger one, mount it in the M.T. taper shank mandrel, which is held in the bore of the spindle of dividing head by means of draw-in bolt. (Fig 3)

Fig 3



The end nut of the mandrel should be as thin as permissible; otherwise it may be milled while cutting the teeth.

Set the dividing head to the root angle (also called cutting angle.) (Fig 3)

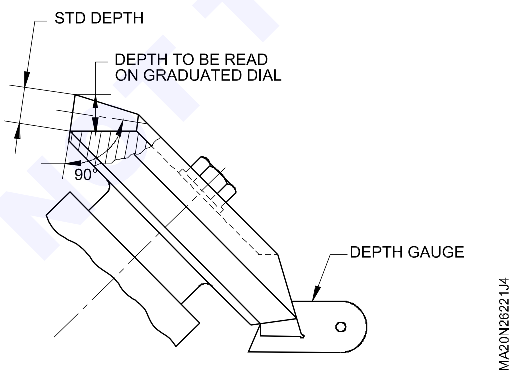
Root angle df= Pitch cone angle minus dedendum angle

$$\text{Dedendum angle } \theta_f = \text{Tan } \theta_f = \frac{\text{Dedendum}}{\text{Cone radius}}$$

Whole depth of cut

The depth should be perpendicular to the pitch line. While cutting bevel gear do not depend entirely on the reading of the graduated dial of the vertical movement. For this reason, before starting cutting, the depth of the teeth is marked by a depth gauge on the back crown of the bevel gear. (Fig 4)

Fig 4



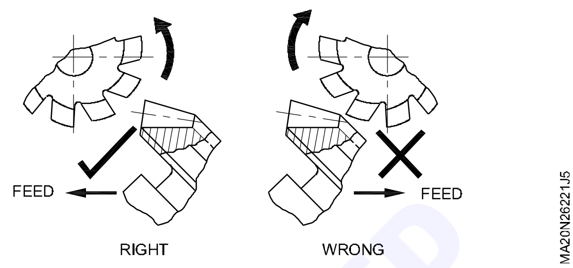
By calculation we get only the standard depth from face to back crown, not perpendicular to the pitch line.

Depth perpendicular to pitch line is equal to standard depth x Sin (90° - Dedendum angle)

Direction and starting of cut

There will be a tendency to lift the gear blank if the cut is started from the smaller end and the cutter is rotated in the direction of "up milling". This method is not advisable. For better result, cut teeth from the larger end and the cutter should be rotated in the up-milling direction. Then there will be less chances for breaking the cutter and no tendency to lift up the gear blank. (Fig 5)

Fig 5



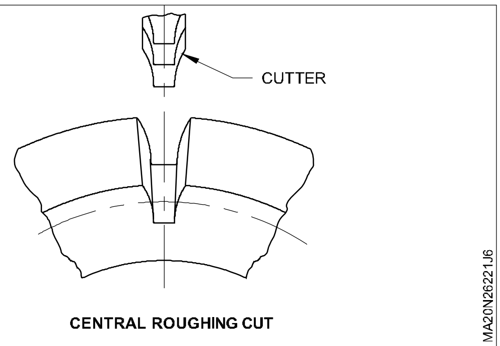
Ensure that the spindle arbor, collars and cutter are clean and that arbor runs true, and set the cutter (pre-selected) on the arbor so that the direction of the cut will be away from the dividing head spindle. Have the cutter as near the machine spindle as practicable.

Check the 'run-true' of the bevel gear blank and centrilise the gear blank under the cutter.

Adjust the table until the revolving cutter just touches the gear blank at the outside diameter.

Raise the table to the whole depth and take a roughing cut at the centre of the tooth space. (Fig 6)

Fig 6



In a cast iron gear of 5DP/5 m or larger or in a steel gear of 8DP/3 m or larger, it is usually advisable to take a central roughing cut before 'trimming'either side.

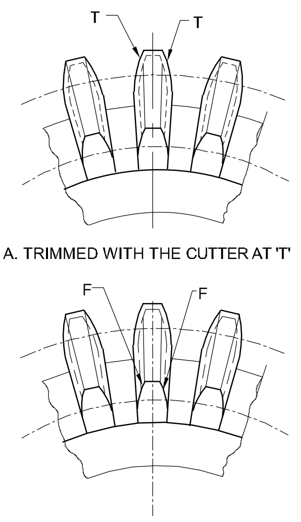
Index for the next tooth, take a cut, and so on, all around the gear blank.

Measure the thickness of the tooth with a gear tooth vernier caliper at the larger end and at the smaller end. Subtract the final thickness from the thickness measured, and divide by 2 to know how much is to be trimmed off from each side, i.e. total stock to be trimmed off.

Trimming/offset of blank

The job of cutting a bevel gear in a milling machine is to get the correct thickness of the tooth at the pitch line, at both ends of the tooth by trimming both sides of the tooth and then to file the small end to the curve. Since the curve of the cutter is correct for the larger end of the teeth, the shape of the tooth at the smaller end is not right. (Fig 7)

Fig 7



A. TRIMMED WITH THE CUTTER AT 'T'

B. FILED TO CORRECT CURVEAT 'F'

CUTTER A BEVEL GEAR

MA20N26221J7

Similarly the thickness of the cutter is correct for the finished space at the smaller end and hence, the thickness of the tooth at the larger end is altogether too great. Hence, trimming the tooth space on both the flanks of the tooth is necessary to have the teeth free to roll in the assembly.

1st off-set (Trimming) (Fig 8A)

$$\text{1st off-set } \frac{C-F}{C} \times \text{Sin} \frac{90°}{N}$$

where C = Cone distance

F = Face width

N = Number of teeth on bevel gear.

1st angular movement of the gear blank = 90°/N.

Offset the table to the amount pre-calculated. Turn the index crank until the cutter just touches the side of the tooth on the smaller end. (Fig 7b)

> **Take out the back lash of cross-feed screw, before attempting the table offset. The angular movement should be opposite to the table movement. Ensure it carefully.**
>
> **Ensure that no material is being cut at the smaller end of the tooth by the cutter after 1st off-set and 1st angular movement, except at the larger end of the tooth. Check the thickness of the tooth at the larger end. It should be the final thickness plus 1/2 of the previously measured difference or half of the trimming stock.**

Trim one side of the tooth in all the spaces of the tooth. (Fig 8A)

2nd Off-set/Trimming (Fig 8B)

Twice the amount of the 1st off-set of the table movement and the angular movement of the gear blank, oppsite to the 1st off-set and angular movement.

$$\text{Rule 2nd off-set} = \frac{C-F}{C} \times \text{Sin} \frac{180°}{N}$$

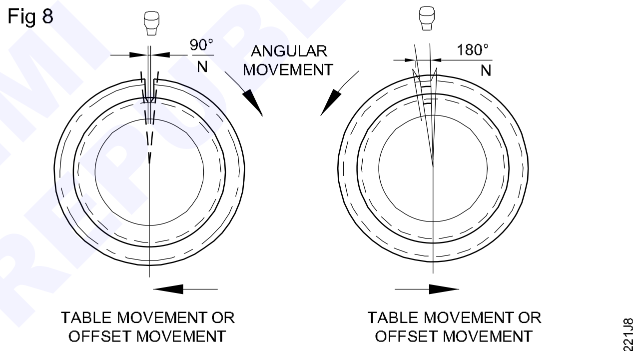
$$\text{2nd angular movement} = \frac{180°}{N}$$

> **Beware of backlash when off-setting the table. First remove backlash and then proceed to off-set.**

Set the graduated dial on the cross-feed screw at '0' after taking out the backlash.Move the table in the opposite direction to the 1st off-set to twice the amount reading in the dial.

Rotate the gear blank in the opposite direction to the table movemen(Fig 8B.) by rotating the index crank till the side of the tooth at the smaller end just touches the cutter.

Fig 8



A. MACHINING OF THE FIRST FLANK   B. MACHINING OF THE SECOND FLANK

OFFSETTING AND ROTATING MOVEMENT OF THE FLANK

MA20N26221J8

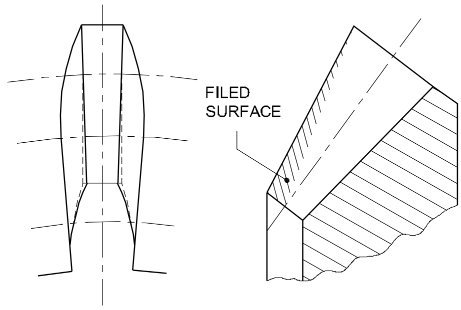
Trim the second side of the tooth and measure the thickness at the larger end. If necessary make slight adjustment in the off-set and angular movement to achieve final thickness.

Index for the next tooth and trim the other side of the teeth, and so on all the teeth.

Remove the cutter from the arbor. Slide the dividing head at the right hand end of the table and clamp it.

Trim with the smooth file the smaller end of all the teeth to the correct form/curve as shown in Fig 9 and deburr.

Fig 9



PART OF BEVEL GEAR TOOTH THAT IS CORRECTED BY FILLING

MA20N26221J9

Milling plate cam

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

- **mark and punch the profile of the cam as per drawing**
- **mount the vertical milling attachment on the universal milling machine**
- **hold the cam blank in the dividing head**
- **connect the dividing head to the table feed screw using the required change gears**
- **mill the cam as per drawing.**

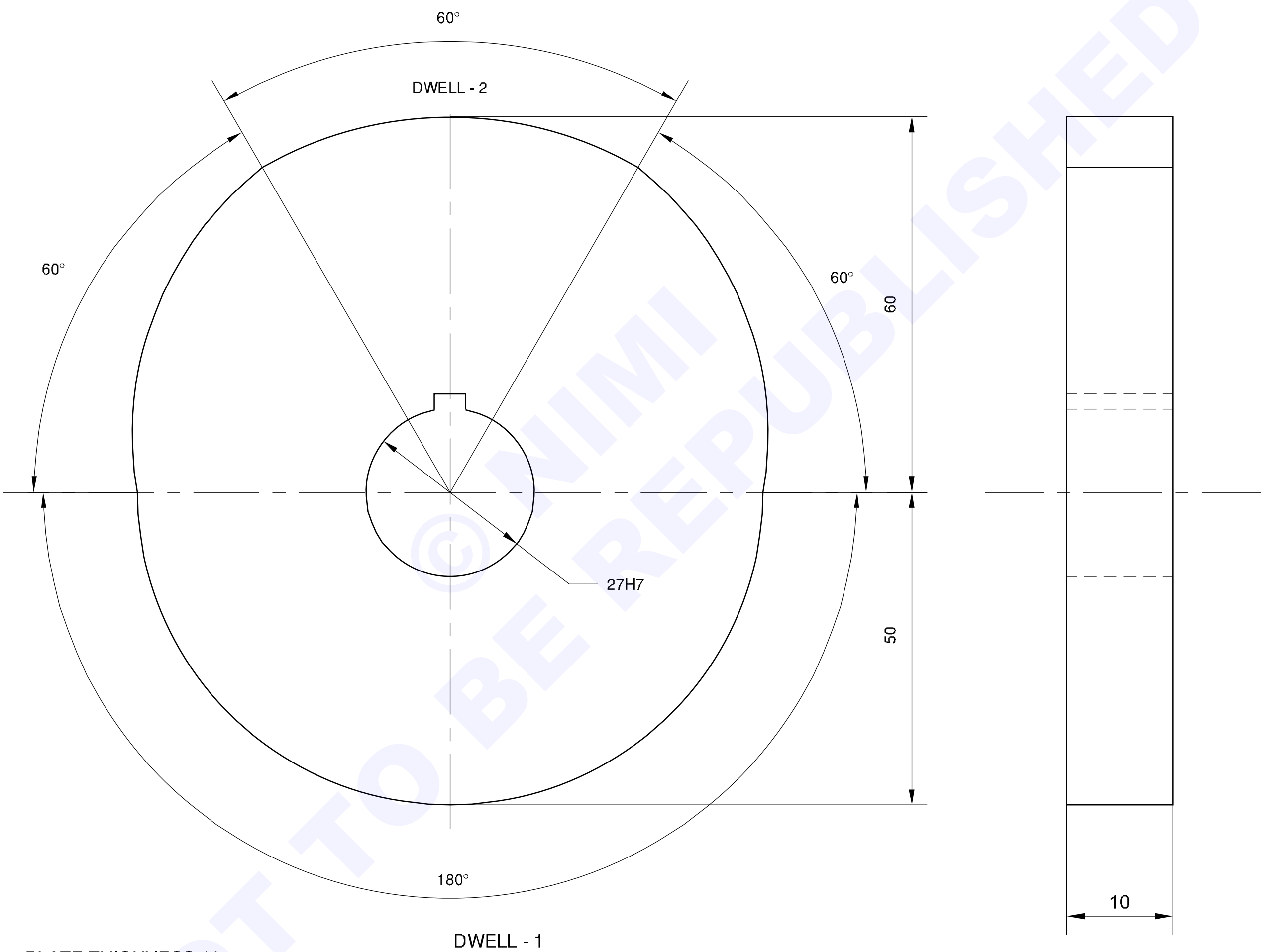


PLATE THICKNESS 10mm
RAISE = 10mm
BASE CIRCLE = Ø100mm

27H7 - $27^{+0.021}_{+0.000}$

PREMACHINED CAM BLANK AS PER DRAWING

ALL DIMENSIONS ARE IN mm

1	Ø 125 X 20		--	--	--	2.6.222
NO.OFF	STOCK SIZE	SEMI PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.

SCALE 1:1	**MILLING PLATE CAM**	DEVIATIONS ±0.05	TIME
		CODE NO. MA20N26222E1	

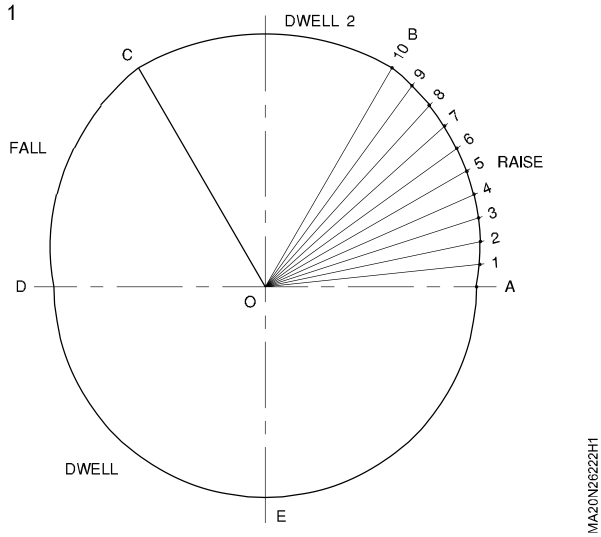
Job Sequence

Milling plate cam

- Mount the vertical milling attachment on the universal milling machine.
- Mount the universal dividing head in line with the axis of the table.

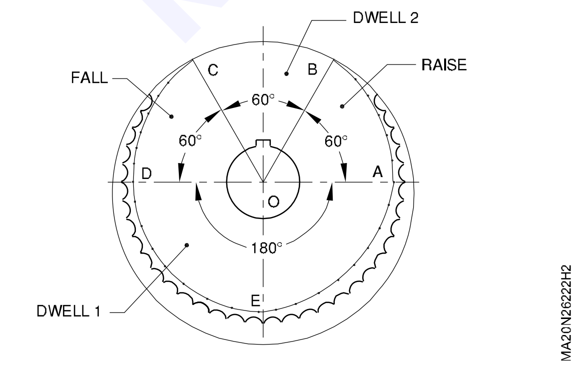
For marking AB and CD (Fig 1)

Fig 1



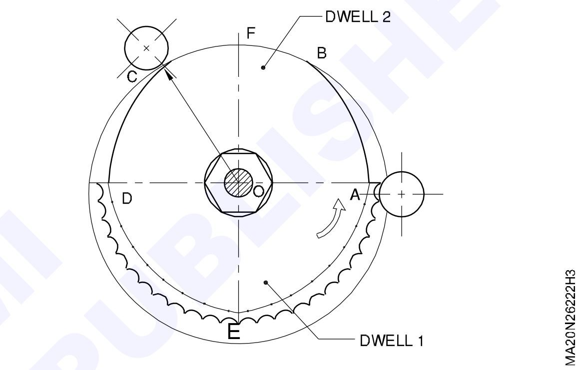
- OA = 50 mm and OB = 60 mm uniform rise from A to B is 10 mm.
- Divide angle AOB(60°) into 10 equal divisions of 6° each.
- Draw the angles of 6° increment from OA.
- On each line/division from OA increase the radius by 1mm and mark 01 (51 mm); 02(52 mm); 03(53 mm) upto 010.
- Join all the points 1,2,3 ..... by a smooth curve by free hand.
- Same procedure has to be followed for arc CD also.
- As regards BC and AED draw regular arcs using a divider.
- Mark and punch the profile of the cam on the blank. (Ref. Job drawing)
- Drill and reduce the excess stock on its circumference. (Fig 2)

Fig 2



- Hold the blank in the dividing head, using an arbor directly, with a draw-in bolt.
- Connect the dividing head to the table feed screw with the selected change gears.
- Tilt the index head at an angle calculated.
- Set the spindle of the vertical attachment in parallel with the axis of dividing head spindle if the index head is set at an angle to obtain the required rise.
- Mount the collet adopter and hold a ø20 mm end mill in the vertical head.
- Manipulate and position the end mill at A (Fig 3) and mill 180° Dwell 1 (ref. job drg) portion up to D by feeding the blank manually by rotating the dividing head crank.

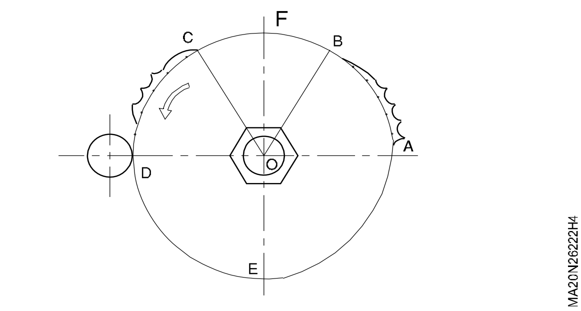
Fig 3



When milling the dwell portion, lock the index plate with a back stopper pin and pull out the index crank plunger, and rotate the feeding movement to the cam blank.

- Disconnect the back stopper pin of the index plate, feed and mill the portion D to C manually by rotating the crank. (Fig 4)

Fig 4



- Stop the machine and bring back the cutter to position A.
- Start the machine and mill the portion A to B of the cam by rotating the crank of the indexing head.
- Connect the back stopper pin and mill the dwell (2) portion along BFC, by rotating the crank.

Skill sequence

Milling plate cams on milling machine

Objective: This shall help you to

- **mill a plate cam (uniform rise) on a universal vertical milling machine.**

Cams are being used in modern manufacture to an increasing extent. Cams provide the easiest method of obtaining the intricate and unusual mechanical movements of automatic machines. Cams can be divided into three classes.

i Radial plate

ii Cylindrical or barrel

iii Pivoted beam

Of the above three classes, the first two classes can be manufactured in small batches on a universal milling machine with vertical attachment and universal dividing head. (Refer to Related Theory for more details, and the terms related with the cams.)

To mill the profile (LOBE) of a uniform rise and fall plate cam, proceed as follows.

Lay out the profile of the cam on the cam blank as per the details given in the job drawing. Dot punch the scribed lines and curves.

Use Prussian blue coating to preserve the profile till it is finished by milling.

Rough out the excess material ie. by chain hole drilling, or hacksawing and rough grinding over the pedestal grinder.

Provide atleast 1 mm material as machining allowance for finishing on the milling machine.

Mount the plate cam blank on a mandrel with a M.T. taper shank to suit the spindle of the dividing head. Mount the mandrel to the spindle and lock it with draw-in bolt.

Mount the vertical milling attachment.

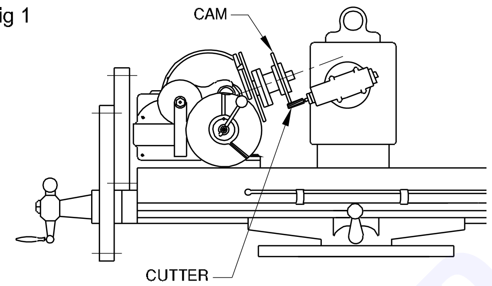
Determine from the job drawing and calculate the rise of the cam (LOBE) for 360° and arrange the gear train between the worm-stud of the dividing head and the feed screw of the worktable. (Fig 1)

$$\text{Gear ratio} = \frac{\text{Gear on worm stud}}{\text{Gear on lead screw}}$$

$$= \frac{\text{Driven}}{\text{Driver}} = \frac{\text{Lead of machine}}{\text{Max. lift per revolution}}$$

Refer to Related Theory on cams, for the formulae, and example for calculating the rise of the lobe for 360°.

Fig 1



MILLING A PLATE CAM BY USING INDEX-HEAD AND VERTICAL MILLING ATTACHMENT

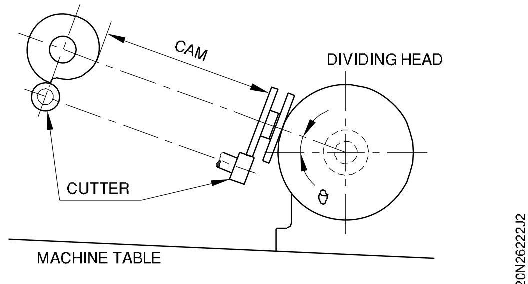
If the rise for 360° is less than what is the minimum lead possible with that index head, (refer to the table of change gears for helical milling supplied by the manufacturers) then arrange change gears for the next higher lead, preferably round figure which is possible by the standard set of change gears.

Watch the direction of rotation of the cam blank with reference to the table movement. If necessary, connect the idler in the gear train to change the direction of rotation.

Tilt the index head to the angle calculated.

The axis of the dividing head spindle and the spindle of the vertical milling attachment must always lie parallel. (Fig 2)

Fig 2



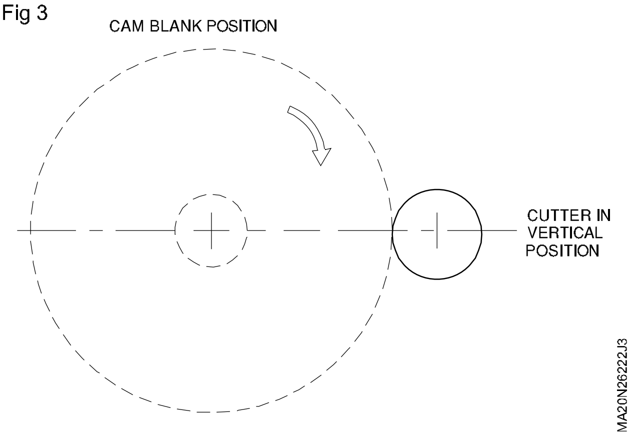
CUTTER AND DIVIDING HEAD SPINDLE MUST REMAIN PARALLEL

Select and mount the end mill cutter whose diameter should be equal to that of the roller of the cam follower.

Do not use a taper shank end mill with tanged end, as it may be pulled down during cutting and cause breakage of the cutter. Use parallel shank with collet or taper shank with tapped hole and draw-inbolt.

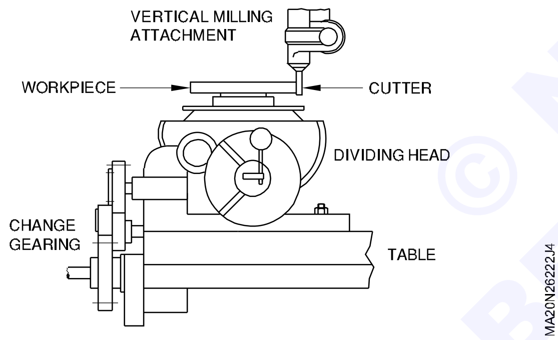
Align the centres of the end mill cutter and the cam blank, so that they lie on the same horizontal line. (Fig 3)

Fig 3



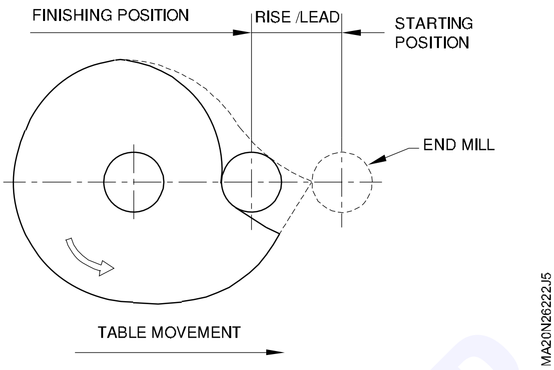
The cutting is done by the teeth on the periphery of the end mill. Whenever possible the job should be set up so that the end mill will cut off the lower side of the blank, as this brings the end mill and the table nearer together and makes the job more rigid. It also prevents chips from accumulating, and enables the operator to see better the scribed lines and punch marks. (Fig 4)

Fig 4



Commence rough milling of the cam lobe starting from the 'rise' to 'fall' of the cam, by rotating the index crank in the clockwise direction with the crank pin inserted in the hole. (Fig 5)

Fig 5



If the diameter of the end mill is less than 12 mm then do not overload the cutter. Apply only light cuts.

Wherever there is no rise marked in the lobe or concentric surface to the centre of the Cam is required, at that point pull out the index pin from the hole of index plate and rotate the index crank until the concentric surface is covered by the end mill.

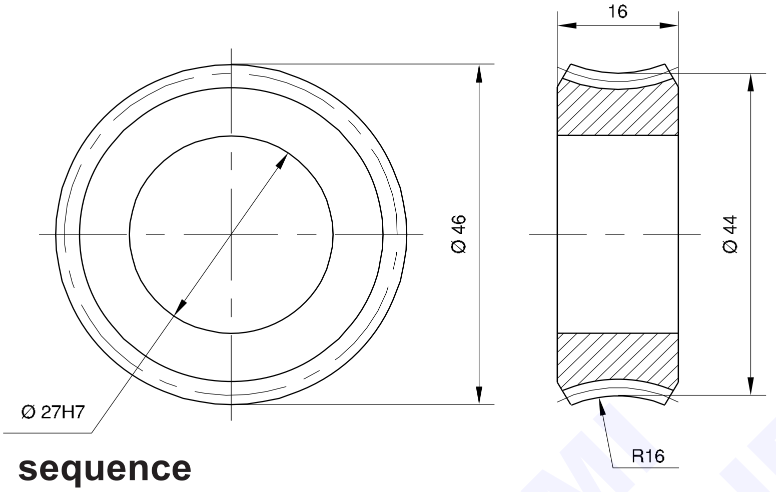
Apply depth of cut for finish milling up to the scribed line and observe half punch mark.

Pull out the index plunger pin from the index plate hole, while applying the depth of cut.

Finish mill the cam lobe and deburr the edge of the bottom face of the cam plate.

Milling worm wheel

Objectives: At the end of this exercise you shall be able to
- **mount a worm wheel blank on the mandrel between centres**
- **swivel the table to the gashing angle according to the hand of worm thread**
- **select and mount a gashing cutter/fly cutter suitable to the wormwheel**
- **mill worm teeth to the exact form.**



MODULE :- 2 AND 20° PRESSURE ANGLE

NO. OF TEETH : 20

LEAD ANGLE : 3°16'

NO .OF START : ONE

HAND THREAD ON WORM : R.H

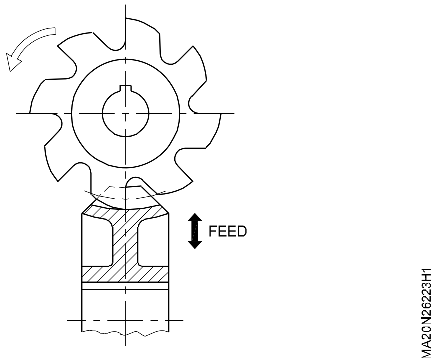
27H7 - 27 $^{+0.021}_{+0.000}$

Job sequence

Milling a worm wheel

- Check the size of the blank, outside dia. 46.0 mm, throat dia. 44 mm and throat radius 16 mm.
- Mount the index head on a universal milling machine.
- Set the indexing head for indexing 20 teeth. (2 full turns in any hole circle)
- Mount the job between centres using a mandrel.
- Check the run out.
- Mount cutter No.6 of 2 module on the arbor.
- Centralise the job longitudinally and crosswise with respect to the cutter. (Ref. Skill Sequence of Fig 3&4.)
- Lock the longitudinal and cross-feed, setting the graduated collars at 'O'(zero).
- Swivel the table to 3°16' for right hand helix.
- Mill one tooth space to a depth of 4.5 mm by plunging the job gradually. (Gashing) (Fig 1) by using vertical feed. Set dial of '0'.
- Lower the job.
- Index 2 full turns for the next tooth space.

Fig 1



- Mill the second tooth space watching the '0' on the dial of vertical feed.
- Check the thickness of the tooth at the throat using a gear tooth vernier caliper.
- If necessary adjust the depth of cut.

The thickness of tooth to be measured normal to the helix of tooth.

- Repeat the above steps and mill all the 20 teeth.
- Deburr and finish the job.

PREMACHINED GEAR BLANK AS PER DRAWING

1	Ø 50 X 20		Fe310	--	--	2.6.223
NO.OFF	STOCK SIZE	SEMI PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.
SCALE 1:1	WORM WHEEL				DEVIATIONS ±0.06 mm	TIME
					CODE NO. MA20N26223E1	

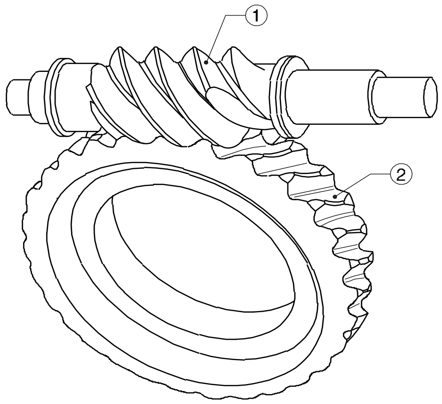
Skill sequence

Cutting a worm wheel on universal milling machine

Objective: This shall help you to
- **cut teeth on a worm wheel.**

Worm gearing is a combination drive obtained by using two members together (1) worm is the driving member (2) worm-wheel is driven. (Fig 1) Worm gearing is employed where large speed reduction ratio is required as in the case of gearbox and where heavy material handling equipments such as chain blocks, cranes and other weight lifting appliances. For repairing purposes a new worm wheel of practical accuracy can be cut on a universal milling machine together with the help of dividing head and suitable rotary cutter, hob or fly cutter. To cut a worm wheel proceed as follows.

Fig 1

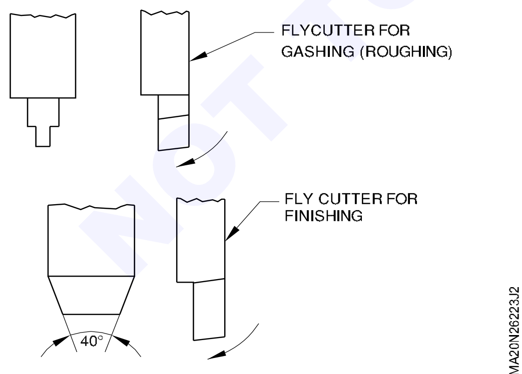


Mount the worm wheel blank on the mandrel and fix the mandrel between centres. Fix the carrier and check the 'run-true' of the blank with a DTI.

Set the indexing plate and sector arms for the required number of teeth.

Select the gashing cutter or fly cutter suitable to the throat radius of the worm wheel and fix it on the arbor. (Fig 2)

Fig 2



Align the longitudinal traverse, at the centre to the tool and lock the table in position. (Fig 3) Move cross-traverse and bring the wheel centre to the gashing cutter and lock it in position. (Fig 4)

Swivel the table to the gashing angle and hand of thread on the worm. Refer to the job drawing. If not given in the drawing then calculate using the formula given below.

Fig 3

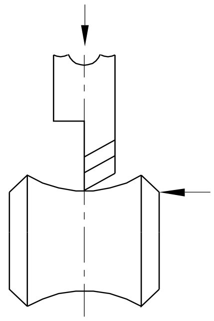
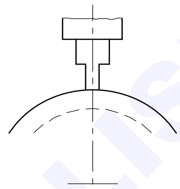


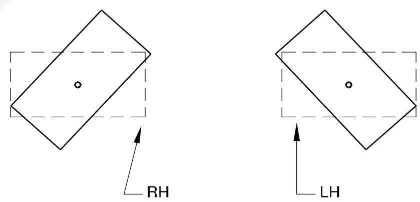
Fig 4



$$\text{Gashing angle} = \text{Tan}\ \vartheta = \frac{\text{Lead of worm}}{3.1416 \times \text{pitch dia of worm}}$$

Swivel the table at the right hand end for the worm having R.H. thread and at left hand end for the worm having L.H. thread. (Fig 5)

Fig 5



Start the machine

Raise the vertical movement and feed gradually up to 0.1 mm less than the full depth. Set dial at '0'.

Lower the table and index for the next tooth.

Cut all the teeth by repeating the above two steps.

Remove the gashing cutter and mount the hob (similar in shape and dimensions, with a bore to suit the arbor of the milling machine.)

> **Swivel the longitudinal table to 'O' and remove the dog carrier.**

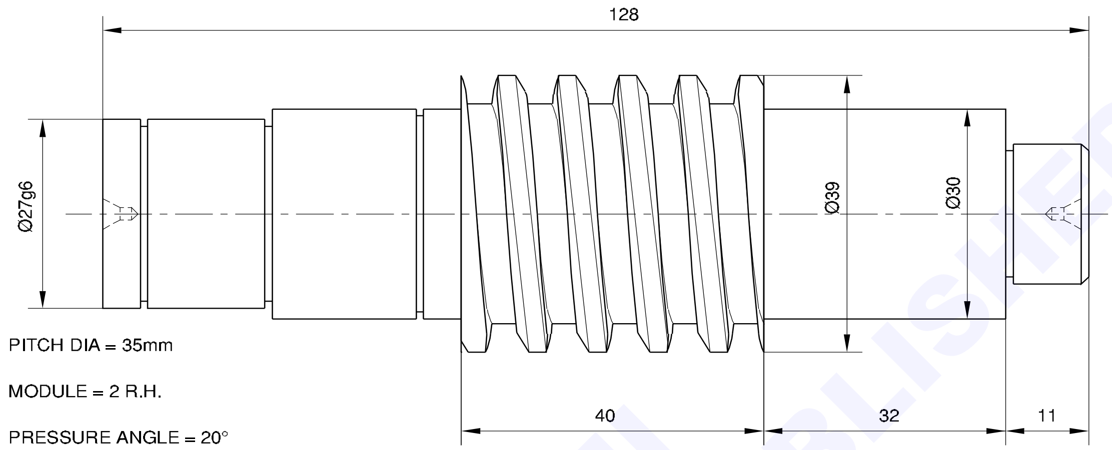
Start the spindle of the machine, raise the vertical feed up to full depth and lock it in position.

Now the hob will drive the wormwheel and mill the teeth to the exact form.

> **Apply enough lubricant at the point of meshing to minimise heat and friction.**
>
> **If suitable hob is not available, then use finishing tool up full depth immediately after'gashing' over, and the table at swivelled position.**

Milling worm thread

Objectives: At the end of this exercise you shall be able to
- **mount worm blank between centres using a driving dog on the dividing head**
- **mount universal milling attachment and swivel it to helix angle required**
- **arrange and set the dividing head using the gear tain for a required lead**
- **cut worm to the desired depth and lead.**



PITCH DIA = 35mm

MODULE = 2 R.H.

PRESSURE ANGLE = 20°

27g6 - $27^{-0.010}_{-0.020}$

Job sequence

Milling a worm

- Check the outside dia. of the worm blank dia. 39.0 mm.
- Mount universal milling attachment into the spindle of a universal milling machine.
- Set the spindle of the attachment at '0'(Zero) and its axis parallel to the longitudinal travel of the table.
- Select and mount a gear cutter 2 module, cutter No.1 on the spindle of the attachment.
- Set the universal index head and the foot stock on the table of the machine.
- Disengage the worm shaft from the worm wheel of the index head by shifting the eccentric bush.

This is essential to mill worm thread with short lead.

- Arrange the gear train to link the spindle of the index head and table feed screw for the lead of worm = 6.28 mm (6.25 mm) nearest possible with standard set of change gears.

$$\frac{\text{Lead of machine}}{\text{Lead of worm}} = \frac{\text{Drivers}}{\text{Drivens}}$$

$$\frac{5}{6.25} = \frac{40}{100} \times \frac{64}{32}$$

Machine lead will be 5 mm if we connect the feed screw and the spindle of the index head through the gear train of 1:1 ratio.

- Mount the job in between centres and align it.
- Centralise the job with respect to the cutter.
- Swivel the table or attachment of the machine whichever is convenient to 3°15' for right hand helix.
- Give a depth of cut 4.5 mm by vertical feed.
- Feed the job longitudinally slowly, gently and mill the worm thread.

During feeding, ensure that the job is not rotating within the dog.

- Deburr the teeth with a smooth file.
- Check the thickness of the tooth at pitch line (3.14 mm) by a vernier gear tooth caliper in a plane normal to the helix angle.

PREMACHINED GEAR BLANK AS PER DRAWING

NO.OFF	STOCK SIZE	SEMI PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.
1	Ø 40 x 130		Fe310	-	-	2.6.224

SCALE 1:1	MILLING WORM THREAD	DEVIATIONS ±0.06	TIME
1:1		CODE NO. MA20N26224E1	

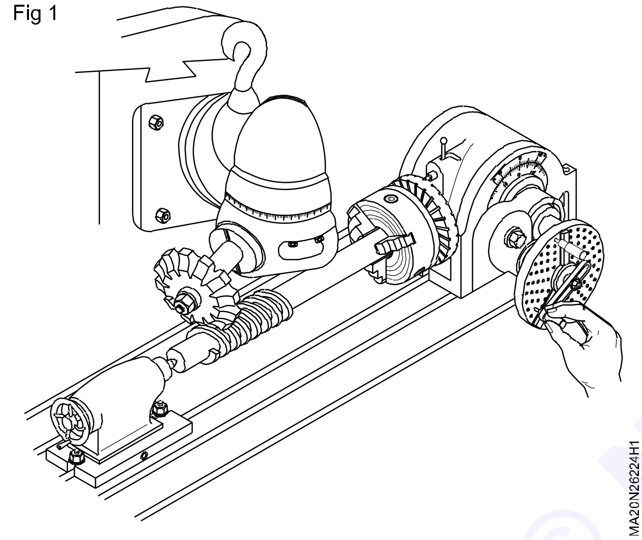
Skill sequence

Cutting worm thread on universal milling machine

Objective: This shall help you to
- **cut worm thread on worm blank/shaft using universal milling attachment.**

Worms with single start threads are usually threaded on tool room centre lathes. But when the number of starts are more than four, and the lead of thread is beyond the capacity of the lathe, then it is the usual practice to have it done on a universal milling machine together with the help of universal milling attachment. (Fig 1) To cut worm threads on worm blank on universal milling machines proceed as follows.

Fig 1



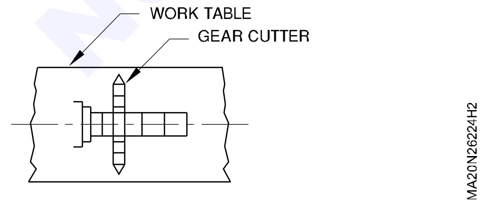
Clean the working surfaces of the spindle bore of the machine column and the table. Mount the universal milling attachment with the spindle and the column of the machine.

Ask your instructor in case of doubt while mounting. Get the help of fellow trainees to lift and mount the attachment. Follow personel safety rules while lifting heavy attachments.

Set the spindle of the attachment at '0' and its axis is parallel to the longitudinal traverse of the work table. (Fig 2)

Select and mount the gear cutter of required DP or module cutter number 1 always (i.e rack cutter).

Fig 2



Mount the worm blank on the mandrel and fix the mandrel between the centres of the universal dividing head and foot stock. Attach the driving dog/carrier.

Ensure that the dog/carrier will not foul any stationary part of the attachment while cutting operation is going on.

Arrange the index plate and sector arms to the required number of starts.

Disengage the back stopper pin from the index plate as the index plate has to be rotated together with the index crank.

Arrange a gear train for the required lead of worm thread between the worm shaft and the feed screw.

(Lead of worm = pitch x number of starts)

$$\frac{\text{Lead of machine}}{\text{Lead of worm}} = \frac{\text{Driving gears}}{\text{Driven gears}}$$

Refer to the table of change gears for helical milling supplied by the index head manufacturers for the correct change gears and their position in the train of gears. AlsoRefer to Related theory on 'worm' calculation.

Ensure that the direction of rotation of the work blank is in accordance with the hand of helix of the thread when the table is traversed against the cutter.

Calculate and determine the helix angle of the worm thread to be cut.

$$\text{Tan helix angle } \theta = \frac{\pi \text{ x PD}}{\text{Lead}}$$

Align the worm-blank against the centre of the gear cutter axis by adjusting the cross-slide. (Fig 3)

Refer to Skill Sequence on 'worm wheel' milling for the direction of swivel of the table in case of doubt.

Swivel the spindle of the attachment or the work table, whichever is convenient to 90° minus helix angle in the direction corresponding to the hand of thread ie. RH or LH. (Fig 3)

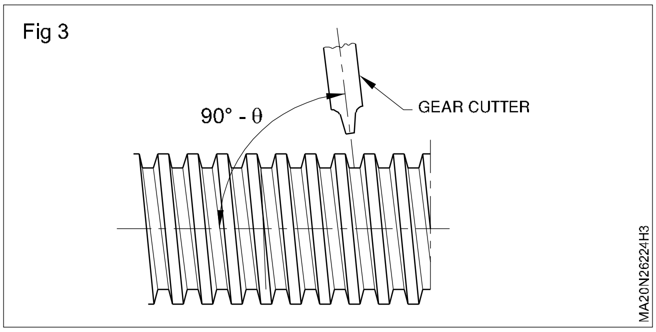


Fig 3

Run the spindle at the suitable r.p.m. for the given material of the blank.

Raise the vertical movement and get the feel of the cutter over the worm-blank. Set the graduated dial at 'O'.

Bring the worm-blank to starting point by traversing the table. Keep atleast 5 mm clearance between the cutter and the face of the worm blank (Fig 4). Apply depth of cut for roughig cut.

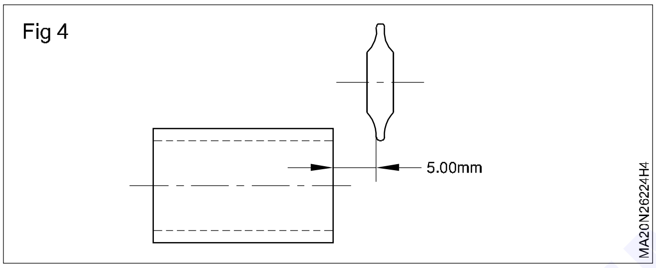


Fig 4

Roughing cut is necessary when the pitch is more than 2 m or 14 DP for roughing 3/4th of the total depth of thread may be applied.

Commence milling the thread groove, by rotating the index crank clockwise along with the index plate as in helical milling. Apply enough cutting fluid at the point of cutting. Continue feeding till the cutter clears the rear face of the blank and stop the spindle. Reverse the feed and take back the blank to the starting point.

Index for the next thread/tooth, start the spindle and mill the next groove following the above sequence of operation.

Ensure that the worm blank and the dog are not loosened on the mandrel while taking roughing cuts, because of cutting pressure.

Complete the roughing operation on all the thread grooves by following the above working steps.

Apply full depth of cut (minimum should be 2.25 x M or 2.157/DP) and finish the 1st and 2nd groove.

Measure the thickness of the tooth by a gear tooth vernier caliper at pitch line and at the plane normal to the helix. (Fig 5) It should be 0.5 times of linear pitch.

Linear pitch = 3.1416 x Module or 3.1416/DP

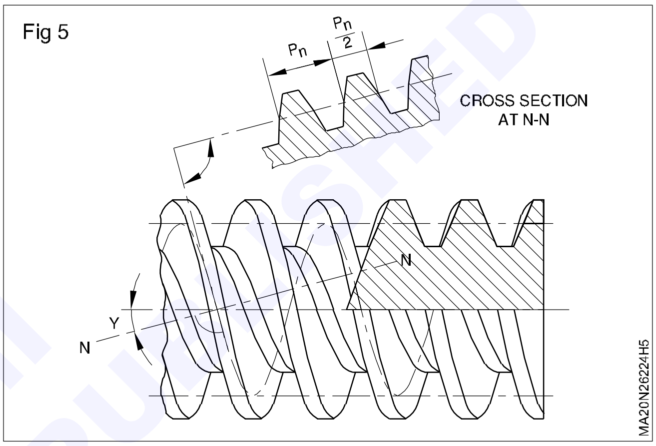


Fig 5

If necessary increase the depth of cut to correct the thickness of the tooth, and then complete all the other grooves, one by one, till all the teeth are of the same thickness. Make it sure by checking with a gear tooth vernier caliper.

Beware of "Backlash effect' during reversal of feed.

Remove the mandrel and deburr the rear face of the worm.