TURNER

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 Years**

Trade : **Turner - 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.235/-

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 **Turner - 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 **Turner 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. G. Mani	-	Junior Works Manager, Dept. Def. prod, MDC, Chennai.
Shri. P.J. Jishnu	-	Junior Instructor, Govt ITI, Trivandrum, Kerala.
Shri. C.V. Pradeep	-	Senior Instructor, Govt ITI, Trivandrum, Kerala.
Shri. H.A. Manukumar	-	Training Officer, NSTI, Chennai.
Shri. M. Sampath	-	Training Officer (Retd.), CTI, Chennai - 32.
Shri. K. Karthi	-	Junior Training Officer, Govt ITI, Dharapuram.
Shri. V. Janarthanan	-	Assistant Professor, Rtd., MDC Member, NIMI, 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 Eight modules

Module 1	**Form Turning**
Module 2	**Lathe Accessories**
Module 3	**Turning with Lathe Attachments**
Module 4	**Boring**
Module 5	**Thread Cutting**
Module 6	**CNC Turning**
Module 7	**Advanced Turning**
Module 8	**Special Operation on Lathe**

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 **Turner 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	**Plan & set the machine parameter to produce precision engineering component to appropriate accuracy by performing different turning operation. [Appropriate accuracy ±0.02mm/ (MT - 3) (proof turning); Different turning operation - Plain turning, taper turning, boring threading, knurling, grooving, chamfering etc.] (Mapped NOS: CSC/N0110)**	**2.1.88 - 2.1.94**
2	**Set & Produce components on irregular shaped job using different lathe accessories. [Different Lathe accessories: - Face plate, angle plate] (Mapped NOS: CSC/N0110)**	**2.2.95 - 2.2.96**
3	**Plan and set the machine using lathe attachment to produce different utility component/ item as per drawing. [Different utility component/ item - Crank shaft (single throw), stub arbour with accessories etc.] (Mapped NOS: CSC/N0110)**	**2.3.97 - 2.3.100**
4	**Set the machining parameters and produce & assemble components by performing different boring operations with an appropriate accuracy. [Different boring operation - eccentric boring, stepped boring; appropriate accuracy - ±0.05mm] (Mapped NOS: CSC/N0110)**	**2.4.101 - 2.4.105**
5	**Calculate to set machine setting to produce different complex threaded component and check for functionality. [Different complex threaded component- Half nut, multi start threads (BSW, Metric & Square)] (Mapped NOS: CSC/N0110)**	**2.5.106 - 2.5.110**
6	**Set (both job and tool) CNC turn centre and produce components as per drawing by preparing part programme. (Mapped NOS:CSC/NO115)**	**2.6.111 - 2.6.148**
7	**Manufacture and assemble components to produce utility items by performing different operations & observing principle of interchangeability and check functionality. [Utility item: - screw jack/ vice spindle/ Box nut, marking block, drill chuck, collet chuck etc.; different operations: threading (Square, BSW, ACME, Metric), Thread on taper, different boring (Plain, stepped)] (Mapped NOS:CSC/NO115)**	**2.7.149 - 2.7.152**
8	**Make a process plan to produce components by performing special operations on lathe and check for accuracy. [Accuracy - ±0.02mm or proof machining & ±0.05mm bore; Special operation - Worm shaft cutting (shaft) boring, threading etc.] (Mapped NOS:CSC/NO115)**	**2.8.153 - 2.8.156**

SYLLABUS

Duration	Reference Learning Outcome	Professional Skills (Trade Practical) with Indicative hours	Professional Knowledge (Trade Theory)
Professional Skill 110 Hrs.; Professional Knowledge 30 Hrs.	Plan & set the machine parameter to produce precision engineering component to appropriate accuracy by performing different turning operation. [Appropriate accuracy - ±0.02mm/ (MT - 3) (proof turning); Different turning operation - Plain turning, taper turning, boring	88. Form turning practice by hand. (8 hrs.) 89. Re-sharpening of form tools using bench grinder. (2 hrs.) 90. Tool machine handle turning by combination feed. (15 hrs.)	Form tools-function-types and uses, Template-purpose & use. Dial test indicator- construction & uses Calculation involving modified rake and clearance angles of lathe tool at above and below the center height. Subsequent effect of tool setting. Jig and fixture-definition, type and use. Chip breaker on tool- purpose and type (09 hrs.)

Duration	Reference Learning Outcome	Professional Skills (Trade Practical) with Indicative hours	Professional Knowledge (Trade Theory)
	threading, knurling, grooving, chamfering etc.] (Mapped NOS: CSC/N0110)	91. Turn Morse taper plug (different number) and check with ring gauge / suitable MT sleeve. (20 hrs.) 92. Make revolving tail stock centre-Bush type (C-40). (Proof machining) (20 hrs.)	Cutting tool material-H.C.S., HSS, Tungsten. Carbide, Ceramic etc, - Constituents and their percentage. Tool life, quality of a cutting material. (13 hrs.)
		93. Make Morse taper sleeve and check by taper plug gauge. (25 hrs.)	Checking of taper with sin bar and roller-calculation involved (04 hrs.)
		94. Make mandrel/ plug gauge with an accuracy of ±0.02mm using tungsten carbide tools including throw-away tips. (20 hrs.)	Cutting speed, feed, turning time, depth of cut calculation, cutting speed chart (tungsten carbide tool) etc. Basic classification of tungsten carbide tips. (04 hrs.)
Professional Skill 40 Hrs.; Professional Knowledge 10 Hrs.	Set & Produce components on irregular shaped job using different lathe accessories. [Different Lathe accessories: - Face plate, angle plate] (Mapped NOS: CSC/ N0110)	95. Setting and turning operation involving face and angle plate (20 hrs.) 96. Make angle plate using face plate. (20 hrs.)	Accessories used on face plate - their uses. Angle plate-its construction & use. Balancing- its necessity. Surface finish symbols used on working blueprints- I.S. system lapping, honing etc. (10 hrs.)
Professional Skill 110 Hrs.; Professional Knowledge 30 Hrs.	Plan and set the machine using lathe attachment to produce different utility component/ item as per drawing. [Different utility component/ item - Crank shaft (single throw), stub arbour with accessories etc.] (Mapped NOS: CSC/ N0110)	97. Holding and truing of Crankshaft - single throw (Desirable). (45 hrs.)	Preventive maintenance, its necessity, frequency of lubrication. Preventive maintenance schedule., TPM (Total Productive Maintenance), EHS (Environment, health, Safety) Marking table-construction and function. Angle plate- construction, eccentricity checking. (12 hrs.)
		98. Turning of long shaft using steady rest (within 0.1 mm). (20 hrs.)	Roller and revolving steadies, Necessary, construction, uses etc. (06 hrs.)
		99. Use of attachments on lathe for different operations. (20 hrs.) 100. Turning standard stub arbor with accessories collar, tie rod, lock nut. (25 hrs.)	Different types of attachments used in lathe. Various procedures of thread measurement thread screw pitch gauge. Screw thread micrometer, microscope etc. (12 hrs.)
Professional Skill 80 Hrs.; Professional Knowledge 18 Hrs.	Set the machining parameters and produce & assemble components by performing different boring operations with an appropriate accuracy. [Different boring	101. Perform eccentric boringand make male & female eccentric fitting. (15 hrs.) 102. Position boring using tool maker's button. (10 hrs.)	Tool maker's button and its parts, construction and uses, telescopic gauge its construction and uses. (05 hrs.)
		103. Boring and stepped boring (within ± 0.05 mm) (10hrs.)	

Duration	Reference Learning Outcome	Professional Skills (Trade Practical) with Indicative hours	Professional Knowledge (Trade Theory)
	operation - eccentric boring, stepped boring; appropriate accuracy - ±0.05mm] (Mapped NOS: CSC/N0110)	104.Cutting of helical grooves in bearing and bushes (Oil groove) (10 hrs.)	Inside micrometer principle, construction graduation, reading, use etc. (Metric & Inch.) (05 hrs.)
		105.Turning & boring of split bearing - (using boring bar and fixture) (35 hrs.)	Care for holding split bearing. Fixture and its use in turning. (8 hrs.)
Professional Skill 110 Hrs.; Professional Knowledge 28 Hrs.	Calculate to set machine setting to produce different complex threaded component and check for functionality. [Different complex threaded component- Half nut, multi start threads (BSW, Metric & Square)] (Mapped NOS: CSC/ N0110)	106.Cutting thread of 8 and 11 TPI. (20 hrs.)	Calculation involving fractional threads. Odd & even threads. (04 hrs.)
		107.Multi start thread cutting (B.S.W.) external &internal. (25 hrs.)	Multiple thread function, use, different between pitch & lead, formulate to find out start, pitch, lead. Gear ratio etc. (04 hrs.)
		108.Multi start thread cutting (Metric) (External & internal). (20 hrs.)	Indexing of start - different methods tool shape for multi- start thread. Setting of a lathe calculation for required change wheel (06 hrs.)
		109.Multi-start thread cutting, square form (Male & Female). (25 hrs.)	Calculation involving shape of tool, change wheel, core dia etc. Calculation involving shape, size pitch, core dia. Etc.(05 hrs.)
		110.Make half nut as per standard lead screw. (20 hrs.)	Helix angle, leading angle & following angles. Thread dimensions-tool shape, gear, gear calculation, pitch, depth, lead etc. (09 hrs.)
Professional Skill 210 Hrs.; Professional Knowledge 62 Hrs.	Set (both job and tool) CNC turn centre and produce components as per drawing by preparing part programme. (Mapped NOS:CSC/ NO115)	111.Personal and CNC machine Safety: Safe handling of tools, equipment and CNC machine. (2 hrs.) 112.Identify CNC machine, CNC console. (3 hrs.) 113.Demonstration of CNC lathe machine and its parts bed, spindle motor and drive, chuck, tailstock, turret, axes motor and ball screws, guide ways, LM guides, console, controlswitches, coolant system, hydraulic system, chip conveyor, steady rest. (6 hrs.) 114.Working of parts explained using Multimedia based simulator for CNC parts shown on machine. (3 hrs.) 115.Identify machine over travel limits and emergency stop. (2 hrs.)	CNC technology basics: Difference between CNC and conventional lathes. Advantages and disadvantages of CNC machines over conventional machines. Machine model, control system and specification. Axes convention of CNC machine - Machine axes identification for CNC turn centre. Importance of feedback devices for CNC control. Concept of Co-ordinate geometry, concept of machine axis. (05 hrs.)
		116.Conduct a preliminary check of the readiness of the CNC turning centre viz., cleanliness of machine, referencing - zero return, functioning	Programming - sequence, formats, different codes and words. Co-ordinate system points and simulations.

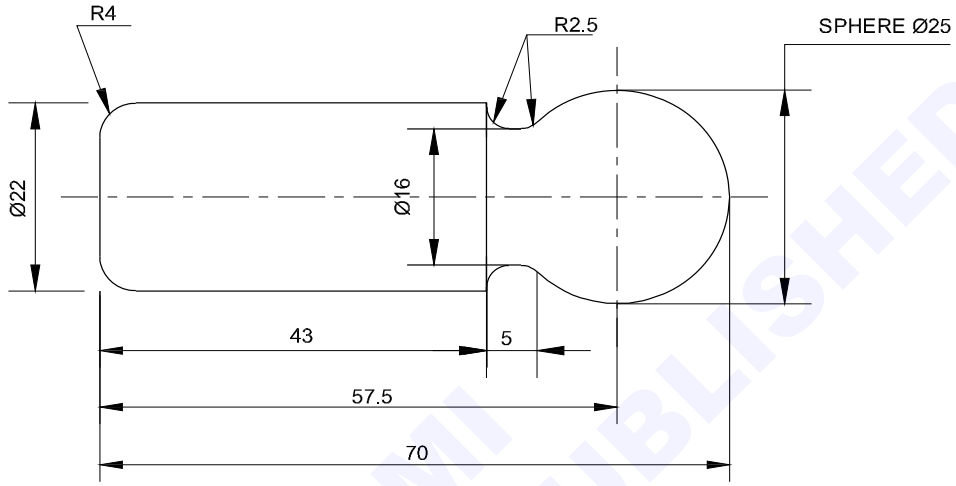
Duration	Reference Learning Outcome	Professional Skills (Trade Practical) with Indicative hours	Professional Knowledge (Trade Theory)
		of lubrication, coolant level, correct working of sub-system. (2 hrs.) 117.Identification of safety switches and interlocking of DIH modes. (1 hr.) 118.Machine starting & operating in Reference Point, JOG and Incremental Modes. (6 hrs.) 119.Check CNC part programming with simple exercises and using various programming codes and words. (05 hrs.) 120.Check the programme simulation on machine OR practice in simulation software in respective control system. (05 hrs.) 121.Absolute and incremental programming assignments and simulations. (05 hrs.) 122. Linear interpolation, and Circular interpolation assignments and simulations on software. (6 hrs.)	Work piece zero points and ISO/ DIN G and M codes for CNC. Different types of programming techniques of CNC machine. Describe the stock removal cycle in CNC turning for OD / ID operation. L/H and R/H tool relation on speed. Describe CNC interpolation, open and close loop control systems. Co-ordinate systems and Points. Program execution in different modes like manual, single block and auto. Absolute and incremental programming. Canned cycles. 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 catalog for various operations. Process planning & sequencing, tool layout & selection and cutting parameters selection. Tool path study of machining operations Prepare various programs as per drawing. (15 hrs.)
		123.Perform Work and tool setting: - Job zero/work coordinate system and tool setup and live tool setup. (10 hrs.) 124.Carryout jaw adjustment according to Diameter and tooling setup on Turret. (10 hrs.) 125.CNC turning centre operation in various modes: JOG, EDIT, MDI, SINGLE BLOCK, AUTO. (10 hrs.) 126.Program entry. (2 hrs.) 127.Set the tool offsets, entry of tool nose radius and orientation. (8 hrs.) 128.Conduct work off set measurement, Tool off set measurement and entry in CNC Control. (8 hrs.) 129.Make Tool nose radius and tool orientation entry in CNC control. (5 hrs.)	Tool Nose Radius Compensation (G41/42) and its importance (TNRC). Cutting tool materials, cutting tool geometry - insert types, holder types, insert cutting edge geometry. - Describe Tooling system for turning - Setting work and tool offsets. - Describe the tooling systems for CNC TURNING Centers. - Cutting tool materials for CNC Turning and its applications - ISO nomenclature for turning tool holders, boring tool holders, indexable inserts. - Tool holders and inserts for radial grooving, face grooving, threading, drilling.(17 hrs.)

Duration	Reference Learning Outcome	Professional Skills (Trade Practical) with Indicative hours	Professional Knowledge (Trade Theory)
		130.Jaw removal and mounting on CNC Lathe. (5 hrs.) 131.Manual Data Input (MDI) and MPG mode operations and checking of zero offsets and tool offsets. (7 hrs.)	
		132.Program checking in dry run, single block modes. (5 hrs.) 133.Checking finish size by over sizing through tool offsets. (5 hrs.) 134.Part program preparation, Simulation & Automatic Mode Execution for the exercise on Simple turning & Facing (step turning) (6 hrs.) 135.Part program preparation, Simulation & Automatic Mode Execution for the exercise on Turning with Radius / chamfer with TNRC. (6 hrs.) 136.Part program preparation, Simulation & Automatic Mode Execution of CNC Machine for the exercise on Blueprint programming contours with TNRC. (6 hrs.) 137.Machining parts on CNC lathe with parallel, taper, step, radius turning, grooving & threading. (10 hrs.) 138.Carryout Drilling /Boring cycles in CNC Turning. (12 hrs.) (First 60 % of the practice ison CNC machine simulator, followed by 40 % on machine.)	Prepare various part programs as per drawing & check using CNC simulator. Processes and Tool selection related to grooving, drilling, boring & threading. (10 hrs.)
		139.Geometry Wear Correction. Geometry andwear offset correction. (4 hrs.) 140.Produce components on CNC Machine involving different turning operations viz., • Stock removal cycle OD • Drilling / boring cycles • Stock removal cycle ID • Carryout threading in different pitches. (12 hrs.) 141.Produce components by involving turning operation and part programme exercises of CNC turning viz., • Grooving and thread cutting OD • Grooving and thread cutting ID • Threading cycle OD • Sub programs with repetition	- Describe Tapping on CNC turning. - Programming for Grooving/ Threading on OD/ID in CNC Turning. - Trouble shooting in CNC lathe machine - Identify Factors affecting turned part quality/ productivity. - Parting off operation explanation. - Bar feeding system through bar feeder. - Input and Output of Data. - DNC system. Interlacing with PC. - Use of CAM Programme. (Optional) (15 hrs.)

Duration	Reference Learning Outcome	Professional Skills (Trade Practical) with Indicative hours	Professional Knowledge (Trade Theory)
		• Using Sub Programs & Cycles in the Main Program. (12 hrs.) 142.Part off: Part Prog. (3 hrs.) 143.Produce job involving profile turning, threading on taper, boring, etc. operations. (15 hrs.) 144.Demo on M/C on bar feeding system. (simulation/ video) (1 hr.) 145.DNC system setup. (Optional) 146.Run the machine on DNC mode. (Optional) 147.CAM programme execution. (Optional) 148.Data Input-Output on CNC machine. (2 hrs.)	
Professional Skill 80 Hrs.; Professional Knowledge 20 Hrs.	Manufacture and assemble components to produce utility items by performing different operations & observing principle of interchangeability and check functionality. [Utility item: - screw jack/ vice spindle/ Box nut, marking block, drill chuck, collet chuck etc.; different operations: threading (Square, BSW, ACME, Metric), Thread on taper, different boring (Plain, stepped)] (Mapped NOS:CSC/ NO115)	149.Thread on taper surface (Vee form). (40 hrs.)	Setting of tools for taper threads-calculation of taper setting and thread depth. Heat treatment - meaning & procedure hardening, tempering, carbonizing etc. Different types of metal used in engineering application. (8 hrs.)
		150.Manufacturing & Assembly of Screw jack/vice/Box nut by performing different lathe operation. (To use earlier produce screw jack). (20 hrs.)	Interchangeability meaning, procedure for adoption, quality control procedure for quality production. (06hrs.)
		151.Prepare different types of documentation as per industrial need by different methods of recording information. (4 hrs.) 152.Turn Bevel gear blank. (16 hrs.)	Importance of Technical English terms used in industry -(in simple definition only)Technical forms, process charts, activity logs in required formats of industry, estimation, cycle time, productivity reports, job cards. (06 hrs.)
Professional Skill 100 Hrs.; Professional Knowledge 28 Hrs.	Make a process plan to produce components by performing special operations on lathe and check for accuracy. [Accuracy - ±0.02mm or proof machining & ±0.05mm bore; Special operation - Worm shaft cutting (shaft) boring, threading etc.] (Mapped NOS:CSC/NO115)	153.Read a part drawing, make a process plan for turning operation and make arbor with clamping nut (hexagonal). (40hrs.)	Terms used in part drawings and interpretation of drawings - tolerances, geometrical symbols - cylindricity, parallelism, etc. (11 hrs.)
		154.Practice of special operations on lathes - worm gear cutting. (Shaft) (20 hrs.)	Automatic lathe-its main parts, types diff. Tools used-circular tool etc. (09 hrs.)
		155.Boring on lathe using soft jaws to make bush with collar (standard) on nonferrous metal andcheck with dial bore gauge to accuracy of +/- 0.05 mm. (25hrs.) 156.Make Arbor support bush. (Proof Machining) (15hrs.)	Related theory and calculation. (8 hrs.)

Form turning practice (By hand)

Objectives: At the end of this exercise you shall be able to
- **manipulate the compound rest and cross slide for turning sphere**
- **use of Dial test indicator for truing a job**
- **check radius by using Radius gauge/Templete.**



Job Sequence

- Hold the job in a 4-jaw chuck and true it by a using dial test indicator.
- Set the turning tool (R.H) to centre height.
- Face one end.
- Turn dia. 22 mm x 43 mm long.
- Set the radius tool and turn radius R4 and check with radius gauge.
- Reverse the job and true it by using dial test indicator on dia 22.
- Face the other end and maintain total length of 70 mm.
- Form turn the sphere of 25 mm diameter, manipulating the cross-slide and compound slide.
- Form R 2.5 mm radius for a width of 5 mm as per drawing.
- Check the turned sphere with a template.

Do not use radius gauge while the job is rotating, so also the fillet gauge.

Use appropriate speed and feed according to the diameter.

Clean and oil the machine after the job is finished.

1	Ø30-75		Fe 310 IS:1977			2.1.88
NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.
SCALE 2:1	**FORM TURNING PRACTICE (BY HAND FEED)**				DEVIATIONS IS:2102(M)	TIME:
					CODE NO. TU20N2188E1	

Skill Sequence

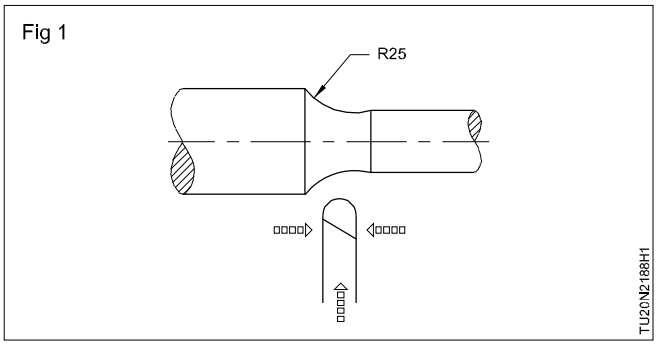
Free hand form turning

Objective: This shall help you to
- **turn concave and convex forms, coordinating the carriage and cross-slide movements manually.**

Note: To produce a concave form easily and accurately, use a tool bit having a radius as close as possible to the desired form.

Turning Concave form (Fig 1): Set the lathe speed as near as possible to the normal turning speed without producing chatter.

Fig 1



Move the carriage to bring the edge of the radius tool bit close to the start of the concave form.

Place one hand on the carriage hand wheel and the other hand on the cross-slide handle.

Slowly turn the carriage hand wheel, feeding the tool bit towards the centre of the form while with the other hand turn the cross-slide handle clockwise, moving the tool into the work.

Co-ordinate the amount of carriage movement in relation to the cross-slide to the desired form. For this, practice is required.

Take successive cuts from each side, always starting at the large end of the form and working towards the centre. It is wise to take a light out until proper co-ordination between the carriage and cross-slide is attained.

Check the small diameters of the form with a caliper and the form with a radius gauge.

Note: If a radius gauge is not available, obtain a piece of steel rod having the correct diameter. Use its circumstance to check the profile.

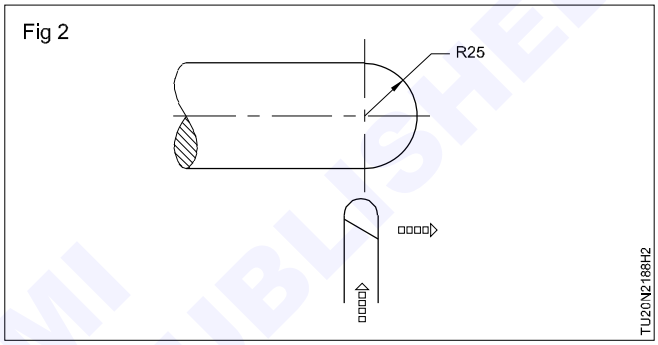
Cut the form close to the diameter.

With a half round file, remove the marks and finish the form.

Improve the surface finish with an abrassive cloth.

Turning a convex form (Fig 2): The same procedure that was adopted to machine a concave form can be used to produce a convex form. The following steps outline the procedure for machining 25 mm radius convex surface on a workpiece.

Fig 2



Using a round nose tool bit, take cuts starting at the 25 mm line and feeding towards the end of the work and centre.

Note: For the first half of the raius the carriage must be moved faster than the cross-slide handles. From this point to the end of the work the cross-slide handle must be moved faster than the carriage hand wheel.

Take successive cuts, always starting at the outside diameter and cutting towards the face centre.

Check the accuracy of the form with a radius gauge.

Machine the radius as close to the size and shape as possible.

With a suitable file remove the rough machining marks and finish the form.

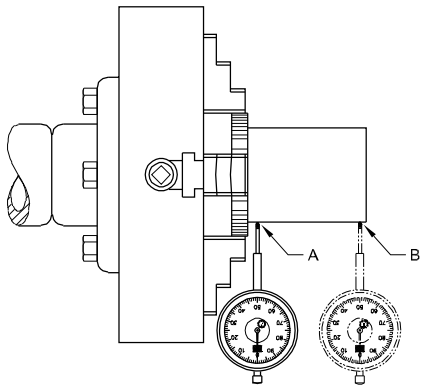
Setting using dial test indicator

Objective: This shall help you to

- **using a Dial test indicator for truing a job.**

- Set the job in four jaw chuck using dial test indicator. (Fig 1)

Fig 1



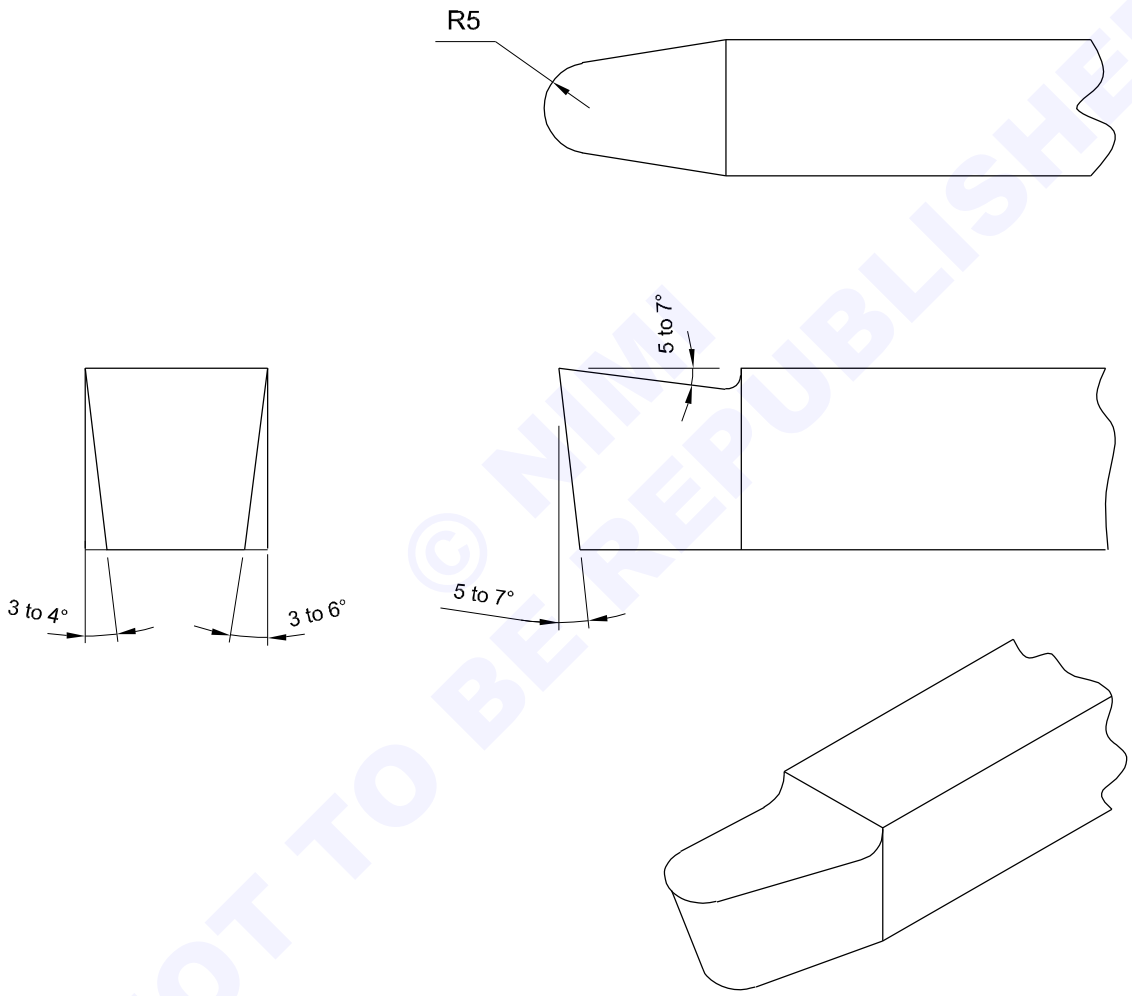
WORK MUST BE TRUED IN THE FOUR-JAW CHUCK
SO THAT THE INDICATOR READINGS AT A & B ARE THE SAME.

TU20N2188.J1

- Mount the dial stand on the machine table.
- Fix the dial and touch on the job.
- Move the dial and rotate the job.
- Check the trueness of the job with the indicator at zero position.
- Incase difference reading the jaws adjust and rotate the chuck.
- The dial moving at A and B are the same reaching at zero.

Resharpening of form tool using bench grinder

Objectives: At the end of this exercise you shall be able to
- **practice the tool movement for grinding a radius form.**
- **checking of radius using gauge/template.**



1	WORN OUT TOOL					2.1.89
NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.
SCALE 1:1	**RE-SHARPENING OF FORM TOOLS (USING BENCH GRINDER)**				TOLERANCE	TIME:
					CODE NO. TU20N2189E1	

Job Sequence

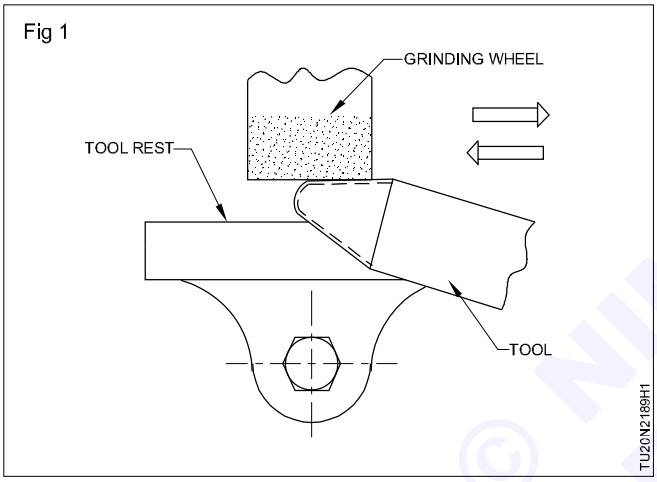
Safety

- Make sure the grinding wheel guards are in place.
- Wear safety goggles.
- Check the wheel for glazing/loading.
- Adjust the tool rest close to the wheel.
- Before attempting to grind the tool, stand on a side and start the machine.

To grind convex profile

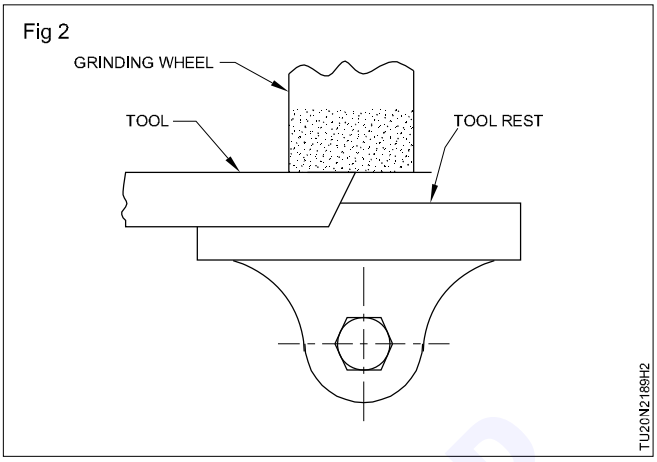
- Place the heel of the tool on the support.
- Tilt the shank about 5° to 7°.
- Hold the middle part of the tool with the left thumb and the forefinger symmetrical to the right hand side. (Fig 1)

Fig 1



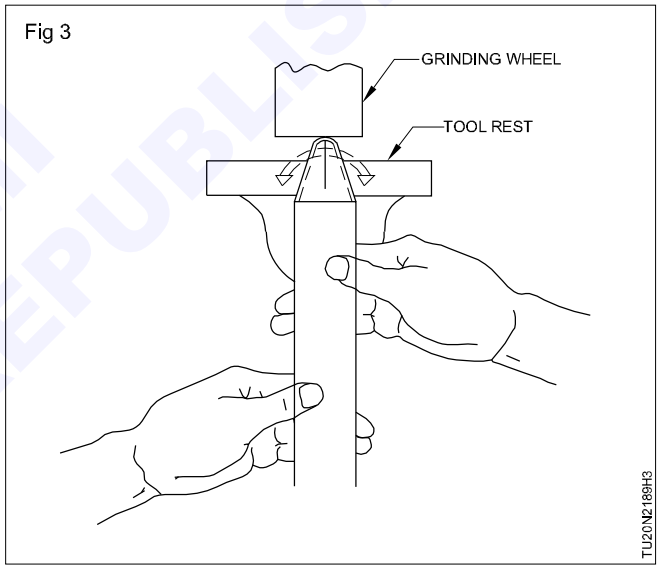
- Move the left hand left to right with the right hand as a pivot, and grind half the slide of the nose.
- Changing the position of the hand, grind the other half side of the nose.
- Hold the base of the tool tip with the right thumb and forefinger and hold the shank by the left hand and file the rake surface by pressing the tool against the wheel face. (Fig 2)

Fig 2



- Hold the tool as shown in Fig 3. Tilt the base of the tool to about 5° to 7°, press the cutting edge surface to the grinding wheel parallel to its outer periphery and grind the side cutting edge and side clearance.

Fig 3

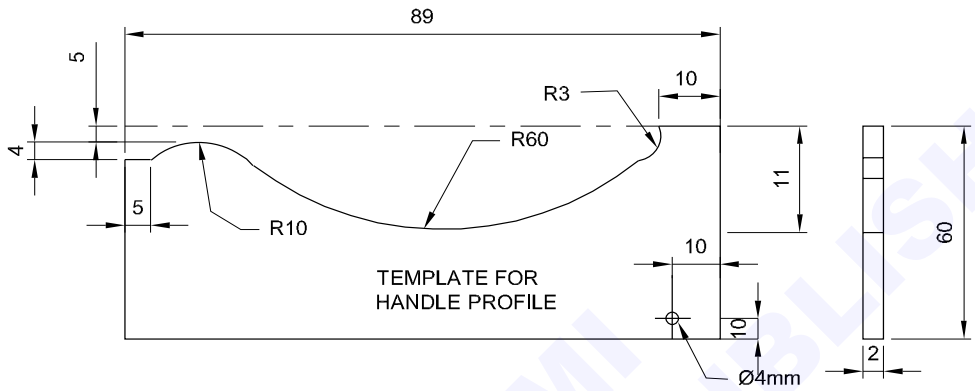


- Hone the ground surface with an oilstone.

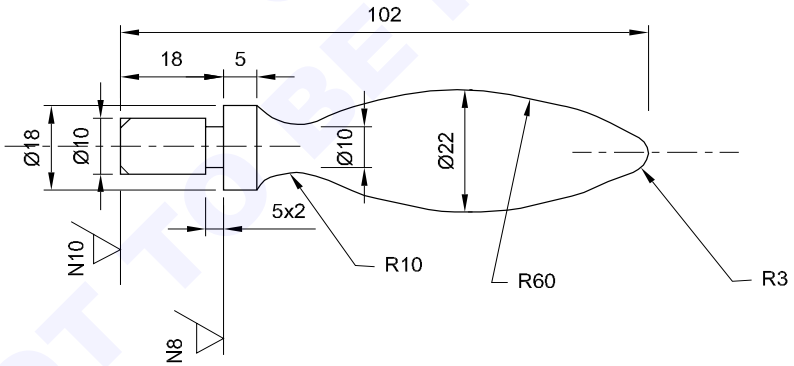
Machine handle turning by combination feed

Objectives: At the end of this exercise you shall be able to
- **make template to the given profile**
- **use a dial test indicator for truing a job**
- **check the form/profile by using template.**

TASK 1



TASK 2



1	ISSH 92x62x2 mm		M.S			2.1.90 TASK-1
1	Ø25-105		Fe 310 IS 1977			2.1.90 TASK-2
NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.
SCALE 1:1	**MACHINE HANDLE TURNING (BY COMBINATION FEED)**				DEVIATIONS	TIME:
					CODE NO. TU20N2190E1	

Job Sequence

TASK 1 : **Making template**

- Mark the given sheet to the required size (keeping 2 mm excess on length & width) i.e., 92 x 62
- Take a cardboard sheet & draw the profile.
- Cut the profile using a scissor.
- Paste the cordboard sheet to the prepared metal sheet.
- Punch as per the profile using dot punch for reference.
- Now by using straight & curved snips cut as per the profile pasted to it (leaving atleast 0.5 mm stock for finishing.
- Finish the profile by using smooth file
- File & maintain dimension as per drawing.
- Flattern the job by using mallet.
- Check squareness with try square.

– – – – – – – –

TASK 2 : **Form turning**

- Hold the job in four jaw chuck projecting 40 mm length (Approximately)
- True it by using universal surface gauge.
- Set the facing tool to the correct centre height.
- Face finish the end.
- Set the RH Turning tool to the correct centre height.
- Turn to dia 18 mm for a length of 33 mm.
- Reverse the job by holding on dia 18 mm by gripping for a length of 20 mm.
- Set the dial test indicator & true it for Ø 18 mm.
- Set the facing tool & face the end to maintain a length of 102 mm.
- Set the form tool & turn to a given profile gradually by taking several rough cuts by combined feed method.
- Check the profile by using template (prepared by Task 1) & finish the job.

Avoid overhang of the cutting tool.

Use plenty of coolant while machining.

Give appropriate speed & feed according to the diameter.

- Reverse the job by holding on dia 18 for a length of 10 mm.
- True it by using dial test indicator.
- Set the RH Turning tool & turn Ø 9.85 mm for a length of 18 mm.
- Make chamfer of 1.5 x 45
- Make a groove of 5 mm width and 2 mm length.
- Form M10 thread by using dia (By using split dia of using adjustable screw plate dia)
- Check for M10 standard (by thread plug gauge)

– – – – – – – –

Skill Sequence

Form turning by cross-slide and compound rest

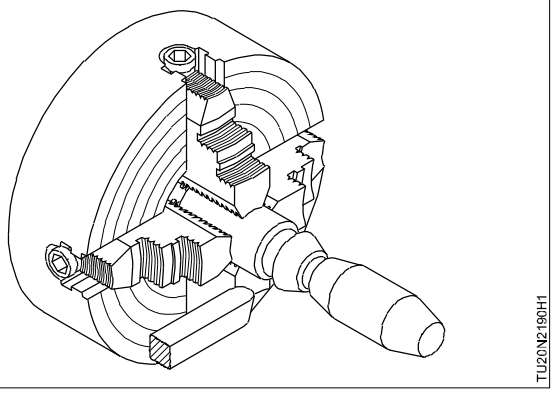
Objectives: This shall help you to
- **turn a given form manipulating cross-slide and compound rest**
- **check the profile with template.**

Form turning to suit template

Turn to shape of profile gradually by taking several roughing cuts, using the cross-slide and the compound slide simultaneously.(Figs 1 & 2)

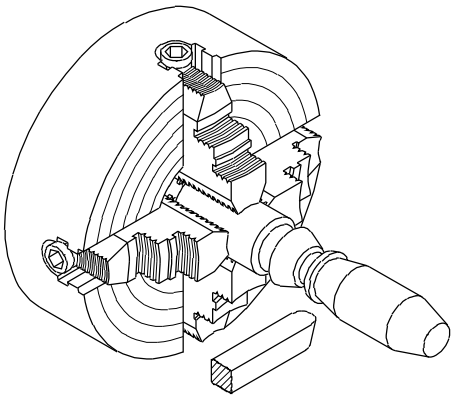
Turn wherever possible from small diameter to larger dia. (Fig 3)

Fig 1



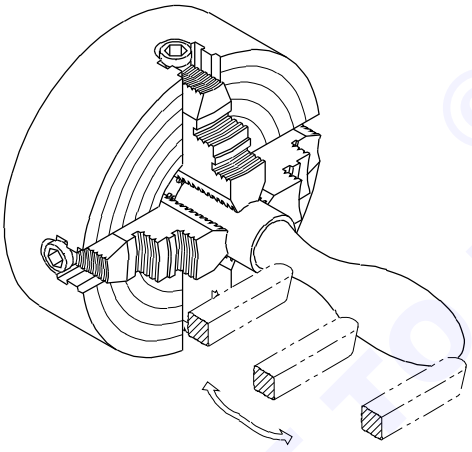
TU20N2190H1

Fig 2



TU20N2190H2

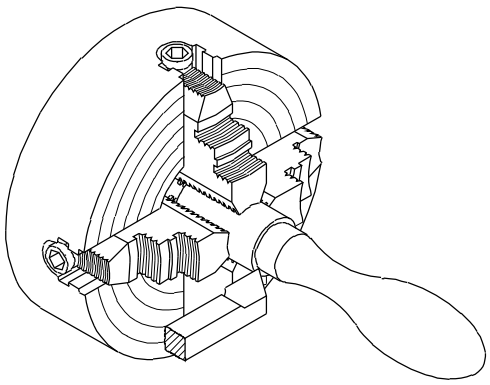
Fig 3



TU20N2190H3

Finish turn the plain step after form turning. (Fig 4)

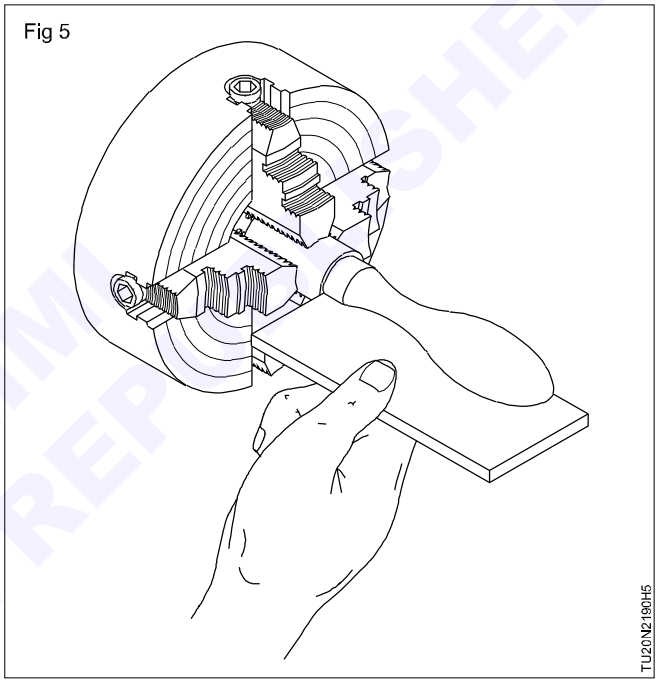
Fig 4



TU20N2190H4

A simple sheet metal template may be used to check the form. (Fig 5)

Fig 5



TU20N2190H5

General information

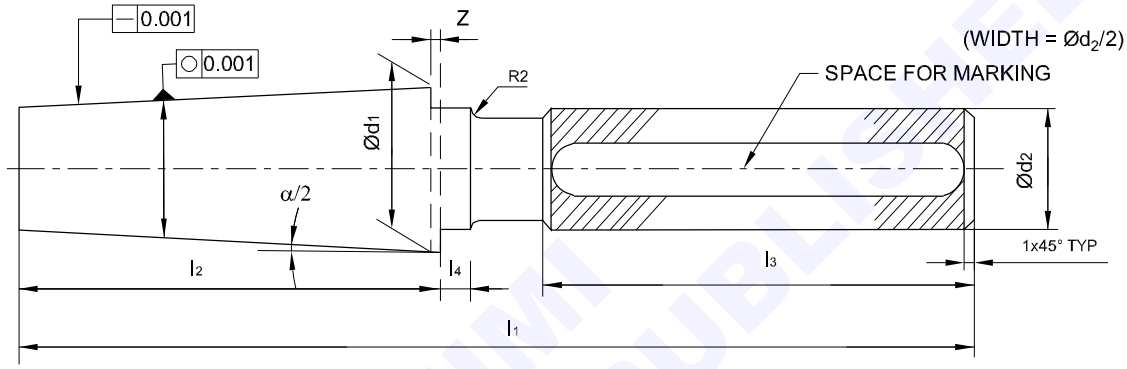
Form tools have a much longer cutting edge than ordinary turning tools.

The tool should be in a very good condition. Care must be taken to ensure maximum rigidity of the tool and workpiece. Slower speed must be used.

Turn morse taper plug (Different numbers)

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

- **use of different cutting tool material**
- **setting of taper turning attachment to turn taper (for Task 1)**
- **using a taper ring gauge**
- **turn taper of compound slide swivel base method (for Task 2).**



DESIGNATION OF TAPER		d1 js5	d2	l1	l2 js8	l3	l4	Z ±0.05	α/2
TASK-1	MT 3	23.825	17.5	176	81	80	5	1.0	1°26'16"
TASK-2	MT 2	17.780	14.5	158	64	80	4	1.0	1°25'50"

1	Ø25-162	-	C-45	-	-	2.1.91 TASK-2
1	Ø25-180	-	C-45	-	-	2.1.91 TASK-1
NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.

SCALE 1:1	**TURN MORSE TAPER PLUG (DIFFERENT NUMBERS)**	TOLERANCE	TIME:
		CODE NO. TU20N2191E1	

Job Sequence

TASK 1 & 2 : **Turning morse taper**

- Set the job in four jaw chuck. Projecting ($l_1 - l_2 + 10$mm) outside the chuck.
- True it by universal surface gauge.
- Set the carbide tip tool to the correct centre.

  (For facing - offset facing tool ISO 3 F 25 P 30

  or

  For facing & turning in one tool - ISO 5 F 25 P 30 offset & turning & facing tool

  For turning - offset side cutting tool ISO 5 F 25 P 30)
- Set the spindle speed as per the cutting speed chart.
- Face one end.
- Turn dia (d_2) for a length equal to ($l_1 - l_2$).
- Step turn dia ($d_2 - 0.2$) for a length of l_3 (for knurling).
- Form grooving (one end R2 & another end square shoulder) after leaving l_3 from the end & maintain dia ($d_2 - 2$mm)
- Chamfer the two ends of dia d_2 to 1 x 45°.
- Form knurling & maintain dia d_2.
- Mark the job for filing space for marking by using 'V' block with 'U' clamp. Vernier height gauge by placing on suitable marking table.
- File the surface & maintain a width of $d_2/2$ mm.
- Reverse the job & hold on knurled dia by giving Aluminium/Copper sheet as a packing.
- True the job by using dial test indicator (By dialling on dia d_2 step).
- Face the end to maintain a length of l_1.
- Turn dia d_1 & check by using Vernier micrometer.
- Set the taper turning attachment to turn a taper of 1° 26' 16" for Task 1 and
- Set the compound slide swivel base to turn a taper of 1°25' 50" for Task 2.
- Turn taper & check the dimensions as per the drawing by using Vernier micrometer & Vernier caliper & Vernier bevel protractor.
- Remove the job & clamp it in bench vice holding on knurled dia by giving soft packing.
- File on taper end, step of $1^{\pm 0.05}$ by using safe edge file.
- Check the depth by using depth micrometer.

Skill Sequence

Producing taper by using taper turning attachment

Objectives: This shall help you to
- **set the taper turning attachment to the required angle**
- **produce taper by using a taper turning attachment.**

A taper turning attachment provides a quick and accurate means of turning tapers. (Fig 1)

The following procedure is to be followed during turning taper using a taper turning attachment.

Check for backlash between the guide bar and the sliding block, and adjust, if necessary.

Clean and oil the guide bar.

Loosen the locking screws, then swivel the guide bar to the required angle.

Tighten the locking screws.

Adjust the base plate until the ends of the guide bar are equidistant from the cross-slide extension.

Set up the cutting tool on exact centre.

Any error will result in an incorrect taper

Mount the workpiece on the chuck or between centres.

Adjust the carriage until the cutting tool is approximately opposite to the centre of the tapered section.

Set up the cutting tool for the correct position.

Wear safety goggles

Set the required r.p.m.

Feed the cutting tool in until it is about 6 mm from the work surface.

Remove the locking screws which connect the cross-slide and the cross-slide nut.

Use the binding lever to connect the cross-slide extension and sliding block.

Insert a suitable plug in the hole on the top of the cross-slide to protect the cross- slide screw from dirt and metal chips.

The compound slide must now be used to feed the cutting tool into the work.

Move the carriage to the right until the cutting tool is about 12 mm away from the right hand end of the workpiece.

This removes any play in the moving parts of the taper turning attachment

Switch on the lathe.

Take a light cut about 2 mm long and check the end of the taper for size.

Set the depth of the roughing cut.

Machine the work as with plain turning.

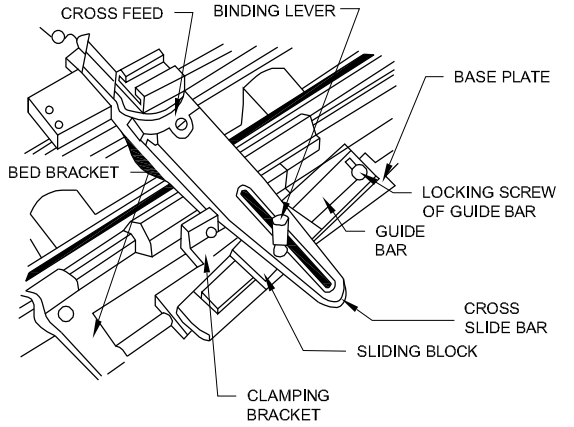
Remove the play by moving the cutting tool 12 mm beyond the right hand end of the work at the beginning of each cut.

Check the taper for fit.

Readjust the taper turning attachment, if necessary; take a light cut and recheck the taper.

Finish the taper to size and fit it to the taper gauge.

Fig 1

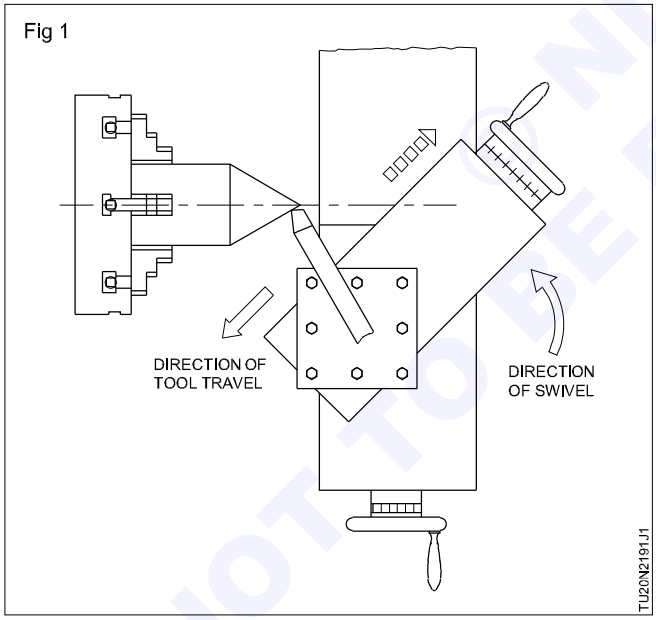


Turning taper by compound slide swivelling

Objectives: This shall help you to
- **turn the taper using a compound slide**
- **check the taper with a vernier bevel protractor.**

One of the methods of turning taper is by swivelling the compound slide and feeding the tool at an angle to the axis of the work by hand feed. (Fig 1)

Fig 1



Set and true the job turned to the bigger diameter of taper.

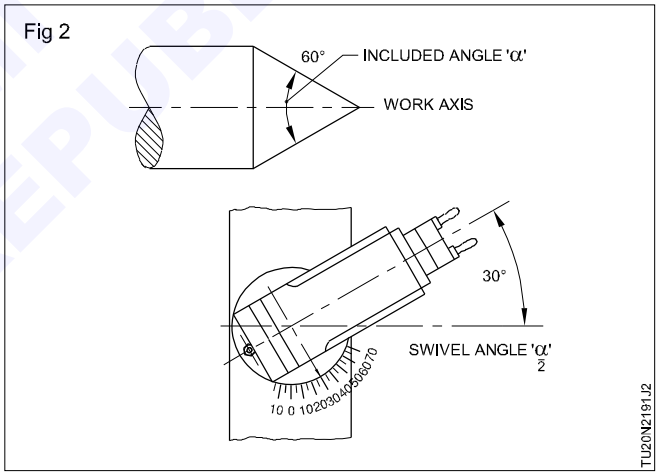
Set the machine to the required rpm.

Loosen the top slide clamping nuts.

Swivel the top slide to half the included angle of the taper as shown in Fig 2.

Ensure that equal pressure is exerted by the spanner for both the nuts.

Fig 2



Fix the turning tool in the tool post to the correct centre height.

Keep a minimum overhang of the tool.

Set the top slide to the rearmost position.

Position the saddle such that the tool is able to cover the full length of the taper to be turned.

Ensure that the top slide does not travel beyond the edge of the base.

Lock the carriage in position.

Touch the tool to the work surface during running and set the cross-slide graduated collar to zero.

Bring the tool to clear off the work by the top slide hand wheel movement.

Give a depth of cut by the cross-slide and feed the tool by the top slide hand wheel till the tool clears from the work.

Feeding by the top slide must be uniform and continuous.

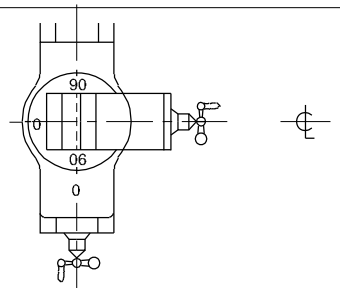
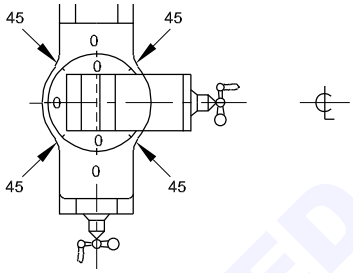
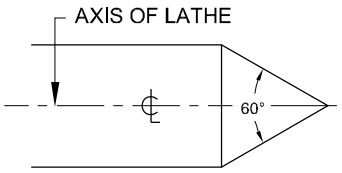
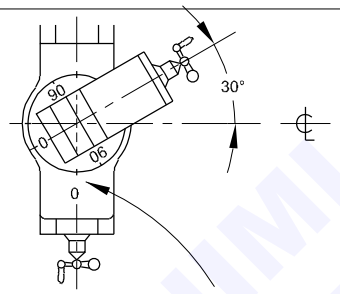
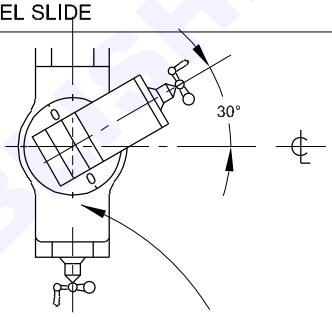
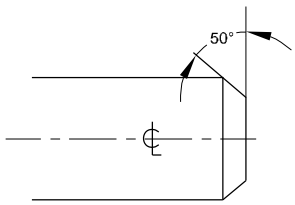
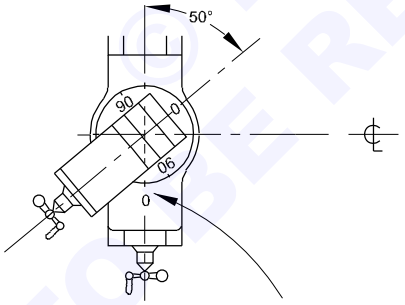
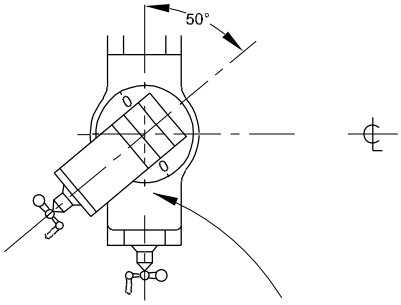
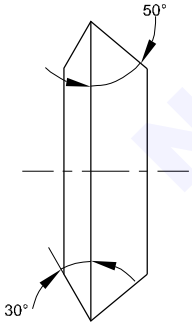
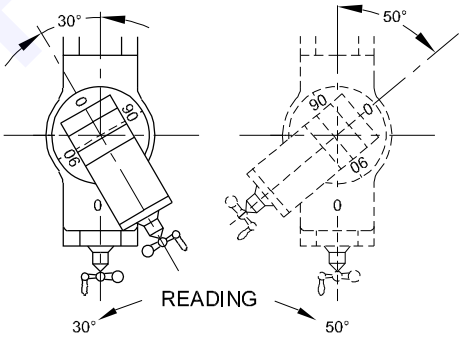
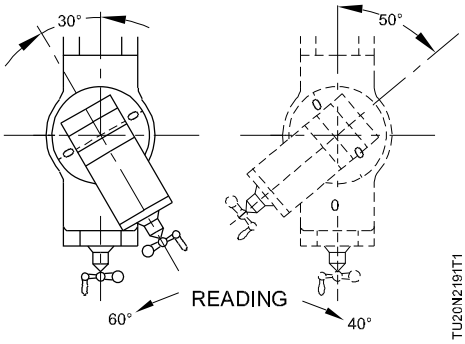
Give successive cuts by the cross-slide, feeding by the top slide each time.

Check the angle of the turned job with a vernier bevel protractor.

Adjust the swivel if there is any difference.

Continue the taper turning and finish the taper.

COMPOUND REST SET UP FOR TURNING VARIOUS ANGLES

	ARRANGEMENT OF GRADUATIONS ON SWIVEL SLIDE	
SPECIAL ANGULAR SETTING ON COMPOUND REST	 GRADUATED FROM 90-0-90	 GRADUATED FROM 0-45-0
EXAMPLES	READINGS ON GRADUATED SWIVEL SLIDE	
 INCLUDED ANGLE MEASURED IN HORIZONTAL PLANE	 READING ON SCALE 60°	 READING ON SCALE 30°
 ANGLE GIVEN FROM A LINE AT 90° TO AXIS OF LATHE	 READING ON SCALE 50°	
		

TU20N2191T1

Checking tapers

Objectives: This shall help you to
- **check the taper using sine bar**
- **check the taper using ring gauges/taper plug gauges.**

Select suitable size sine bar and clean.

Calculate the height of slip gauges required for checking the angle.

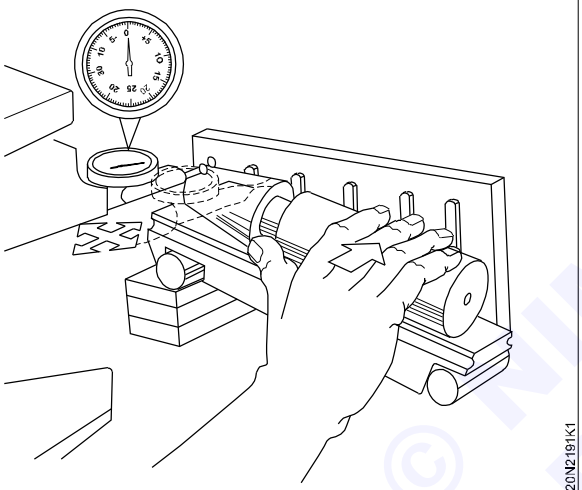
Select minimum number of slip gauges to check required angle.

De-burr and clean the job.

Clean and “Wring” slip gauges together.

Position the job on the sine bar but against plate on surface plate as shown in Fig 1.

Fig 1



Keep the tapered job on the top of the sine bar and with a reference diameter but against the angle plate.

Ensure that the sine bar angle plate and job are parallel to each other.

Position dial indicator on surface plate.

Place the styles to one end of workpiece taper.

Slide stylus across width of taper to check the highest point.

Set the indicator dial to zero position.

Check for zero reading at other end of taper.

Note amount of error.

If angle is correct, a zero reading should be obtained at both ends of taper.

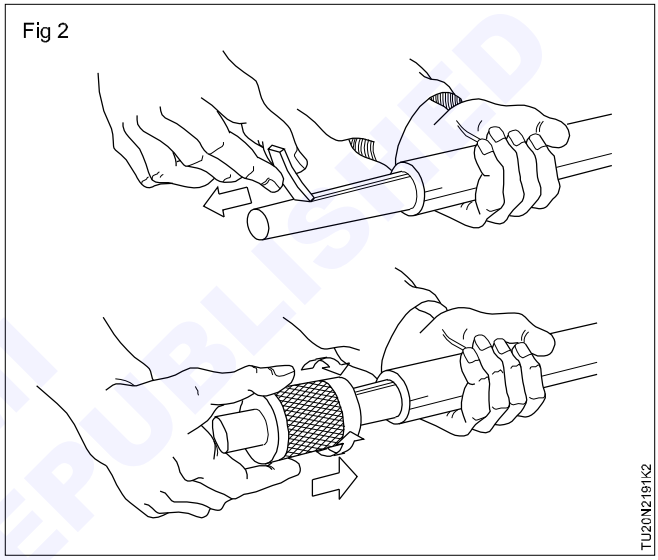
Standard external tapers are checked with the help of taper ring gauges and internal tapers are checked with taper plug gauge.

Select appropriate taper ring gauge/taper plug gauge.

Clean gauge and taper portion of the job.

Apply a thin line of marking medium down the length of the workpiece taper in three positions. (Fig 2).

Fig 2

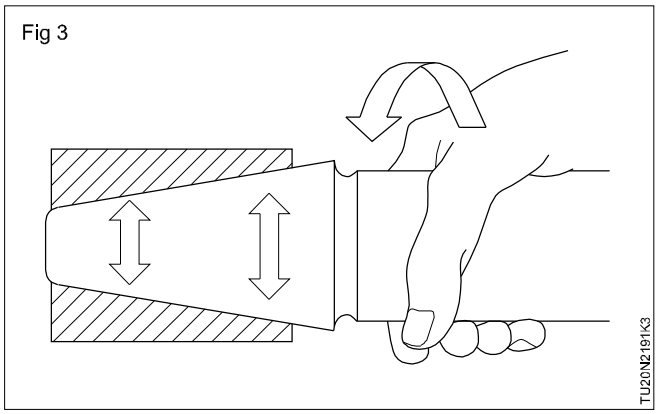


Carefully insert workpiece taper into gauge.

Push the gauge on taper as far as possible and feel for any ‘rock’.

Apply slight pressure and turn gauge on taper (Fig 3).

Fig 3



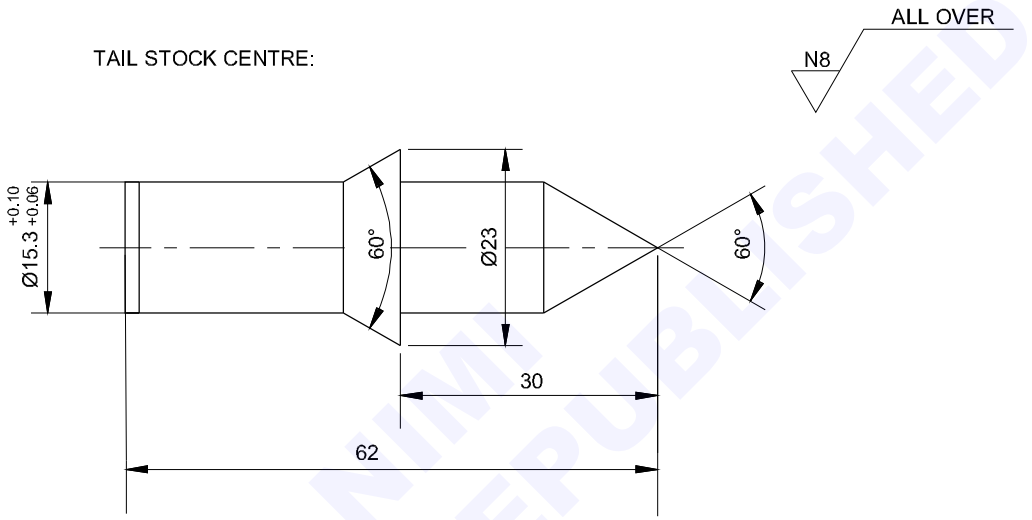
Remove gauge and visually check marking medium is spread equally along and around taper.

If the marking medium is unequal, taper is incorrect

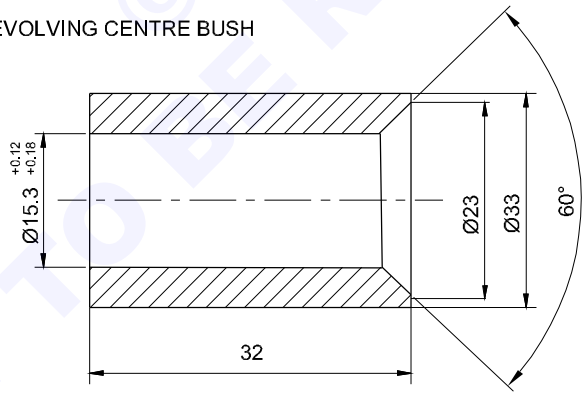
Make revolving tail stock centre (Bush Type)

Objectives: At the end of this exercise you shall be able to
- **rough turn between the chuck and the centre**
- **check the Bore using three point internal micormeter**
- **finish turn work to an accuracy of ±0.06mm and check with a micrometer**
- **turn taper with an accuracy of ±5' using a compound slide**
- **use a L.H. turning tool.**

TASK-1 TAIL STOCK CENTRE:



TASK-2 REVOLVING CENTRE BUSH



NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.
1	Ø36-50	-	CuSn8Zn4	-	-	2.1.92 TASK-2
1	Ø25-90	-	C40	-	-	2.1.92 TASK-1

SCALE 1:1	MAKE REVOLVING TAIL STOCK CENTRE (BUSH TYPE)	TOLERANCE ±0.01 UNLESS OTHERWISE STATED	TIME:
		CODE NO. TU20N2192E1	

Job Sequence

TASK 1 : **Tail Stock Centre**

- Check the raw material for its size.
- Hold the job in a four jaw chuck, keeping about 75 mm outside and true.
- Face the job at the overhanging end.
- Centre drill and support with the centre.
- Turn a dia. Ø 23 for maximum possible length.
- Turn a step dia. $Ø 15.3^{+0.1}_{+0.06}$ to 35 mm length.
- Undercut to Ø 15.4 mm at 68 mm + the width of the undercut tool from face.
- Widen the undercut width to permit the L.H. tool entry. (Widen towards the chuck side)
- Fix the L.H. tool in the tool-post and position it in the undercut already made.
- Allow the tool tip to touch Ø 15.4 mm and feed the tool towards the tailstock up to 42 mm from the right hand face.
- Finish turn $Ø 15.3^{+0.1}_{+0.06}$
- Swivel the compound slide for 30°, turn taper using a L.H. tool.
- Remove the tailstock centre and face the work to remove the centre hole and maintain 30 mm length from the face.
- Turn taper at the end to form and including angles of 60°.
- Part off the work maintaining a length of 62.2 mm.
- Reverse and hold the work on $Ø 15.3^{+0.1}_{+0.06}$ mm and true.
- Face the work for a total length of 62 mm.
- Remove burrs.
- Check the taper angle with a vernier bevel protractor.

This is not an easy method of doing this work. This procedure is followed only to practise the use of a L.H. turning tool.

TASK 2 : **Revolving Centre bush.**

- Check the raw material for its size.
- Hold the job in four jaw chuck, keeping about 40 mm outside & true.
- Face the end.
- Turn the job to Ø 33 for maximum length.
- Centre drill the job.
- Drill to dia 15 mm through hole.
- Set the boring tool & plain bore to $Ø 15.3^{+0.12}_{+0.18}$.
- Check the bore using three point internal micrometer.
- Assemble it with Task 1 job to check for sliding fit.
- Set the compound slide to form taper of 30°.
- Turn taper to maintain dia 23 mm & at the same time suit with the Task 1 job.
- Adjust the taper if any changes & finish it.
- Set the parting off tool.
- Part off the job to a length of 32.2 mm.
- Set the job & face to maintain 32 mm.
- Deburr all sharp corners if any.

Skill Sequence

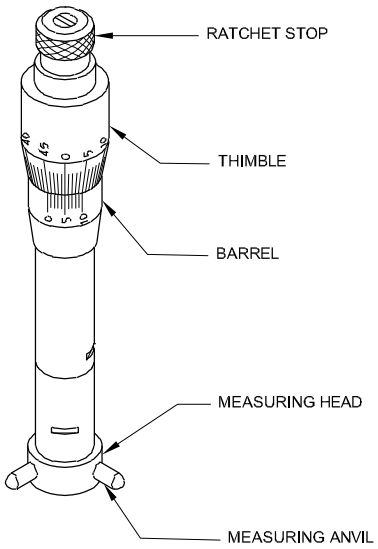
Checking the bore using three point internal micrometer

Objective: This shall help you to
- **check the internal diameter accurately.**

Checking the bore using three point internal micrometer. (Fig 1)

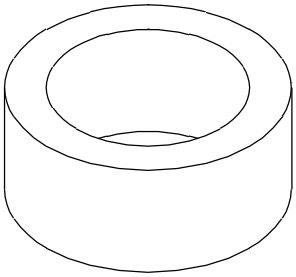
The component to checked Fig 2 is given below.

Fig 1



TU20N2192H1

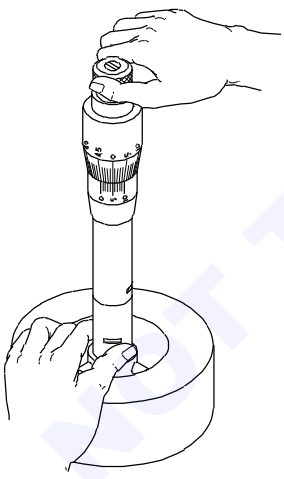
Fig 2



TU20N2192H2

Before taking measurement, the zero setting has to be checked using the setting ring. (Fig 3)

Fig 3



TU20N2192H3

The position of the anvil can be reset by loosening the barrel using a screw driver provided for this purpose.

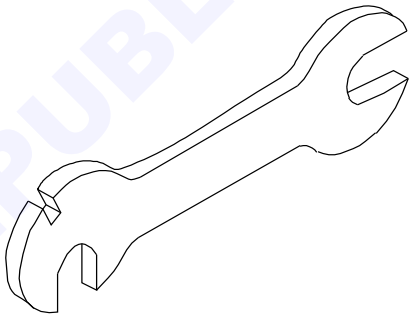
Depending on the depth of the bore the length of the micrometer can be varied using the extension rod (Fig 4).

Fig 4

TU20N2192H4

A set of spanners is provided for assembling and disassembling the extension rod. (Fig 5)

Fig 5



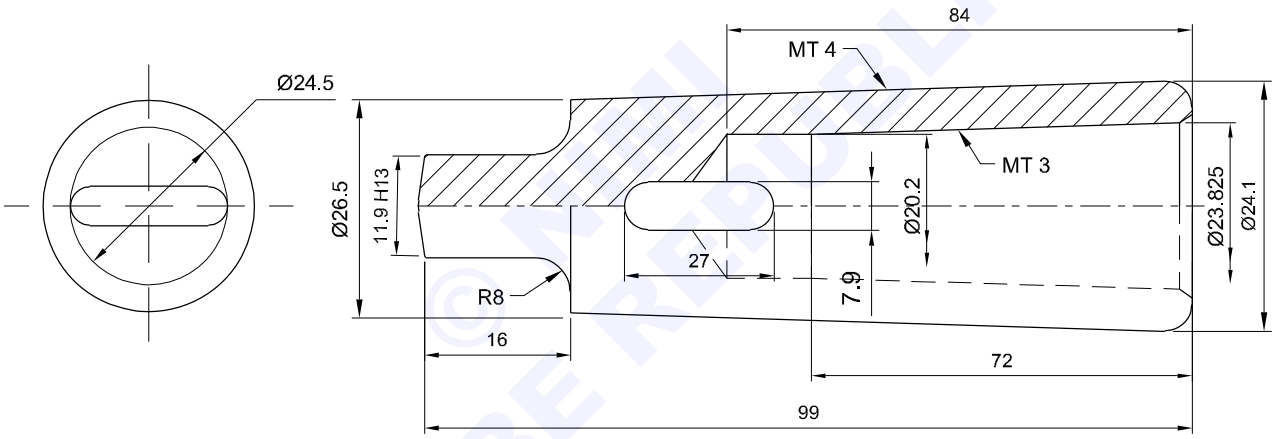
TU20N2192H5

The instruments are available in various sizes and forms for measurring different sizes.

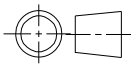
They are also available in analogue or digital read-outs.

Make morse taper sleeve & check by taper plug gauge

Objectives: At the end of this exercise you shall be able to
- **turn taper by attachment method**
- **check the internal taper by using taper plug gauge.**



1	Ø40 -150	-	C-45	-	-	2.1.93
NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.

SCALE NTS	**MAKE MORSE TAPER SLEEVE AND CHECK BY TAPER PLUG GAUGE**	TOLERANCE	TIME:
		CODE NO. TU20N2193E1	

Job Sequence

- Check the raw material for its size.
- Hold the job in four jaw chuck, keeping about 125 mm outside the chuck & True it.
- Set the tool for the correct centre height.
- Face the end.
- Turn to dia 32 mm for maximum possible length.
- Centre drill it.
- Drill to Ø 20 for a depth of 84 mm.
- Set the boring tool & plain bore to dia 20.2 mm for 84 mm depth.
- Set the taper turning attachment to bore a taper hole of MT-3.
- Bore taper to maintain a given dia & depth (D_1 & l_2 respectively).
- Check the taper with MT-3 taper plug gauge (Finished product of 2.1.91)
- Now support the job with pipe centre.
- Set the grooving tool.
- Make groove to dia 24.5 mm leaving 109 mm from the face to a length of 117 + 4 mm (Parting Allowance)
- Set the taper turning attachment to turn a taper of MT-4.
- Set the R.H turning tool & turn taper & maintain the dimensions.
- Set the parting off tool.
- Part off job for a length of 117.2 mm.
- Mark the job to make a slot & Tang portion.
- Chain drill & file the slot to maintain the given size. (7.8 mm width & 27 mm length)
- Hold/Set the MT-3 taper plug gauge, taper mandrel in four jaw chuck & true it by dial test indicator.
- File the tang portion to a thickness of 11.9 h_{13} & make radius of R 8 at the step.
- Insert the prepared job to it.
- Set the facing tool & face the job to maintain a length of 117 mm.
- Make radius of R 20 at the face end.
- Remove the job from a taper plug gauge/taper mandrel using a drift.
- Check the external taper by using rollers & slip gauges.

Skill Sequence

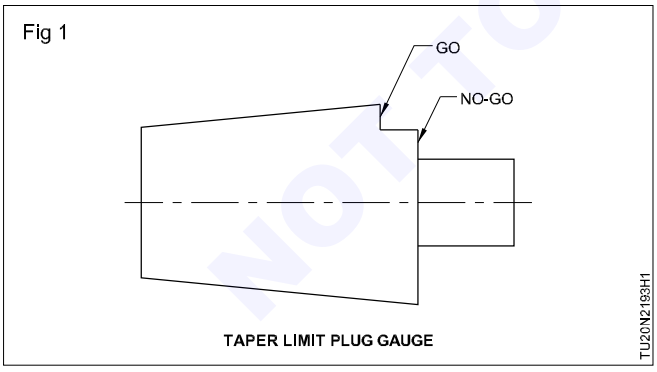
Checking a tapered bore using a taper limit plug gauges

Objective: This shall help you to

- **check the internal taper with taper plug gauge.**

A taper limit plug gauge ensures the accuracy of the angle and the linear dimensions of the taper bore. (Fig 1)

Fig 1

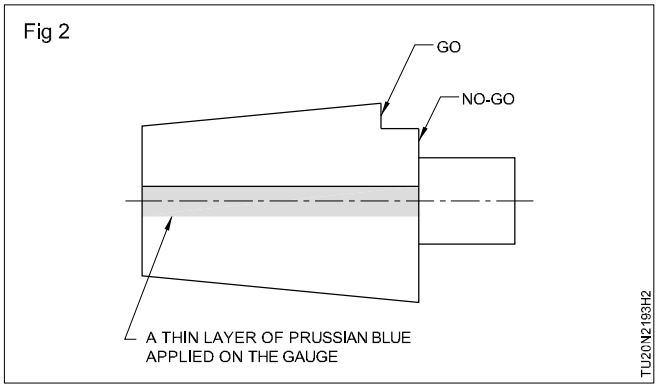


Clean the tapered bore.

Clean the taper limit plug gauge.

Apply a thin layer of Prussian blue on the taper limit plug gauge along its length. (Fig 2)

Fig 2

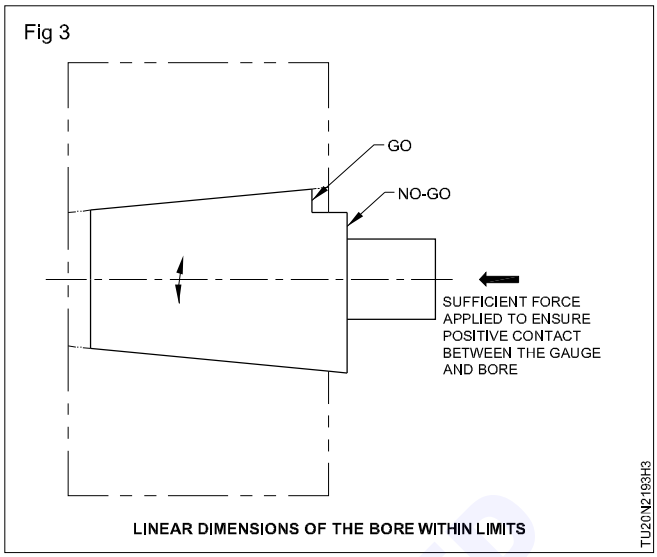


Assemble the taper plug gauge inside the tapered bore carefully with sufficient force to ensure positive contact between the gauge and the bore, and give one quarter twist to the plug gauge.

Carefully remove the taper limit plug gauge and check if the Prussian blue is rubbed off uniformly, atleast to about 75% of its area. This ensures the accuracy of the angle required.

Then once again insert the taper plug gauge inside the taper bore and check. If the big dia. end of the bore falls within the 'Go' and 'No-Go' limits marked on the gauge, this ensures the dimensional accuracy of the tapered bore. (Fig 3)

Fig 3

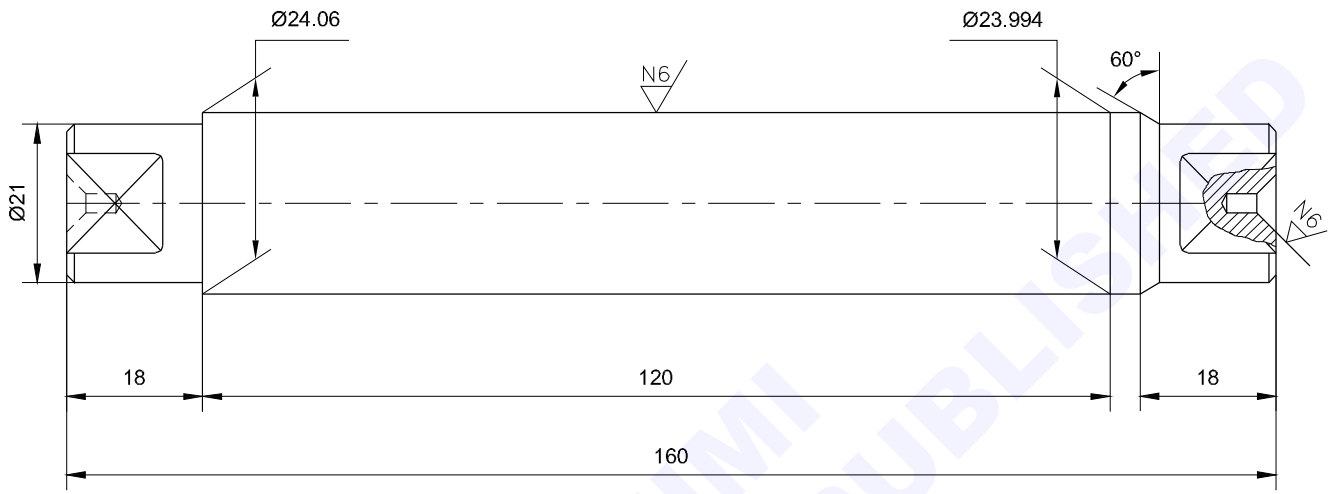


LINEAR DIMENSIONS OF THE BORE WITHIN LIMITS

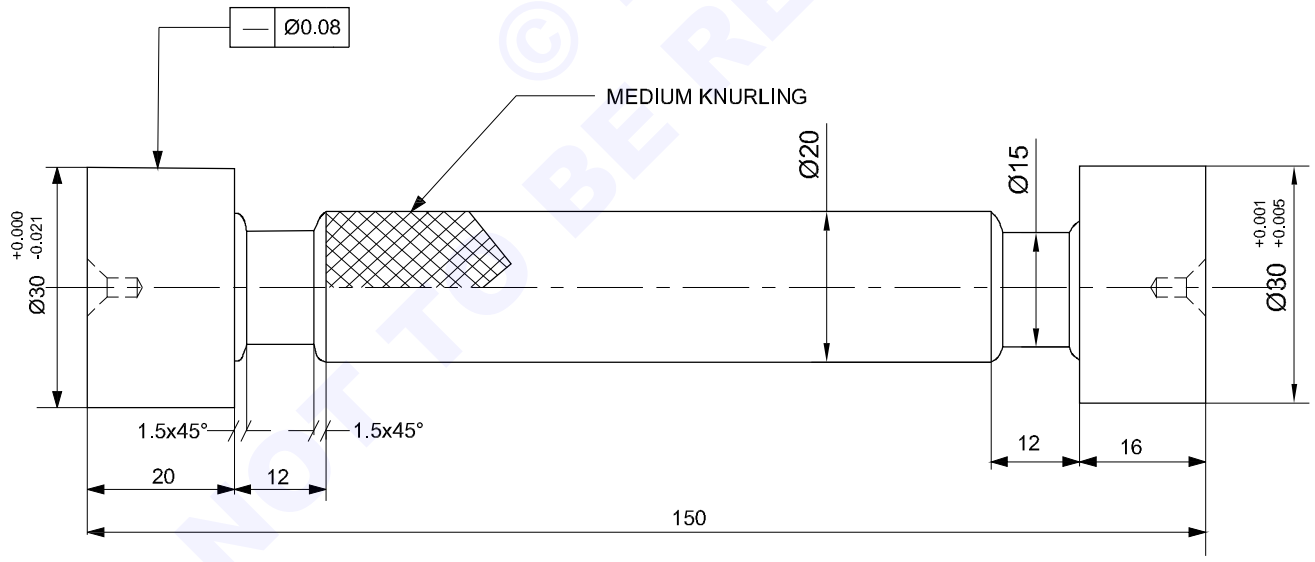
Make mandrel & plug gauge

Objectives: At the end of this exercise you shall be able to
- **use of carbide tip tools**
- **turn with an accuracy of 0.02 mm**
- **use of plain ring gauge to check dia of plug gauge.**

TASK-1



TASK-2



ALL DIMENSIONS ARE IN MM

1	Ø32 - 155	-	C40	-	TASK-2	-
1	Ø25 - 170	-	15 Ni 16 Cr5	-	TASK-1	2.1.94
NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.

SCALE NTS	**PREPARING MANDREL & PLUG GAUGE**	DEVIATIONS ±0.02	TIME
		CODE NO. TU20N2194E1	

Job Sequence

TASK 1 : **Making of Mandrel**

- Check the size of the raw material.
- Hold the job in a four jaw chuck and true it.
- Set the spindle r.p.m and face one end.
- Centre drill the end.
- Reverse and true the job.
- Face to length 160 mm and centre drill.
- Crown centre holes using the special centre drill.
- Dismount the four jaw chuck from the lathe spindle and mount the driving plate on the spindle nose.
- Clamp the job on a suitable lathe carrier and support the work between centres.
- Set the spindle r.p.m. and rough turn diameter to 24.16 mm. to maximum possible length.
- Turn step Ø 21 for 18 mm length and chamfer 2 x 45°.
- Reverse the job and turn Ø 21 mm for 18 mm length and chamfer 2 x 45°.
- Set the form tool to 60° and turn taper on step dia. as shown in the sketch.
- Check the size with an outside mircometer.
- Set the taper turning attachment for the small taper.
- Adjust in such a way that 24 mm diameter at the centre of the mandrel is achieved, with an acceracy of ± 0.02 mm.

– – – – – – –

TASK 2 : **Making of a plug gauge**

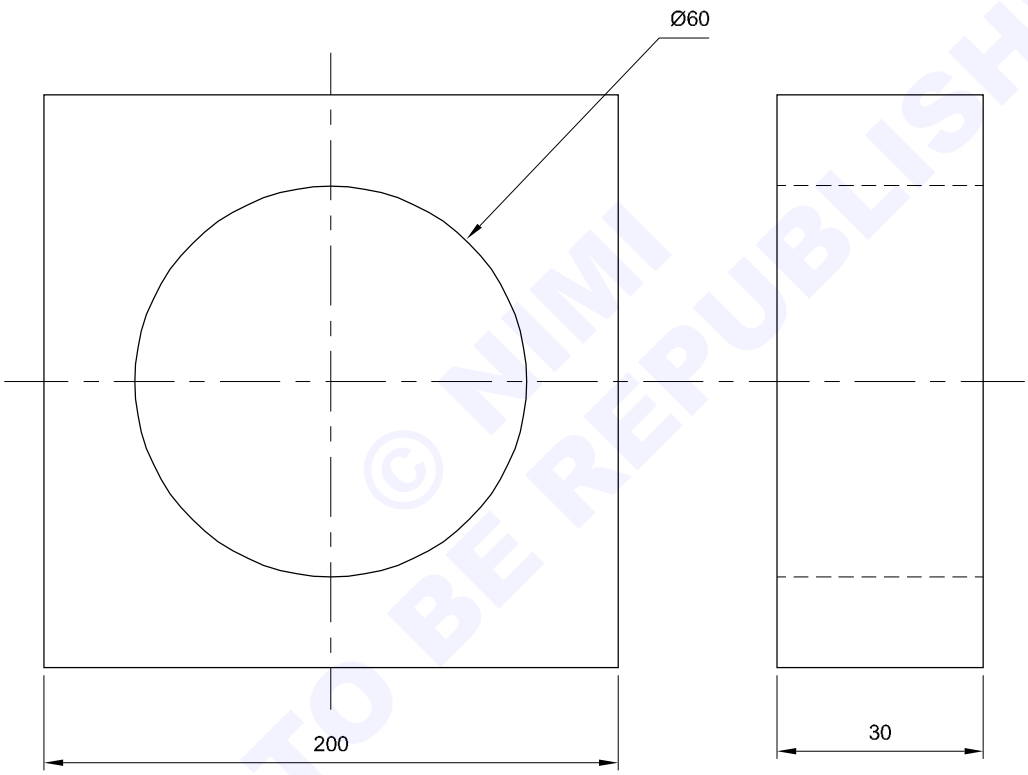
- Check the size of the raw material.
- Hold the job in four jaw chuck & true it.
- Face the end.
- Centre drill it.
- Reverse the job & true it.
- Face the end to maintain 150 mm length.
- Dismantle the chuck from the spindle & mount the driving plate. Spindle adopter & line centre to turn the job between centres.
- Set the job in between centres.
- Turn $Ø30^{+0.001}_{+0.005}$ for a 32 mm length. (For 'No Go' gauge)
- Check dia with vernier micrometer.
- Make a groove to Ø 20 mm for a width to suit a R.H turning tool leaving 16 mm from the face.
- Set the R.H turning tool & turn to dia $20^{-0.02}$ for a length as per drawing.
- Make a groove/under cut with corner rounded as per drawing. (12 mm width at two corners)
- Set the knurling tool & form knurling and maintain a dia 20 mm.
- Reverse the job & turn $Ø30^{+0.000}_{-0.021}$ remaining length (20 mm) (For Go - to the Gauge)
- Deburr all sharp corners.
- Check the dimensions as per the drawing.

– – – – – – –

Setting & turning operation involving face plate & angle plate

Objectives: At the end of this exercise you shall be able to
- **setting the square job on face plate**
- **truing the job using scribing block**
- **drill and boring**

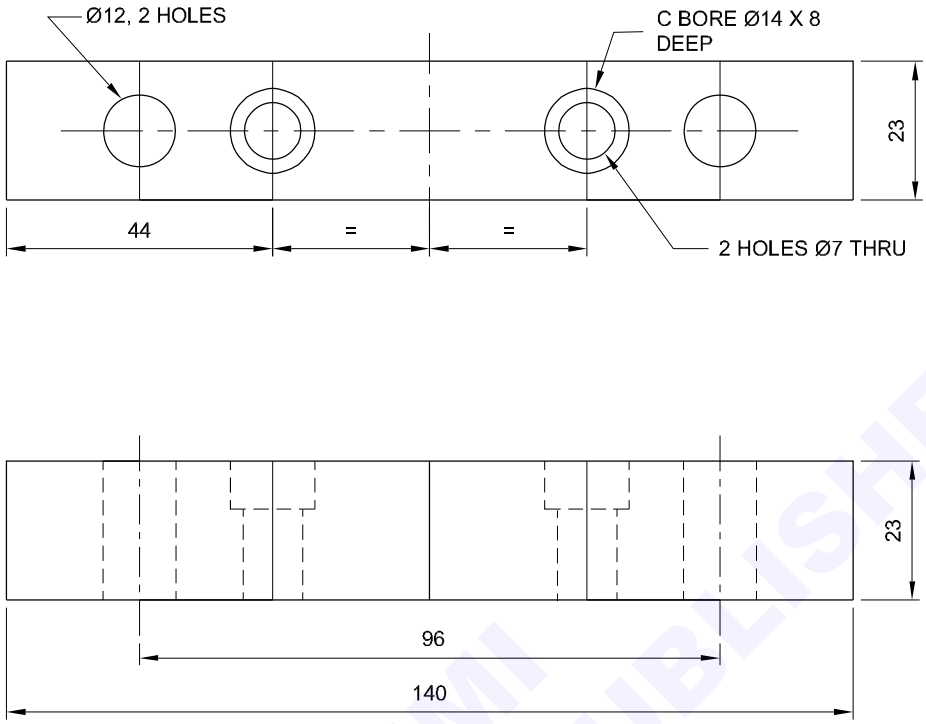
TASK-1



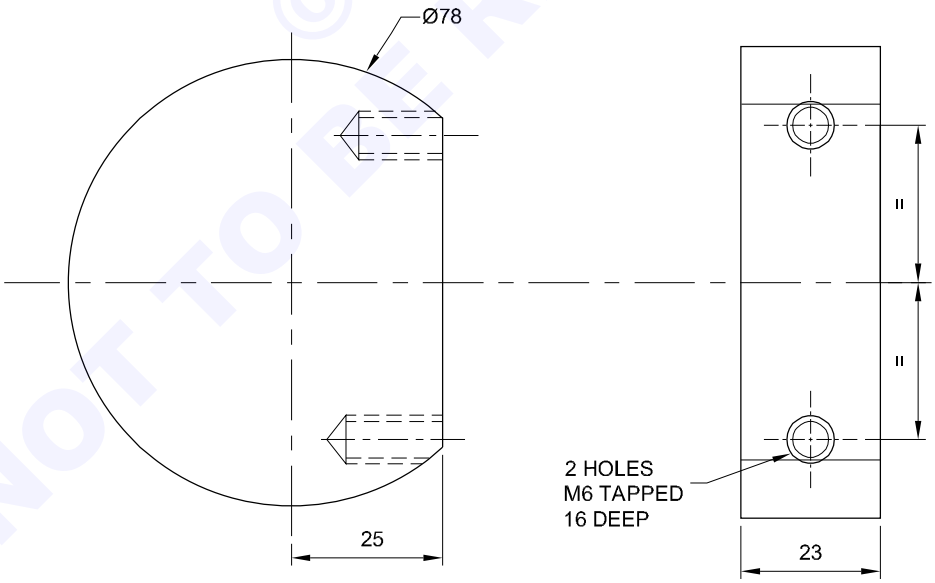
1	Ø80-25	-	Fe 310	-	TASK-2B	-
1	SQ 25-145	-	Fe 310	-	TASK-2A	-
1	200ISF30x200	PRE MACHINE	Fe 310	-	TASK-1	2-2-95
NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.

SCALE NTS	**SETTING & TURNING OPERATION INVOLVING FACE PLATE & ANGLE PLATE**	TOLERANCE. ±0.02mm	TIME:
		CODE NO. TU20N2295E1	

TASK-2A



TASK-2B



TU20N2295E2

Job Sequence

TASK 1: **Boring operation using face plate**

- Check material size as per drawing.
- Apply marking media.
- Mark the centre using steel rule and scriber.
- Punch the centre using centre punch and hammer.
- Draw the circles of Ø 60 mm using the divider.
- Punch the identification mark using dot punch 60° and hammer.
- Clamp the workpiece on slotted face plate using bolt and clamp.
- True the workpiece as per the punch mark using scribing block.
- Make the centre drill and pilot hole of 8 mm.
- Enlarge the hole Ø 8 mm to Ø 20 mm, Ø 30 mm, Ø 40 mm and Ø 50 mm drills.
- Set the boring tool to correct centre height.
- Bore the drilled hole to Ø 60 mm through.
- Deburr the sharp corner of workpiece.
- Check the bore size with a vernier caliper.

– – – – – – –

TASK 2A : **Fixture for clamping in face plate**

Check the raw material size.

- Hold the raw material about 5 mm outside the chuck and run true.
- Set the facing tool and face one end of job.
- Reverse and hold the workpiece and face it to maintain a total length 140 mm.
- Remove and hold workpiece and face it to maintain a total length 140 mm.
- Face the job 4 sides squareness right angles and true it.
- Make the drill hole and counterbore.
- Centre punch the centre.
- Drill through hole Ø 7 mm.
- Drill through hole 12 mm 2 holes.
- Mount and fix the 14 mm counterbore tool on the drilling machine spindle.
- Counterbore the holes to a depth 8 mm.
- Check the workpiece to verify the correctness of dimensions.
- Check the raw material size.

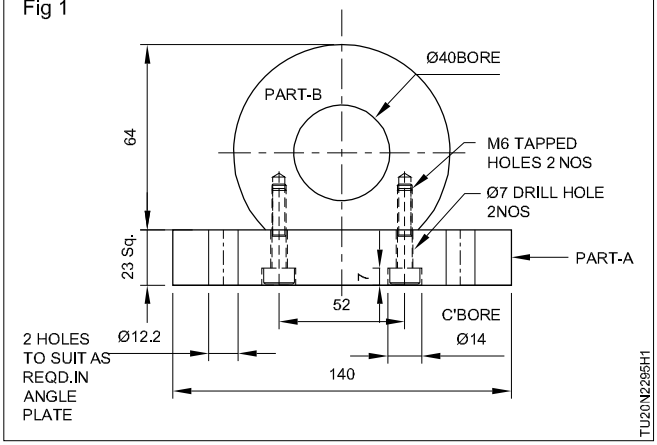
TASK 2B : **Fixture for guiding the work**

- Hold the raw material about 18 mm outside the chuck and run true.
- Set the cutting tool and face one end of job and turn Ø 78 mm x 15 mm length.
- Reverse and hold work and face it to maintain a total length 23 mm and turn Ø 78 mm to remaining layer. Setting of centre axis of spindle.
- Remove and hold the join in 4 jaw chuck and setting to centre axis of spindle.
- Face 14 mm depth 4 or 5 number or cut. Maintain face end to diameter maximum point height 70 mm.
- Mount and face the side on the Task 2A drilled position equal to length 2 sides.
- Clamp the Task 2A and Task 2B job. (Use 'C' clamp)
- Drill a 5 mm 2 hole depth 15 mm guide through 7 mm drill hole for part A.
- Dismount - Task 2A and Task 2B job.
- Cut M6 internal thread in the two Ø 5 mm drilled holes.
- Deburr the sharp corners of the workpiece.
- Fix the Task 2A and Task 2B using M6 x 30 mm length allen screw.
- Check the workpiece to verify the correctness of dimensions.

– – – – – – –

Turning job using face plate & angle plate (Fig 1)

Fig 1



- Place a wooden board on the lathe bed below the spindle, to prevent any damage to the guide ways.
- Place the angle plate on the wooden board.
- Clean the face plate and angle plate throughly.
- Apply a thin film of oil on the mating area of spindle and face plate.
- Clean the angle plate.
- Locate the angle plate on the face plate and clamp it properly with clamps.
- Clamp the workpiece with sufficient bolts and clamps effectively.
- Mount the face plate on lathe spindle properly.
- Set the workpiece true to centre.
- Clamp the counter weight equal for balancing the rotation of spindle.

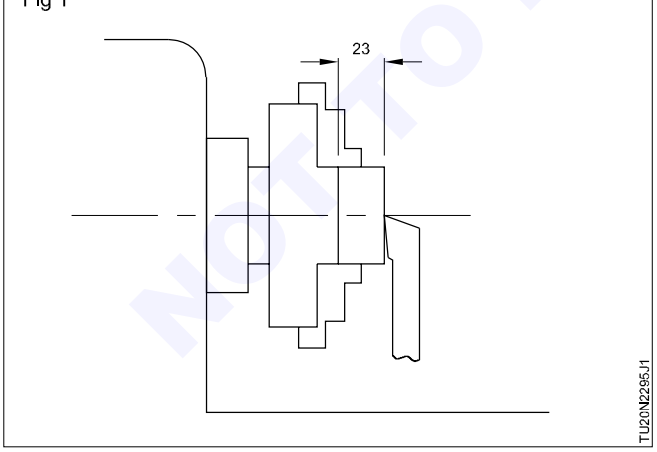
Skill Sequence

Bearing block

Objectives : This shall help you to
- **mark on round job for flat face**
- **set the round job for facing in four jaw chuck.**

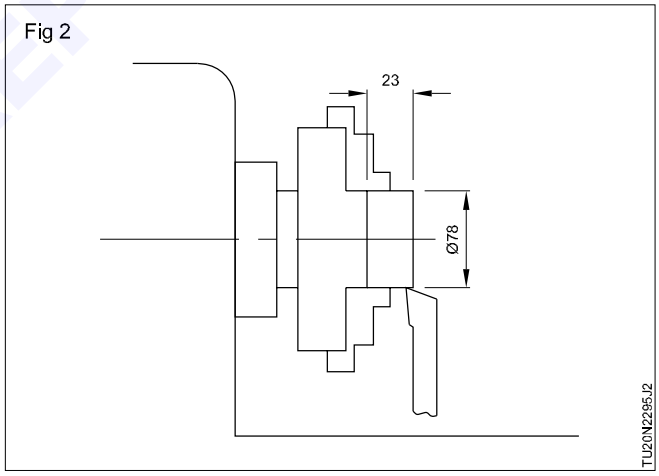
- Check the material for its size.
- Hold the job in four jaw chuck and true it.
- Face one end of the job.
- Reverse and hold the job in chuck and true it.
- Face of other end total length maintain 23 mm. (Fig 1)

Fig 1



- Turn ∅78 mm and maximum length.
- Reverse the job and hold in four jaw chuck.
- Turn ∅78 mm remaining length. (Fig 2)

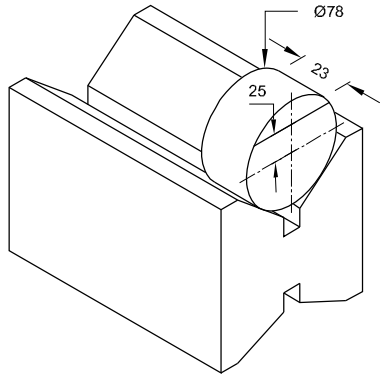
Fig 2



Use V block with clamp for hold the round rod rigidly while marking.

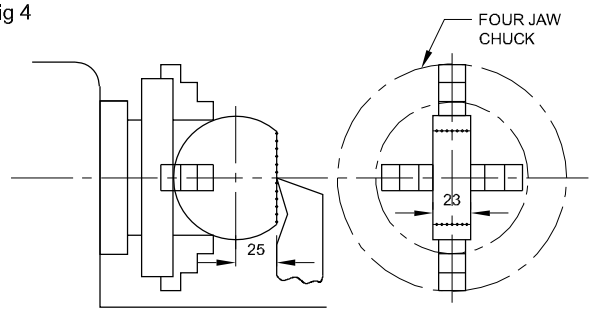
- Scribe the centre line on the face of round block using a vernier height gauge.
- Scribe the 25 mm distance above from centre line and continue on both sides (Fig 3)
- Hold the job against 23 mm thickness (Fig 4) in four jaw chuck.
- Check against the scribed line, using surface gauge for trueness.

Fig 3



TU20N2295J3

Fig 4

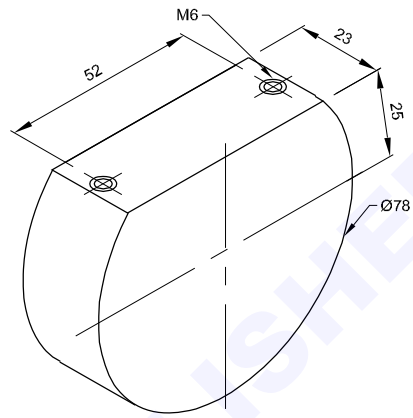


VARIOUS TYPE OF TOOL SUITABLE FOR TURNING

TU20N2295J4

- Move the cross slide against the marked line for parallelism using surface gauge.
- Turn to finish the flat surface on the curved face job as per drawing.
- Mark the tapped holes location as per the drawing.
- Drill ∅5 mm holes as per the drawing.
- Tap the holes with M6 tap set. (Fig 5)

Fig 5



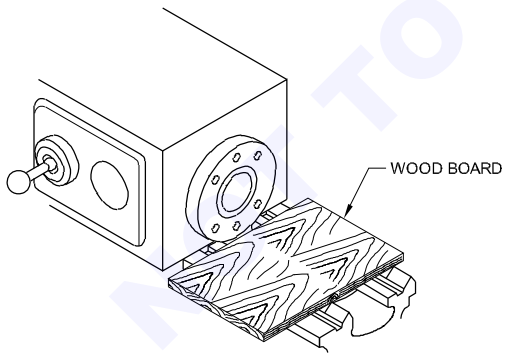
TU20N2295J5

Setting of face plate

Objectives : This shall help you to
- **dismount of chuck from lathe spindle**
- **mount of face plate.**

- Switch OFF the motor.
- Place a wooden board or cradle on the lathe bed, to prevent damage to slideways (Fig 1)

Fig 1



BOARD TO PROTECT SLIDEWAYS

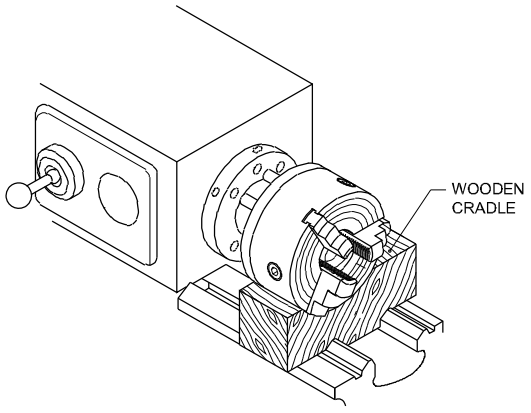
TU20N2295X1

- Dismantle the workholding device(Three jaw chuck) already assembled in the spindle.(Fig 2)

In addition to protecting the bed slideways it makes fitting the chuck easier and safer.

- Ensure that the correct face plate for the lathe and for the job in hand before attempt to mount the face plate.

Fig 2



CHUCK SUPPORTED ON WOODEN CRADLE

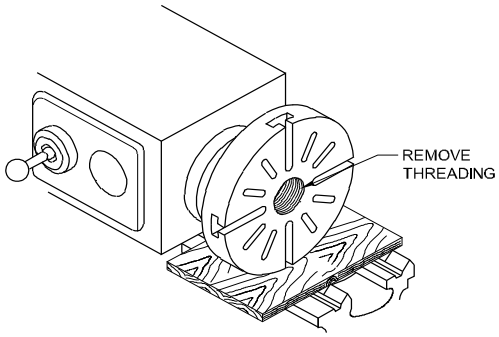
TU20N2295X2

- Seek assistance always when mounting large and heavy chuck (or) face plate.
- Clean the mating parts of the face plate and lathe spindle as otherwise, the dirt on these surface could result in run out of true.

Do not use power to mount a face plate on spindle noses.

- Place the face plate on the wooden board. Slide it close to the spindle nose. (Fig 3)

Fig 3



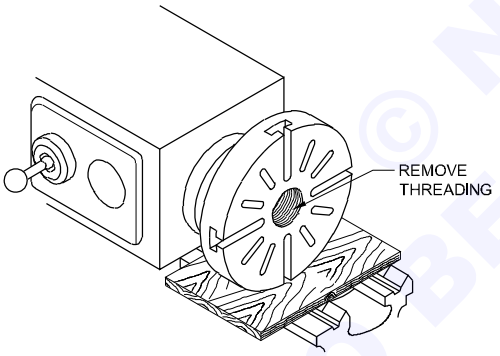
- Turn the spindle by hand until the key on the spindle nose lines up with the keyway in the face plate.
- Ensure the face plate mounted for its trueness for mounting the workpiece.

Setting of angle plate and balancing weight

Objectives : This shall help you to
- **set the angle plate on face plate**
- **clamp the counterweight.**

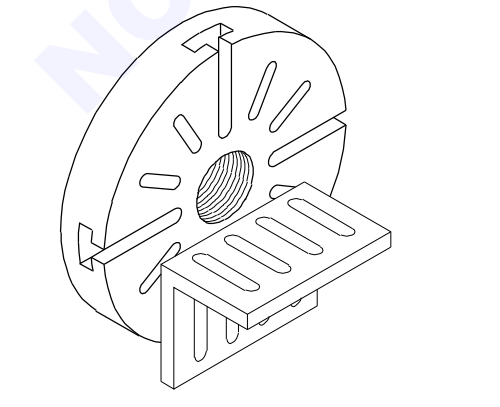
- Place a wooden board on the lathe bed to prevent damage to slide ways, due to the weight of face plate.
- Clean the face plate and angle plate seating surface. (Fig 1)

Fig 1



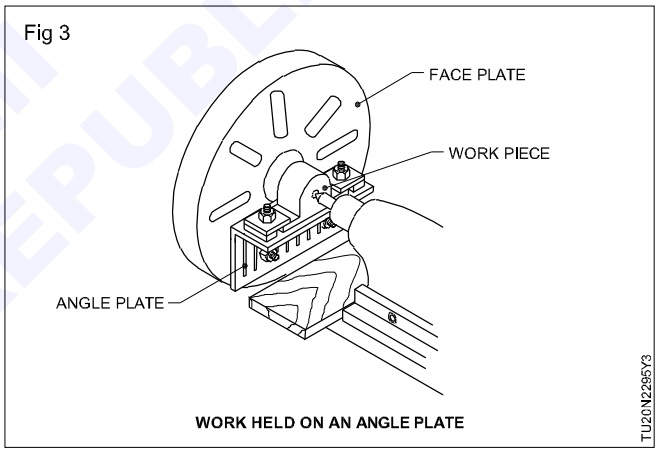
- Clamp the angle plate to position approximately to the spindle axis. (Fig 2)

Fig 2



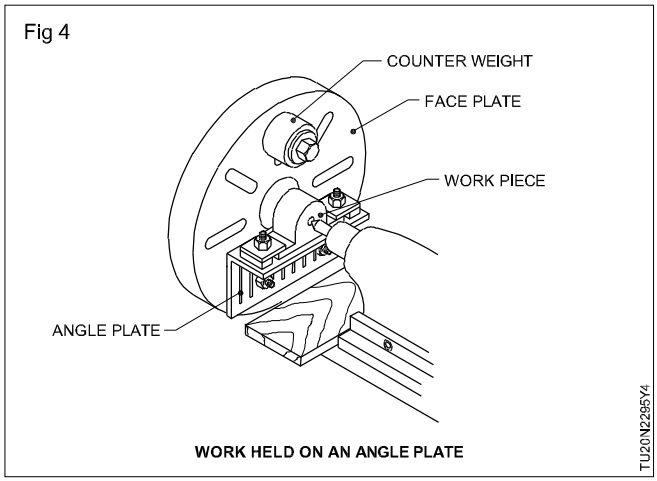
- Mount the workpiece on the angle plate (Fig 3)

Fig 3



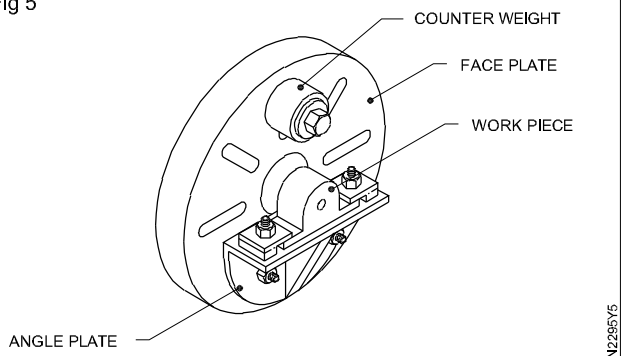
- Adjust the workpiece centre to spindle axis.
- Tighten the clamping bolts and nuts of angle plate and workpiece.
- Locate and position the counterweight above the workpiece in the slot of face plate (Fig 4)

Fig 4



- Rotate the lathe spindle by hand to check the face plate is balanced with counterweight.
- Ensure the counterweight is enough to balance the workpiece.
- Observe the counterweight and workpiece remain in line (Fig 5)

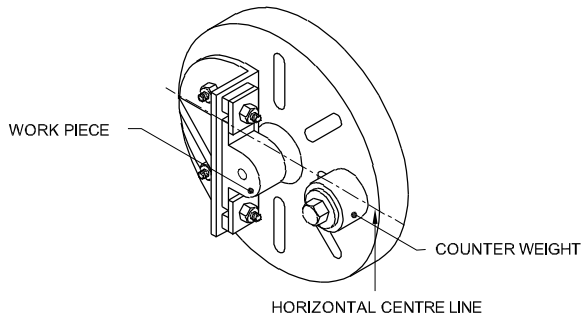
Fig 5



BALANCING WORK ON A FACE PLATE

- Check the counterweight and workpiece remain in horizontal line (Fig 6)

Fig 6



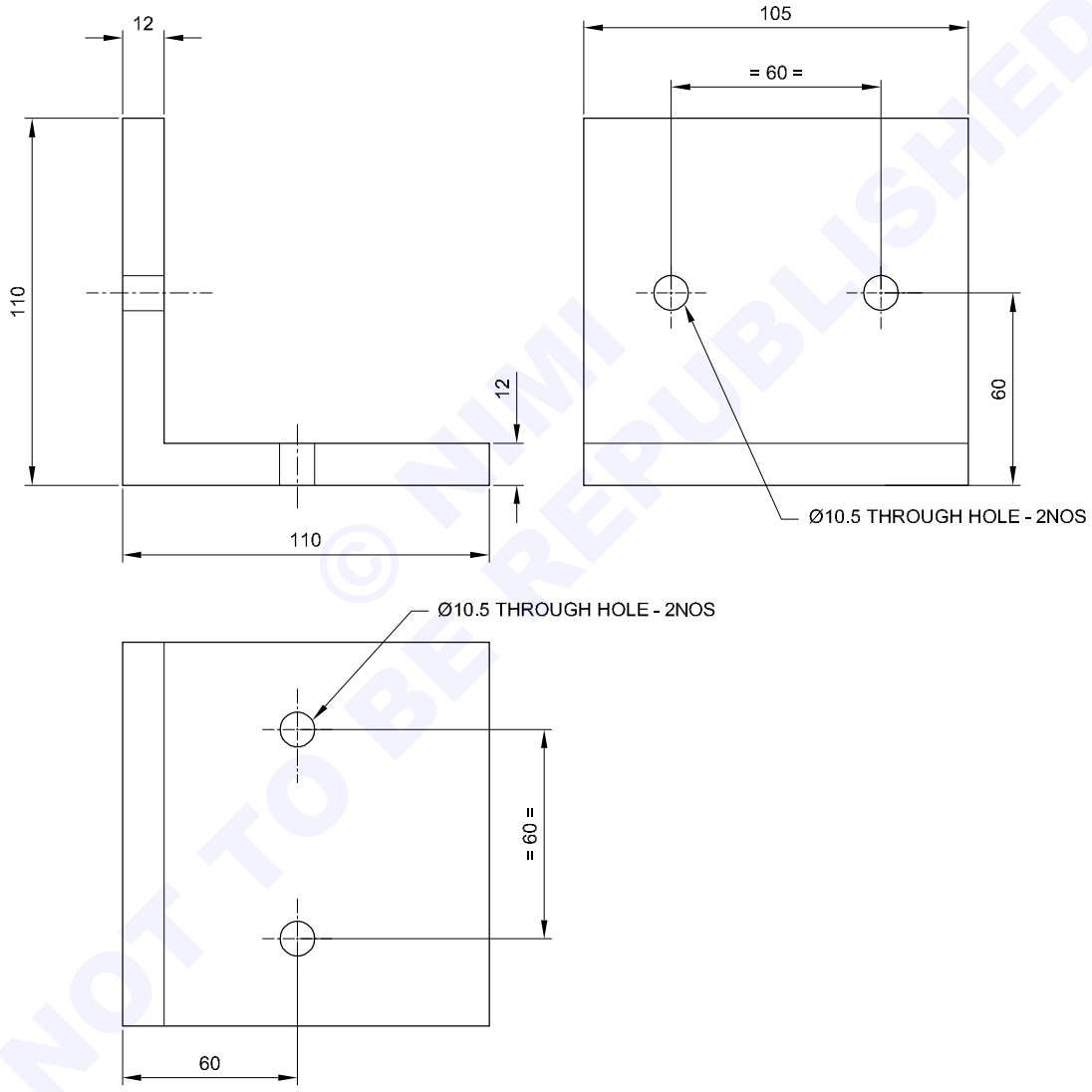
CHECKING BALANCE OF FACE PLATE

- Rotate by hand to ensure the face plate is balanced by counterweight.
- Repeat to continue till the face plate is balanced by increasing or decreasing the weight of the counter weight to set the correct position.
- Tighten all the clamping bolts and nuts rigidly before operating the lathe.

Making angle plate using face plate

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

- **turn two faces of an angle plate**
- **check squareness of angle plate**
- **hold odd jobs in a face plate.**

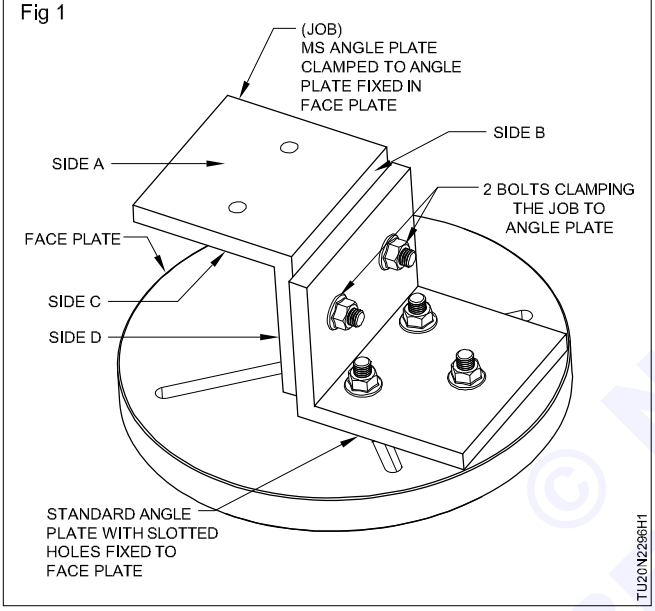


1	MS ANGLE ISA 110x110x10-110	-	-	-	-	2.2.96
NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.
SCALE NTS	**MAKE ANGLE PLATE USING FACE PLATE**				TOLERANCE.	TIME:
					CODE NO. TU20N2296E1	

Job Sequence

- Check the raw material for its size.
- File the size 108x108mm. (C and D side) Fig 1.
- Mark the given dimension for drilling location as per size.
- Clamp the job on drilling machine an keeping parallel bocks under the job.
- Using a cetre drill locate the hole position and drill upto 1mm depth.
- Without changing the position of the job remove the centre drill and fix ∅10.5mm drill and drill a through hole size A. (Fig 1)
- Similarly drill the other B size (Fig 1)

Fig 1



- Place a wooden board on the lathe bed to prevent damage to the lathe bed ways, while fixing the face plate on to the lathe spindle.
- After fixing the face plate, clamp the job to the standard angle plate already screwed to the face plate. (Fig 1)
- Balance the angle plate by clamping suitable counter weight.
- Affter ensuring that the angle plate is firmly clamped to the face plate run the lathe spindle and check for true running.
- Set the facing tool and take a rough facing cut.
- Continue the facing by giving less feed till entire face is machined to the required accuracy.
- After completion of one side machining dismatle from the face plate and clamp for machining the other face.
- Face the other side and check the flatness and squareness of the workpiece.

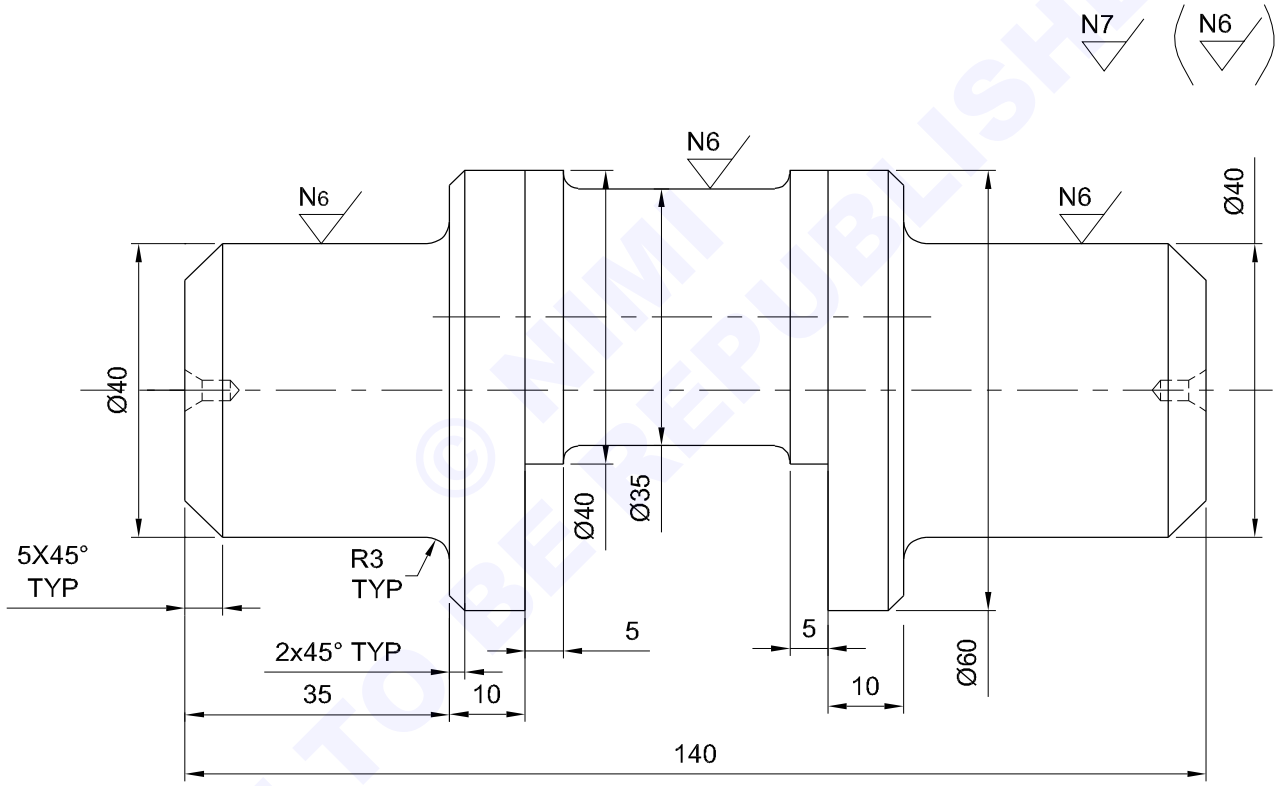
Note:

- **The angle plate is an useful fixture used for checking squareness of a component and for marking purposes.**

Holding and truing of crank shaft single throw

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

- **mark eccentric axis**
- **turn eccentric diameter**
- **check eccentric dia run out/off set.**



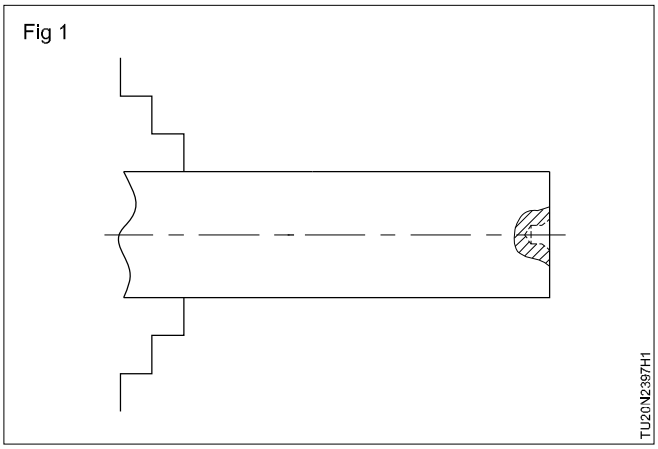
1	Ø63x145	-	C14 IS:2073	-	-	2.3.97
NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.

SCALE NTS	HOLDING AND TRUING OF CRANK SHAFT (SINGLE THROW)	TOLERANCE.	TIME:
		CODE NO. TU20N2397E1	

Job Sequence

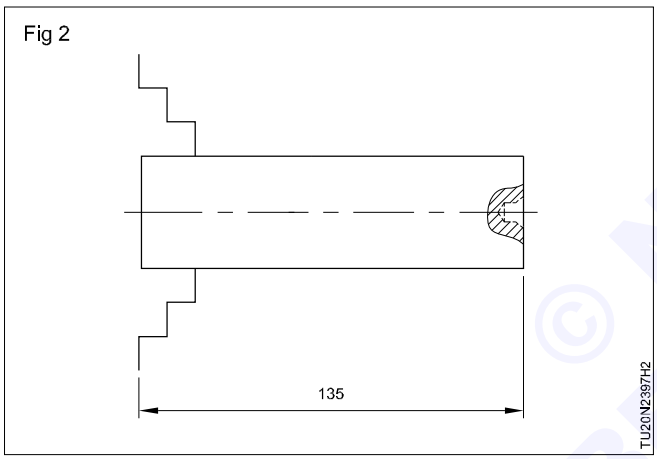
- Check raw material as per drawing.
- Hold in chuck face and centre drill. (Fig 1)

Fig 1



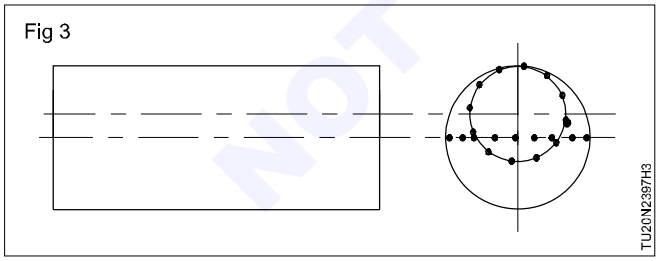
- Turn to Ø 61 mm to 70 mm length.
- Remove, reverse and hold. Face to length 135 and centre drill. (Fig 2)

Fig 2



- Turn to Ø 61 mm to unturned portion.
- Using 'V' block and height gauge mark eccentric axis on both faces. Dot punch eccentric centre both sides. From eccentric centre draw circle Ø 40 both faces and dot punch. (Fig 3)

Fig 3



- Hold job in 4 jaw chuck align eccentric centre to m/c axis and centre drill. (Fig 4)
- Similarly on the other face also.
- Hold job between centres of eccentric axis mark eccentric portion for turning. (Fig 5)
- Using parting to remove excess material to Ø 42. (Fig 6)

Fig 4

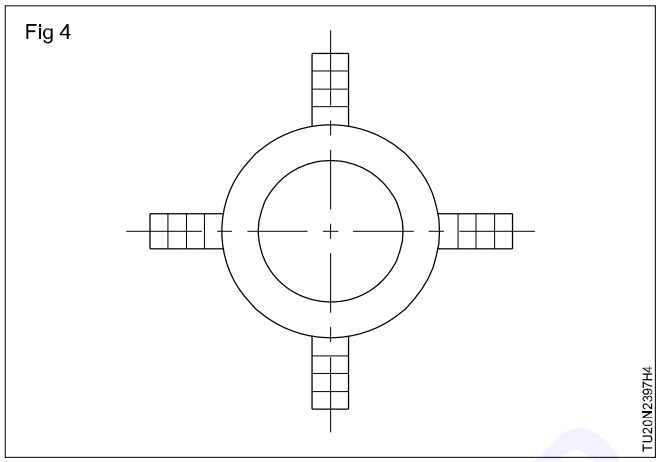


Fig 5

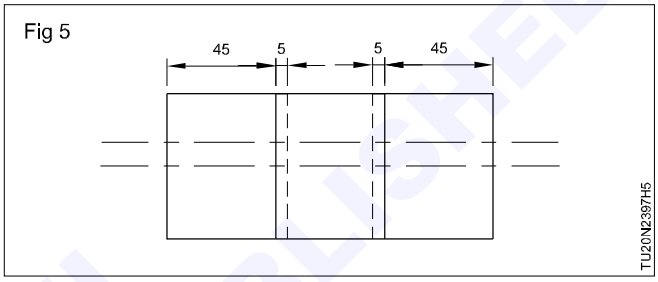
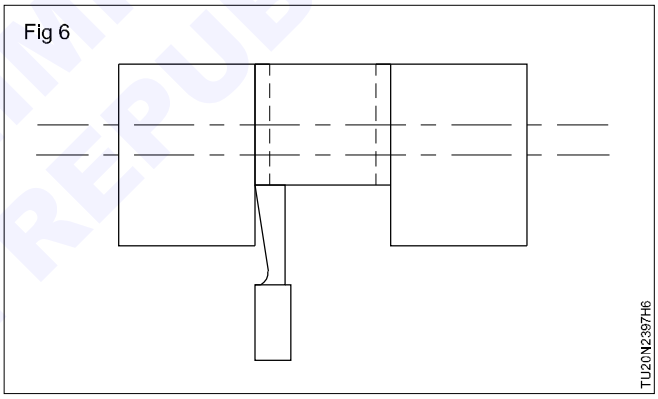
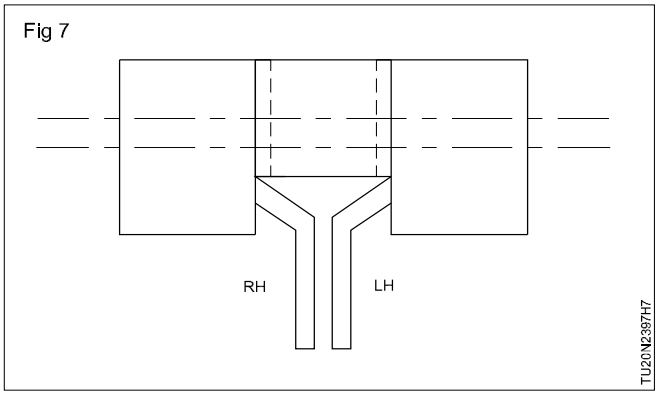


Fig 6



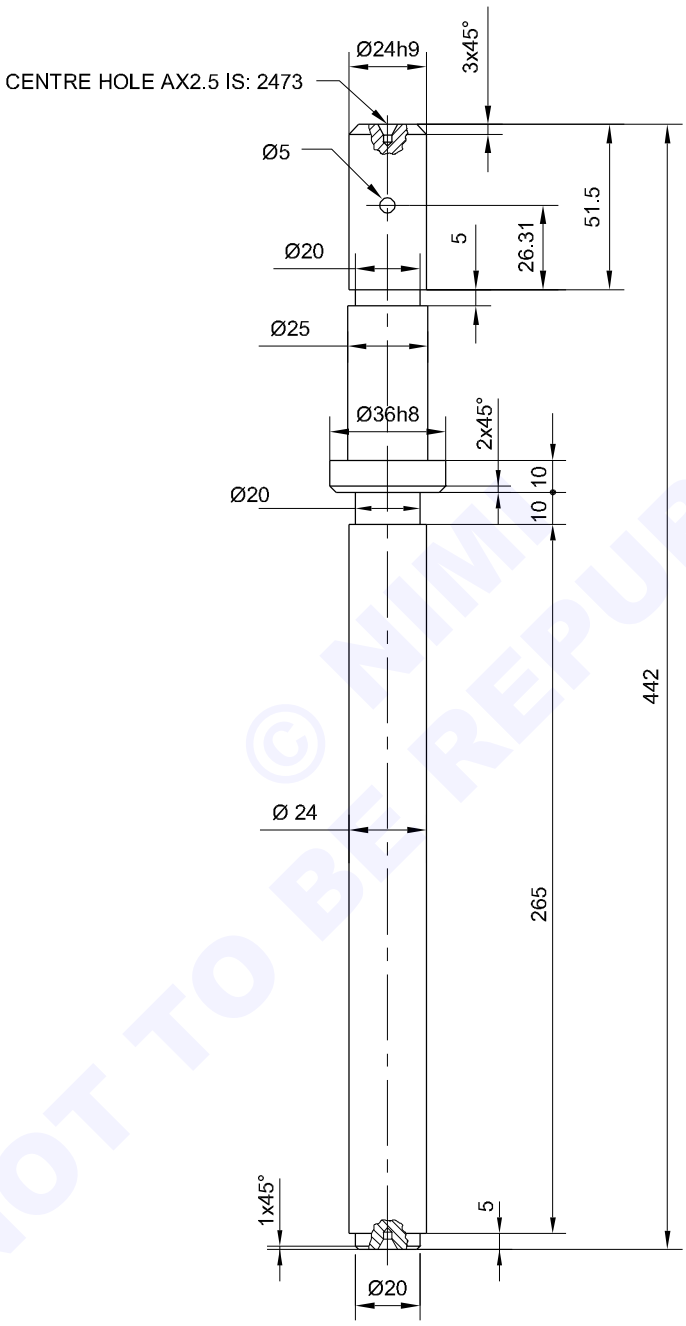
- Using LH & RH turning tool with R2 nose rod turn to Ø 40 to 45 mm length and Ø 30 to 35 mm length as per drawing. (Fig 7)
- Remove hold job between centre. Rough and finish turn to Ø 60 x 45 mm length Ø 40 to 35 mm length. Form radius chamfer as per drawing.
- Place 'V' blocks using dial test indicator, check eccentric throw dist of the job. Note the offset deviation.

Fig 7



Turning of long shaft (using steady rest)

Objective: At the end of this exercise you shall be able to
- **turn long jobs supported with the travelling and fixed steadies.**



1	Ø40 - 460		Fe 310 -O			2.3.98
NO.OFF	STOCK SIZE	SEMI PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.
SCALE 1:2	**TURNING OF LONG SHAFT (USING STEADY REST)**				DEVIATIONS IS:2102 (M)	TIME
					CODE NO. TU20N2398E1	

Job Sequence

- Hold the job in a four-jaw independent chuck, true and face both ends to a length of 442 mm and centre drill on both ends.
- Remove the job from the chuck and remove the chuck from the lathe spindle.
- Hold the job in between centres with the help of a bent tail carrier.
- Turn the job for the maximum possible length diameter of 24 h8.
- Reverse job and finish full length to a diameter of 36 h8.
- Turn the job to a length of 152 mm to Ø 25 mm.
- Turn the job to a length of 56.5 mm to a diameter of 24 h9.
- Undercut 2 x 5 mm as per drawing.
- Chamfer 3 x 45° as per drawing.
- Reverse the job and turn Ø 24 mm to a length of 280 mm with the support of the follower steady.
- Support the work with a fixed steady on Ø 24 mm.
- Undercut 2 mm x 10 mm as per drawing.
- Chamfer 2 x 45 mm from the collar Ø 36 h8.

Safety precautions

- Use soft packing pieces for holding the job in between the job and the carrier to avoid spoilage of the finished surface.
- Ensure that the fingers of the follower rest is not hard pressed against the job, and it is frequently lubricated.

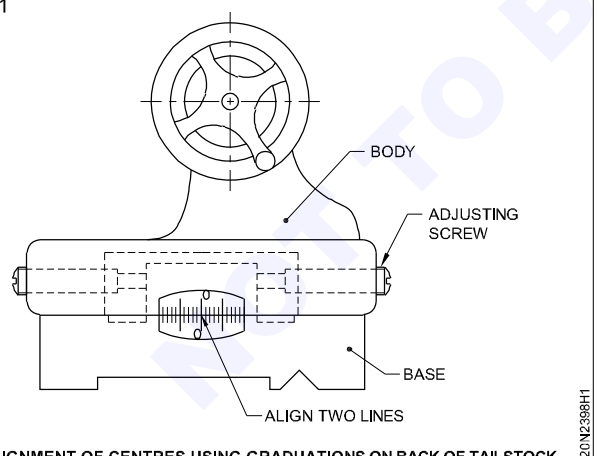
Skill Sequence

Aligning lathe centres by different methods

Objectives: This shall help you to
- **align the marked centre lines visually on the rear end of the tailstock**
- **align visual points of the live and dead centres**
- **take trial cuts at each end of the work and check the finished diameter with a micrometer and align the lathe centres**
- **use a parallel test bar and a dial indicator to align the lathe centres**
- **use a micro-set adjustable centre and align the lathe centres.**

Using tailstock graduations. (Fig 1)

Fig 1



ALIGNMENT OF CENTRES USING GRADUATIONS ON BACK OF TAILSTOCK

Loosen the clamp holding the tailstock to the bed.

Locate the adjusting screws on the tailstock body.

Slacken one of the adjusting screws several turns.

Tighten the other screw slowly.

Observe the direction of movement of the tailstock centre. Continue adjusting the screws until the line on the tailstock body coincides with the line on the base plate.

Tighten the loose adjusting screw to secure the upper part of the tailstock in position.

Visual alignment.

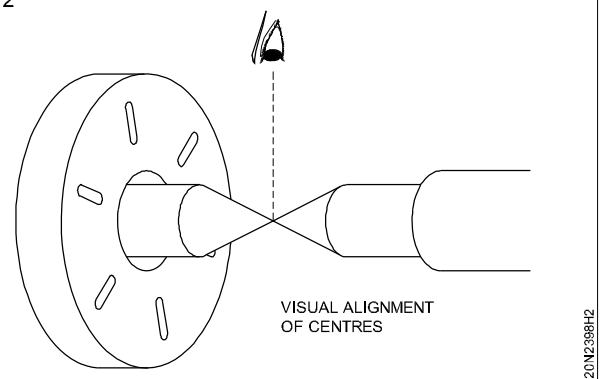
Loosen the tailstock clamp lever or nut.

Turn the tailstock hand wheel until the tailstock spindle barrel extends about 25 mm.

Slide the tailstock to the left until the dead centre is approximately 10 mm from the live centre.

Turn the tailstock hand wheel until the points of the centres almost touch. (Fig 2)

Fig 2



Place a sheet of white paper on the lathe bed under the centres.

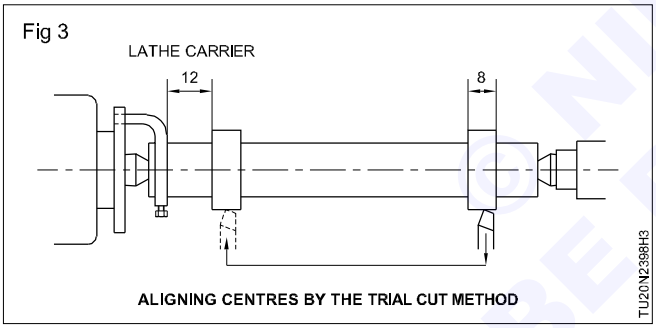
Look down at the top of the centres and check the alignment of the lathe centre points.

Adjust the tailstock by means of the two adjusting screws until the points of both centres are in line.

Tighten the loose adjusting screw to the upper part of the tailstock in this position.

Using the trial cut method. (Fig 3)

Fig 3



ALIGNING CENTRES BY THE TRIAL CUT METHOD

Mount the workpiece between the centres of the lathe.

Take a light cut at the tailstock end of the work, but deep enough to produce a true cylindrical surface. The cut should be about 8 mm long.

Stop the lathe and note the reading on the cross-slide graduated collar.

Back the cutting tool away from the work using the cross slide handle.

Move the carriage until the cutting tool is about 25 mm from the lathe carrier.

Make sure that the lathe carrier does not strike the top slide.

Turn the cross-slide handle slowly until the graduated collar is at the same setting as it was for the tailstock end.

Machine this end for a length of about 12 mm.

Stop the lathe and measure the diameters of both the turned surfaces with a micrometer.

The lathe centres are in alignment if the diameters of the two machined sections are the same.

If the diameters are different, the tailstock must be adjusted by one half the difference between these two diameters. This can be done by using a feeler guage of the correct thickness between the tool point and the workpiece.

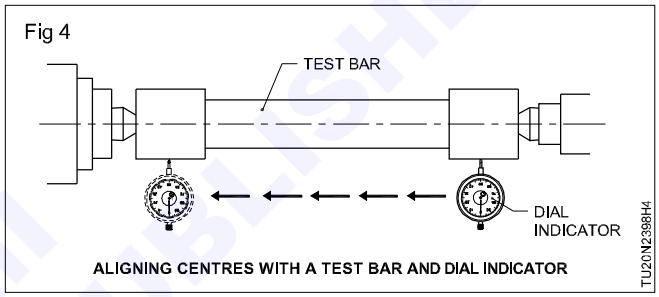
Adjust towards the cutting tool, if the diameter at the tailstock end is larger.

Take another light cut from both machined diameters using the same graduated collar setting for each out. Measure the diameter.

Continue adjusting the tailstock and taking light trial cuts until both diameters are the same.

Using a test bar and a dial indicator (Fig 4)

Fig 4



ALIGNING CENTRES WITH A TEST BAR AND DIAL INDICATOR

Clean the centres of the lathe and the centre drilled holes in the test bar.

Mount the test bar between the centres and tighten the tailstock spindle clamp.

Mount a dial indicator in the tool post or on the lathe carriage.

The contact point should be the centre of the bar and the indicator plunger should be in horizontal position.

Adjust the cross-slide so that the indicator needle registers about one half of a revolution on the diameter at the tailstock end.

Move the carriage to the left by hand until the indicator register on the diameter at the headstock end. Note the indicator reading.

If the readings are not the same, move the carriage until the indicator again registers on the tailstock end of the diameter.

Loosen the tailstock clamp nut.

Using the tailstock screws, move the tailstock in the proper direction. The amount of movement should equal the difference between the indicator readings.

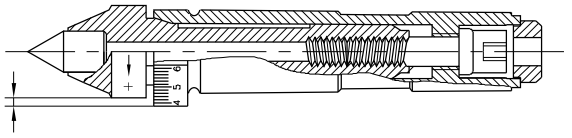
Tighten the adjusting screw to lock the upper part of the tailstock in place.

Tighten the tailstock clamp nut and recheck to make sure that the test bar still fits snugly between the centres.

Repeat these steps until the indicator readings at the two ends are the same.

Using micro-set adjustable centre (Fig 5)

Fig 5



MICRO-SET ADJUSTABLE CENTRE

TU20N2398H5

A micro-set adjustable centre fitted into the tailstock spindle provides a fast and accurate method of aligning lathe centres.

Some of these centres contain an eccentric; others contain a dovetail which permits slight adjustment of the centre itself to correct the alignment.

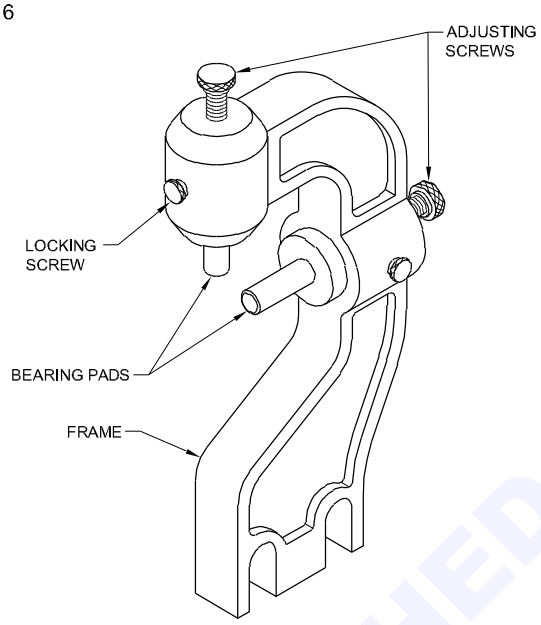
Use either the trial cut or test bar and dial indicator method to determine the accuracy of the lathe centre alignment.

Adjsut the micro-set adjustable centre by the amount that the centres are out of alignment.

A follower steady is fixed to the saddle of the lathe. As it follows the tool it gives support just behind the cutting point. In case of the follower steady the support is continuous to the entire length of cutting. (Fig 6)

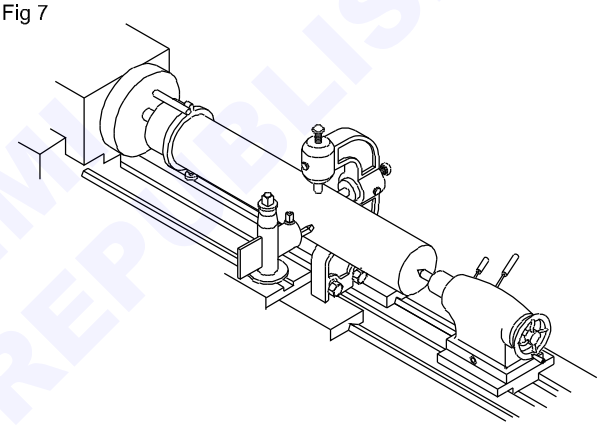
It has usually two pads. One pad is located opposite to the cutting tool and the other pad bears the top of the workpiece to prevent it from springing up. Fig 7 shows the travelling steady rest in position.

Fig 6



TU20N2398H6

Fig 7

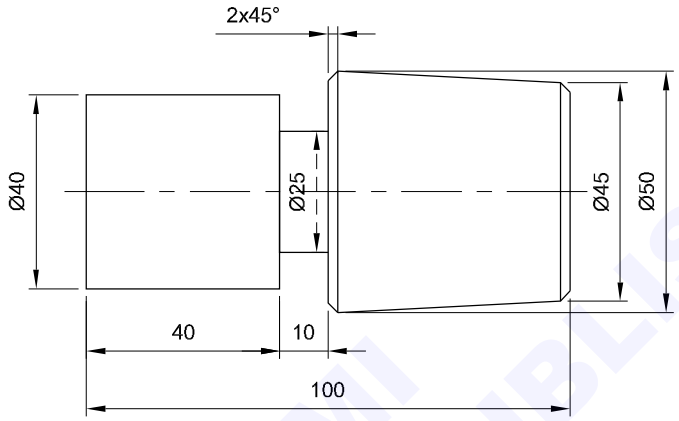


TU20N2398H7

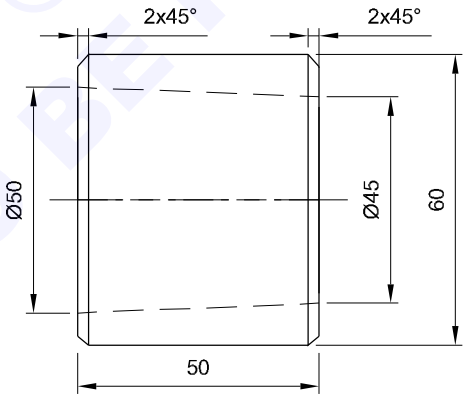
Use of attachments on lathe for different operations

Objectives: At the end of this exercise you shall be able to
- **taper turning using taper turning attachment**
- **external taper thread by taper turning attachment**
- **internal taper thread by taper turning attachment.**

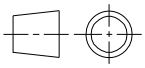
TASK-1



TASK-2



1	Ø63x55	-	Fe 310	-	TASK-2	2.3.99
1	Ø55-105	-	Fe 310	-	TASK-1	2.3.99
NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.

SCALE NTS	**TAPER TURNING (BY TURNING ATTACHEMENT)**	TOLERANCE.	TIME:
		CODE NO. TU20N2399E1	

Job Sequence

TASK 1 : **External taper thread**

- Check the raw material for its size.
- Hold the job in a four jaw chuck keeping 10 mm outside the chuck and true it.
- Set the tool for facing.
- Face one end and turn Ø 50 mm to Ø 50 mm length.
- Turn to Ø 40 mm to the length of 50 mm.
- Reverse the job, hold in the chuck and true it.
- Face and maintain Ø 50 mm as per drawing.
- Turn the undercut Ø 25 x 10 mm length as per drawing.
- Taper calculation.

$$Tan\frac{\phi}{2}=\frac{D-d}{2l}$$

$$=\frac{50-45}{2\times 50}=\frac{5}{100}=0.05$$

- Set natural tan value = 2° 52'.
- Set the taper turning by taper turning attachment for the 2° 52' amount of taper.
- Turn taper by engaging the power feed and complete the taper turning.
- Remove burrs.

– – – – – –

TASK 2 : **Internal taper thread**

- Check the raw material for its size.
- Hold the job in a four jaw chuck and true it.
- Set the tool for facing.
- Face one end and turn Ø 60 mm for maximum length.
- Reverse the job, hold in the chuck and true it.
- Face and turn to maintain total length 50 mm and Ø 60 mm.
- Drill dia 10 mm through hole.
- Enlarge the through hole to Ø 25 mm.
- Fix the boring bar in the tool post and finish bore dia 40 mm.
- Taper calculation.

$$Tan\frac{\phi}{2}=\frac{D-d}{2l}$$

$$=\frac{50-45}{50\times 2}=\frac{5}{100}=0.05$$

- Set the natural tangent value 2° 52'.
- Set the taper turning attachment for the 2° 52' amount of taper.
- Taper bore by engaging the power feed for a total length.
- Complete the taper boring.
- Remove burrs.

– – – – – –

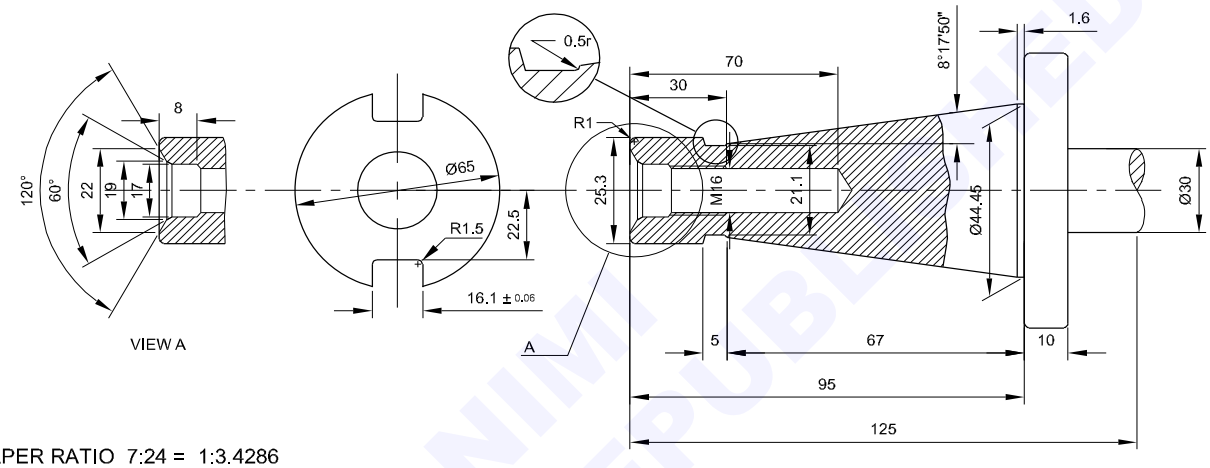
Turning standard stub arbor (ISO 40) With accessories collar, tie rod & lock nut

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

- **turn taper surface in between centres**
- **enlarge and thread cutting**
- **cut and files slot.**

TASK-1

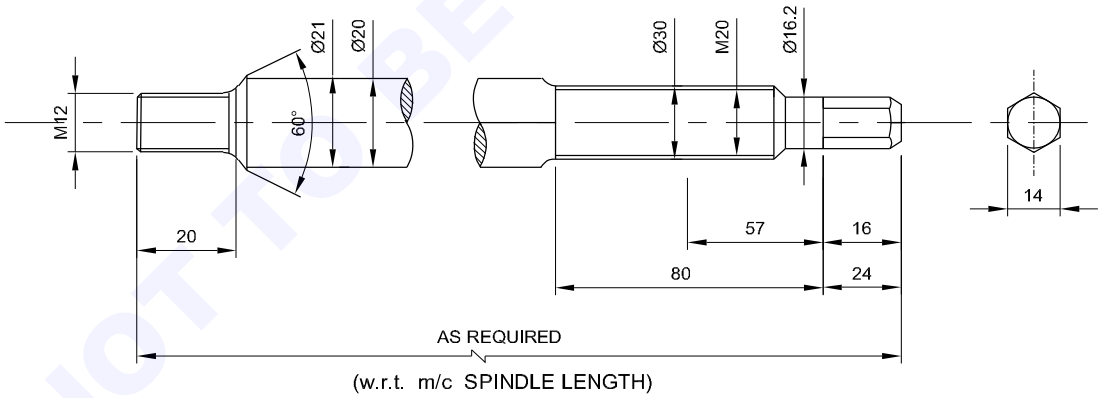
1.STUB ARBOR ISO 40



TAPER RATIO 7:24 = 1:3.4286

TASK-2

2. TIE ROD



(w.r.t. m/c SPINDLE LENGTH)

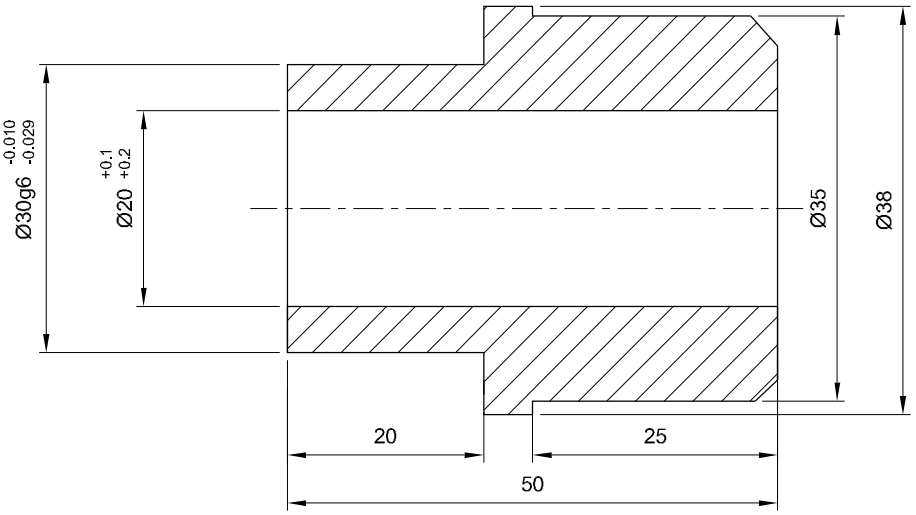
ALL DIMENSIONS ARE IN MM

NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.
1	Ø45x35	-	Fe 310	-	TASK-4	2.3.100
1	Ø40x55	-	Fe 310	-	TASK-3	2.3.100
1	Ø25xINSITTER	-	Fe 310	-	TASK-2	2.3.100
1	Ø65x140	-	Fe 310	-	TASK-1	2.3.100

SCALE NTS	TURNING OPERATION USING FACE PLATE	TOLERANCE.	TIME:
		CODE NO. TU20N23100E1	

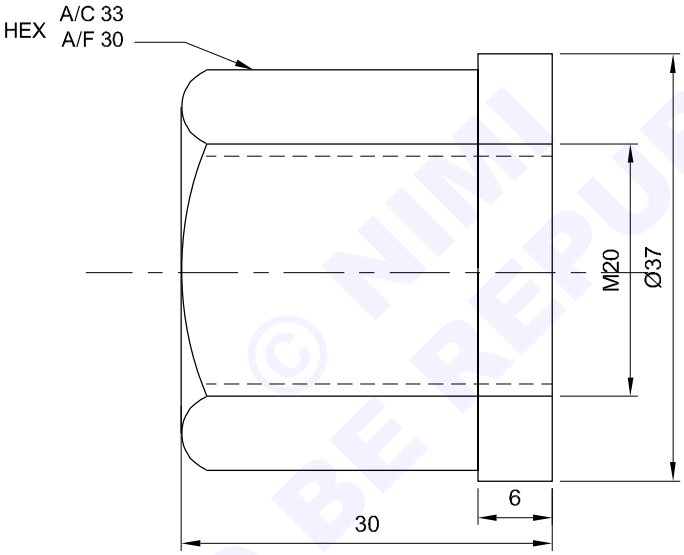
TASK-3

3. COLLAR



TASK-4

4. LOCK NUT



ALL DIMENSIONS ARE IN mm

TU20N23100E2

Job Sequence

TASK 1 : **Turning Stub Arbor**

- Check raw material.
- Prepare machine for operation. (Cleaning, Oil level, back clash adjustment etc.,)
- Hold raw material on 3 jaw chuck face, turn OD Ø 63 x 40, turn step dia Ø 30.5x 22, centre drill/drill to tap M12 x 25 deep, provide chamfer to turn between centre and chamfer corners. (Fig 1)
- Reverse and hold to face to length 125 and centre drill, drill to tap M16 x 70 deep and tap to 30 length. Provide chamfer to turn. (Fig 2)
- Remove job, set machine to turn job between centres arrangement.
- Arrange Taper turning attachment to 8° 17' 50' and turn to finish Ø 44.45. (Fig 3)
- Hold job betweeen centres and turn dia Ø 46 x 95 turn step dia Ø 25.3 x 28 and under cut 5 x Ø 24.
- Remove and finish dia 25.3 x 25 and chamfer corners.
- Remove the job from the chuck.
- Using 'V' block and height gauge mark the slots 16.1 mm width and 9.0 mm depth, both the sides.

Fig 1

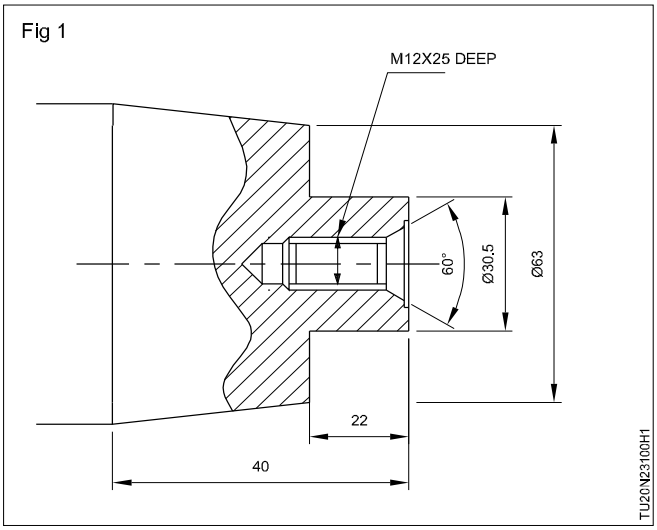


Fig 2

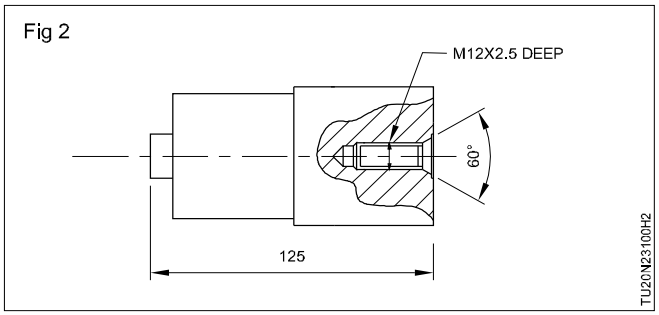
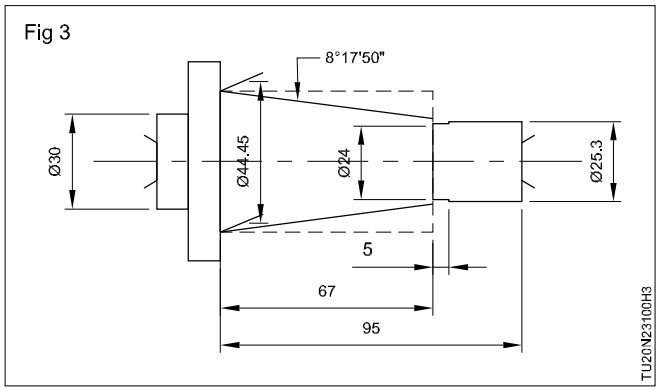


Fig 3



- Punch the marking using prick punch 60° and hammer.
- Remove the excess material using chain drilling.
- File the slot for the size of 16.1 x 9mm.

TASK 2 : **Turning Tie bolt**

- Check the raw material size.
- Hold the job in three jaw chuck.
- Set th tool in correct centre.
- Face the one end make centre drill.
- Reverse the job face to the correct length.
- Hold the job between centres.
- Turn Ø 20 to required length.
- Turn Ø 16.2 to the length of 24 mm.
- Chamfer the corner.
- Cut M20 thread to the length of 80 mm as per drawing.
- Reverse job turn Ø 12 to the 20 mm depth.
- Chamfer the corner and neck.
- Cut M12 thread.
- Remove the job from the machine.
- Mark the hexagon across flat 14 mm on dia 16 mm.
- File and finish the hexagon.

TASK 3 : **Turning Collar**

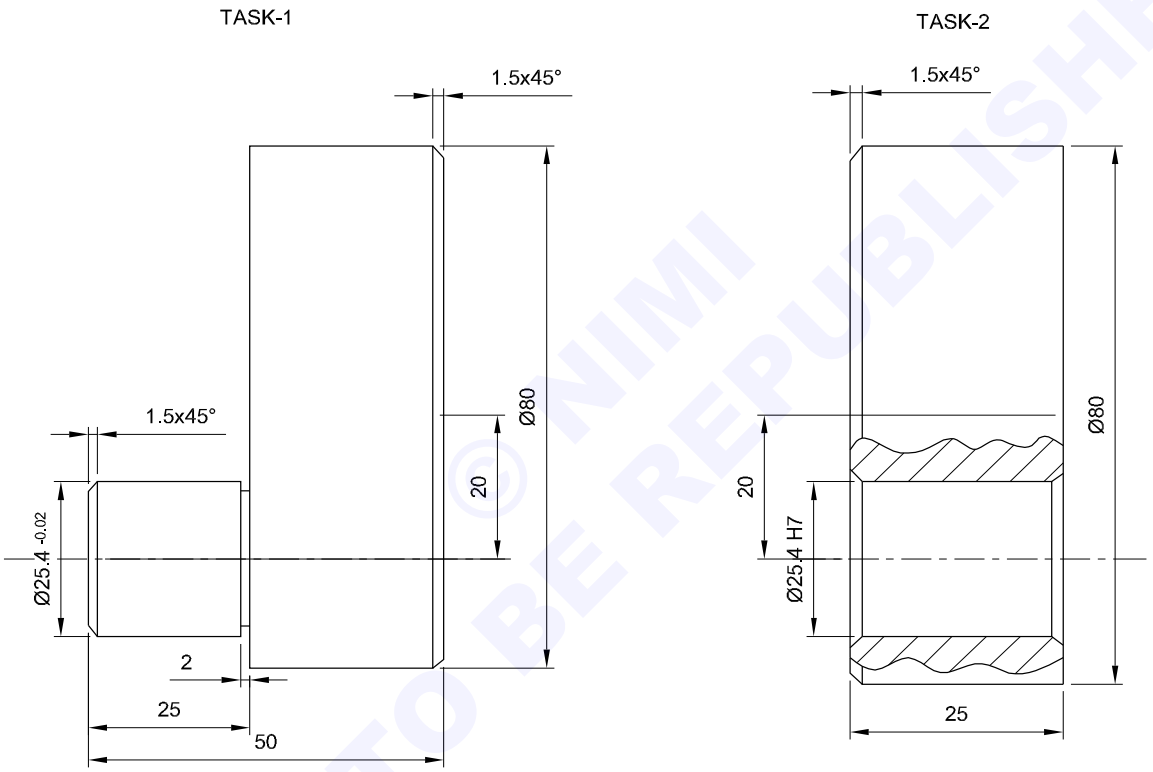
- Check the raw material.
- Hold the job in 3 jaw chuck face end
- Turn to a Ø 38 upto possible length.
- Turn Ø 35 to the depth of 25.
- Make centre drill and pilot drill of 8, 12 and 15 mm.
- Set the boring tool and finish the bore Ø $20^{+0.2}$ mm.
- Reverse the job face to the length of 50 mm.
- Turn Ø 30 for 20 mm depth.
- Remove and deburr the job.

TASK 4 : **Turning Lock nut**

- Check the raw material size.
- Hold the job in 3 jaw chuck face and turn Ø 34.6 to a depth of 24 mm chamfer the corners.
- Reverse and hold in chuck on Ø 33 face to a length of 30 mm.
- Make centre drill, pilot drills and Tap drill size for M20 to full length and chamfer the corners.
- Cut internal thread of M20 using the tap.
- Remove the job from the machine make hexagon to the depth of 24 mm.
- Deburr the job.

Perform eccentric boring (Male & Female eccentric fitting)

Objectives: At the end of this exercise you shall be able to
- **mark eccentric centre & guide circle**
- **set eccentric axis**
- **assemble male and female eccentric portion**
- **check axis mismatch.**



ALL DIMENSIONS ARE IN mm

NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.
1	Ø85-55	-	Fe 310	-	TASK-1	2.4.101
1	Ø85-30	-	Fe 310	-	TASK-2	2.4.101

SCALE NTS	PERFORM ECCENTRIC BORING (MALE AND FEMALE ECCENTRIC FITTING)	TOLERANCE. ±0.06 UNLESS OTHERWISE STATED	TIME:
		CODE NO. TU20N24101E1	

Job Sequence

TASK 1: **Male Part**

- Check the raw material for its size.
- Clamp the facing tool, chamfering tool, turning tool and the undercut tool in a four-way tool post to the correct centre height.
- Hold the material in a 4 jaw independent chuck and true it with the help of the surface gauge.
- Face the end of the job.
- Reverse the job clamp and true it by keeping it 30 mm outside the chuck.
- Face the job to the required length of 50 mm.
- Turn diameter Ø 80 mm with the turning tool.
- Chamfer 1.5 x 45° with the chamfering tool.
- Remove the job from the chuck.
- Mark the centre line and eccentric centre line according to the drawing with the help of marking tools, i.e., surface plate, angle plate, 'V' Block and vernier height gauge.
- Punch both the centre lines with the help of a centre punch and hammer.
- Draw a guide circle of Ø 25 mm with the eccentric centre as the centre point.

TASK 2: **Female Part**

- Check the raw material size.
- Hold it in the independent chuck and true it with a surface gauge by keeping the maximum length outside the chuck.
- Set the facing tool and face one end.
- Set the R.H. turning tool and turn a diameter Ø 80 mm to a maximum possible length.
- Reverse and true the workpiece.
- Set the facing tool face the other end to maintain a total length of 25 mm.
- Set the chamfering tool and chamfer the end to 1.5 x 45°.
- Mark off centre for eccentric drilling and boring with a height gauge. Use surface plate and 'V' Block.
- Scribe Ø 20 mm guide circle for eccentric setting.
- Centre punch and eccentric centre and punch dots for the guide circle. Use centre punch and dot punch.

Open the jaws of the independent chuck and position the workpiece in the jaws such that the centre lines are centered with respect to the jaws. Keep the chamfered face outside the chuck.

- Punch the intersecting 4 points.
- Clamp the job in a 4 jaw independent chuck on the turned diamter by keeping 30 mm outside the chuck.

Ensure that the centre lines are centered with respect to the java.

- True the concentric circle.
- Open one jaw and tighten the opposite jaw so that the centre of the work moves.
- Repeat the operation till the eccentric centre comes in liine with the lathe centre.

Bring the tailstock centre close to the work and ensure that the eccentric centre point coincides with the tip of the tailstock centre.

- Check and true the eccentric circle with the help of the surface gauge for concentricity.
- Turn eccentric diameter 25.4 - 0.02 mm to a length of 25 mm.
- Chamfer 1 x 45° using a chamfering tool.
- Undercut 2 x 0.5 mm and deburr by a flat, smooth file.

– – – – – – – –

- Bring the dead centre very close to the eccentric centre and adjust the opposite jaws to coincide the dead centre point with the eccentric bore centre.
- True the Ø 20 mm eccentric circle with the help of the surface gauge. Open cut and lighten the opposite jaws. such that the centre of the Ø 20 mm moves and coincides with the dead centre.
- Centre drill at the eccentric hole to be drilled.
- Drill hole using three drills Ø 8, Ø 12 and Ø 16.
- Set the boring bar and bore Ø 16 mm hole to Ø 25.1 mm.
- Set Ø 25.4 mm reamer and ream the hole to Ø 25.4 H7.
- Use reduced speed for reaming.
- Set the chamfer tool and chamfer Ø 25.4 mm hole.
- Set the boring bar at the rear side of the hole and chamfer.
- Ensure the fit with the finished exercise 2.4.101 (Task 1)

Skill Sequence

Marking eccentric centres for eccentric turning and eccentric boring

Objectives: This shall help you to
- **mark the eccentric centre for turning the external diameter**
- **mark eccentric centre for internal boring.**

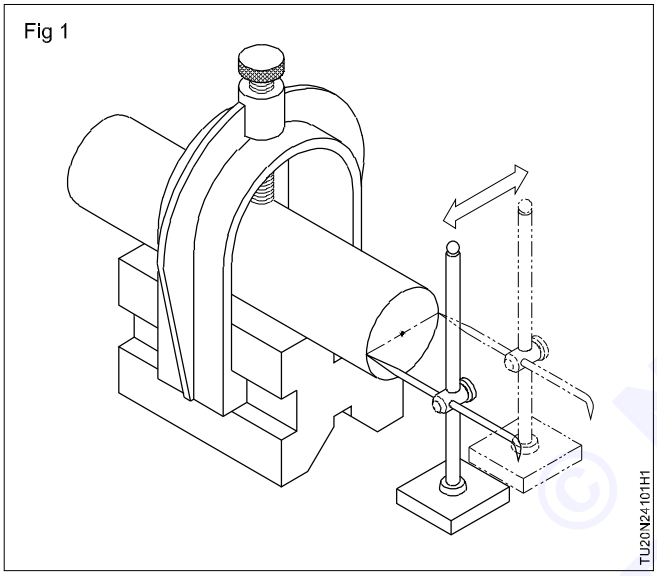
Marking by using 'V' Block, surface gauge and try square

Before marking, the job must be faced at both ends and must be free from burrs and sharp edges.

Clamp the workpiece to the 'V' Block.

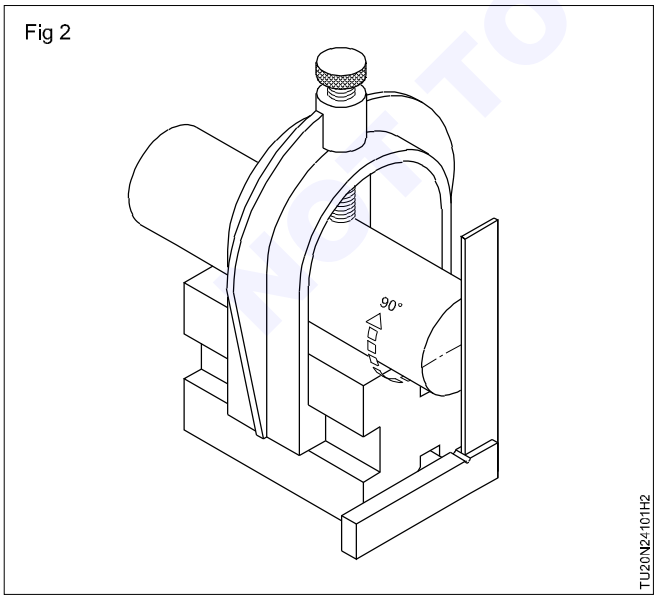
Apply marking media to both the end faces and scribe horizontal lines through the centre of the face (both ends). (Fig 1)

Fig 1



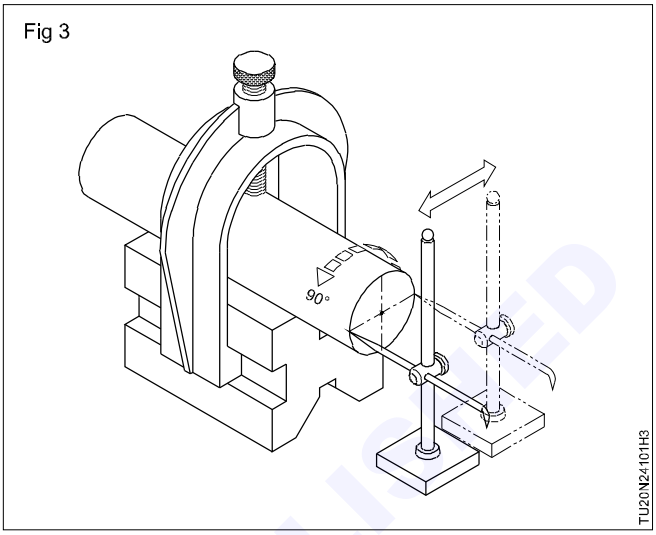
Unclamp and rotate the workpiece through 90°. Set the scribed line vertical using the try square, and reclamp. (Fig 2)

Fig 2



Scribe horizontal lines through the centre of the workpiece on both the end faces. (Fig 3)

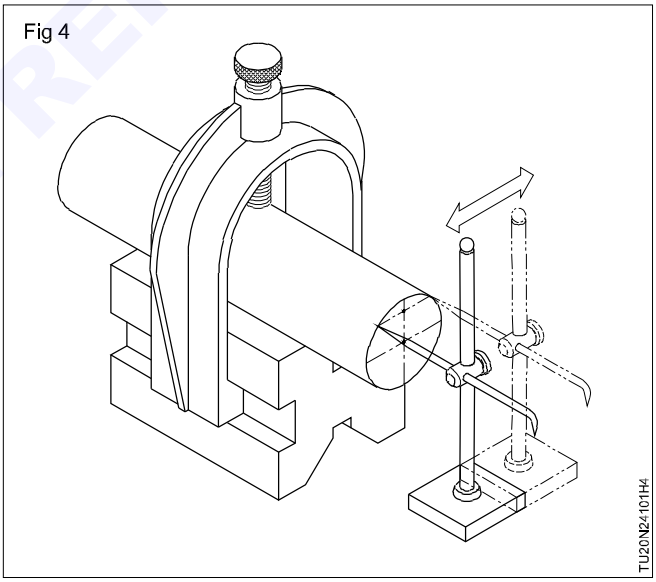
Fig 3



Set the pointer of the scriber at a distance above the centre of the workpiece equal to the amount of eccentricity required. (Fig 4)

Scribe the line at this height on both the end faces of the workpiece. (Fig 4)

Fig 4



Remove the workpiece from the 'V' Block.

Dot punch intersection points on both the end faces, to have concentric and eccentric centres.

When marking off centres of a workpiece, a 'V' Block, which can be laid on its side, may be used. This will avoid the need to unclamp the workpiece and square.

Centre drill the workpiece on both the end faces on the drilling machine.

While centre drilling make sure that the centre drill locates the centre dots accurately.

If the eccentric turning is meant for a 4 jaw chuck, scribe guide circles with eccentric centre as the centre of the guide circle, using a divider.

Marking by using angle plate, clamps, surface gauge and try square

This marking method is used for securing the workpiece on a face-plate. Before marking, the job must be faced at both ends and it should be free from burrs and sharp edges.

Check the locating face of the workpiece for flatness. Use a straight edge and light to ensure the flatness of the locating surfaces.

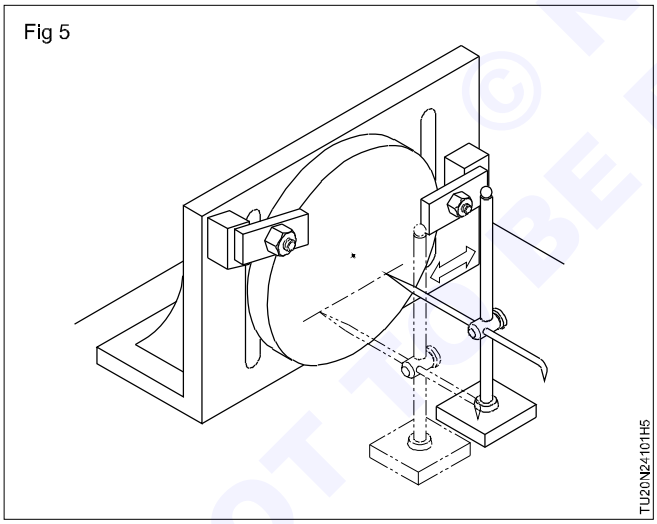
The workpiece must be clamped square to the angle plate to ensure squareness of the marking with reference to the previously machined faces.

Clamp the workpiece lightly to the angle plate with the outside diameter of the workpiece touching the marking off table. (Fig 5)

Measure the outside diameter of the workpiece; set the scriber point to half of the diameter minus bore eccentricity.

Scribe a horizontal line. (Fig 5)

Fig 5



Unclamp, swivel the work through 90°, and with a try square, check that the scribed line is vertical, and reclamp. (Fig 6)

Set the scriber point to mark off centre of the workpiece. Scribe a horizontal line to intersect the vertical line already scribed. (Fig 7)

Fig 6

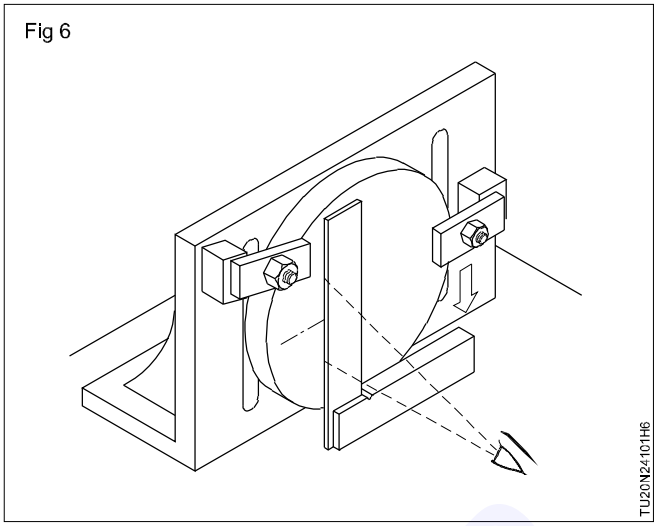
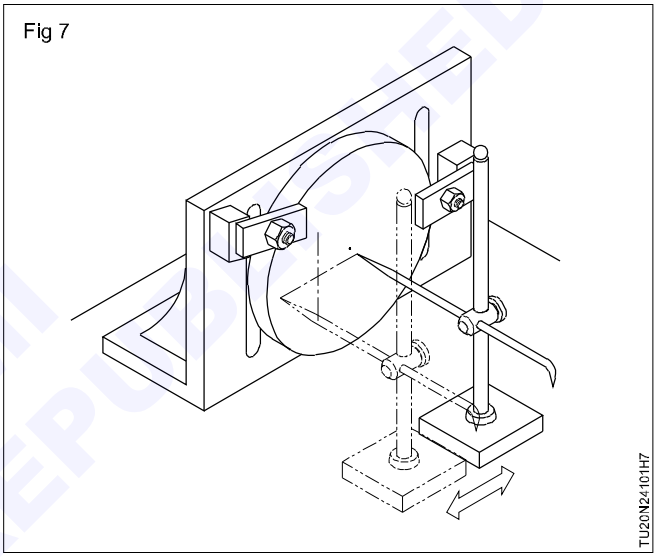


Fig 7



Unclamp and remove the workpiece from the angle plate. Place flat on the marking off table and with the marked off face uppermost. Dot punch intersection points.

Set the divider to the radius of the finished bore size.

Scribe a circle using the centre dot as centre point.

Unclamping of the workpiece may be avoided by laying the angle plate on its side. In such cases, the workpiece has to be set on the angle plate, such that the edges of the workpiece touch the marking table in both the positions i.e., before laying the angle plate on its side and after laying the angle plate on its side.

Marking eccentricity of a job with vernier height gauge to given dimensions

Objective: This shall help you to

- **mark concentric and eccentric centre lines of a job by using a vernier height gauge.**

The height gauge marking is more accurate than the scribing block marking.

The marking surface must be free from sharp edges and unevenness.

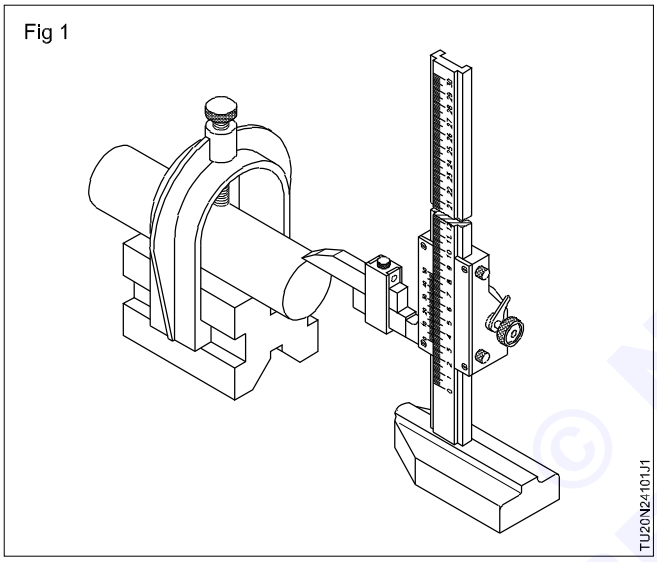
Clamp the finish turned rod in the 'V' Block with the help of the clamps.

Apply marking media on both faces of the job.

Set the scriber point on the top edge of the job. (Fig 1)

Move the height gauge scriber over the round surface to get a feel that the scriber bottom face is contacting the work periphery. (Fig 1)

Fig 1

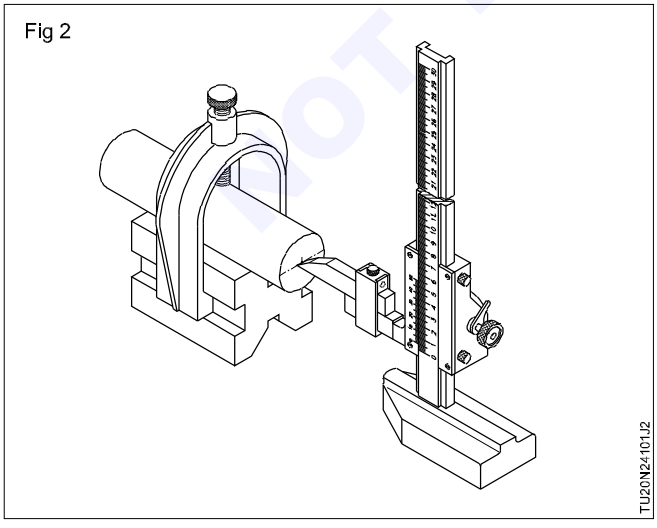


Lock the slides and note down the readings of the scales.

Subtract half the diameter from the reading and set the height gauge for that reading. (Fig 2)

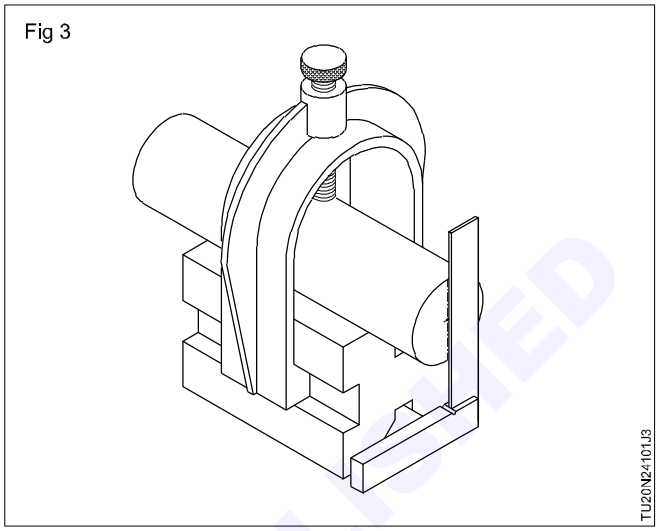
Scribe a horizontal line on both faces. (Fig 2)

Fig 2



Release the workpiece from the clamp and rotate the workpiece through 90°. Set the line at 90° with the help of a try square. (Fig 3)

Fig 3



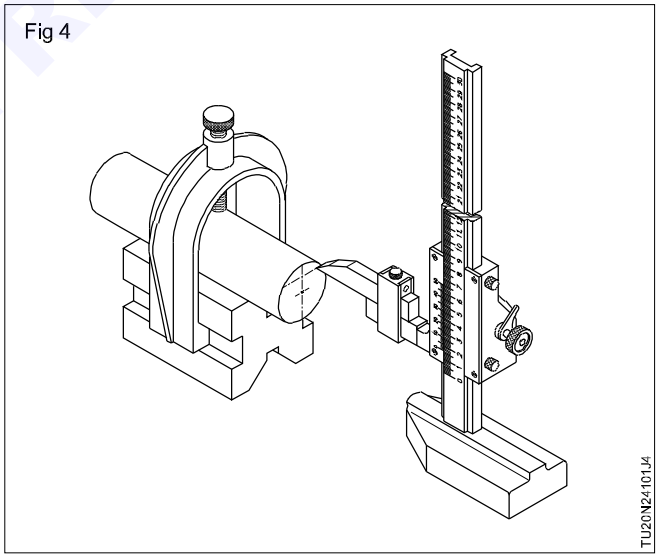
Clamp the workpiece to the 'V' block.

Scribe horizontal lines on both faces with the same reading which is set for centre position. (Fig 4)

Add eccentricity amount to the above reading and reset the height gauge for the new rading (Fig 4)

Scribe horizontal lines on both faces. (Fig 4)

Fig 4



Release the workpiece from the 'V' Block.

Punch mark on both sides both concentric and eccentric centre points.

Truing a job for eccentric turning and eccentric boring

Objectives: This shall help you to
- **true the job for external eccentric turning**
- **true the job for internal eccentric boring.**

Truing the eccentric job in a four jaw chuck

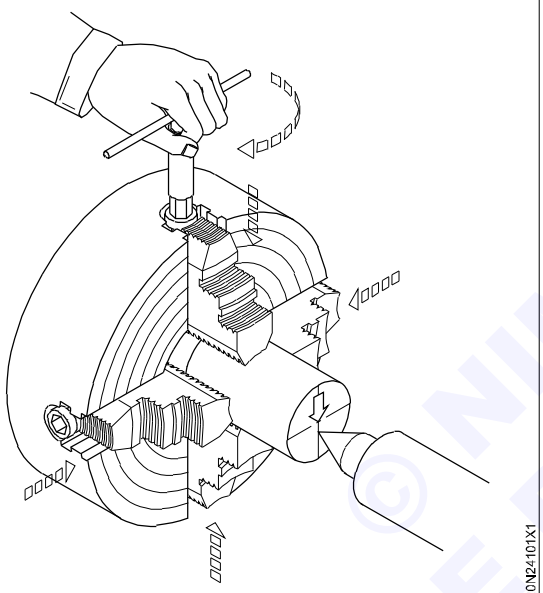
Open all the four jaws to give clearance to the workpiece.

Hold the workpiece up to the chuck face with the scribed lines towards the tailstock.

Insert the tailstock centre and slide the tailstock over the bed towards the headstock.

Position the workpiece until the tailstock centre locates in the eccentric centre dot on the workpiece. (Fig 1)

Fig 1



Move the tailstock centre until the pressure applied holds the workpiece against the chuck face.

Move the chuck jaws, tighten each jaw lightly in turn, taking care not to shift the workpiece.

Check and adjust the position of the workpiece so that it will protrude enough from the jaws to allow the total length of the eccentric portion to be machined.

Tighten the jaws.

Remove the tailstock.

Truing of eccentric job held in a four jaw chuck by using a surface gauge

Most of the eccentric truing is done with the help of guide circles scribed on the face of the chuck and the surface gauge. Since the guide circle has been scribed in concentric with the eccentric marking, truing the work to the guide circle gives the exact centre point of the eccentric turning.

Open all the four jaws to give clearance to the workpiece.

Hold the workpiece up to the chuck face.

Set the scriber of the surface gauge over the lathe bed.

Rotate the chuck by hand and check the running of the centre dot or guide circle with the surface gauge pointer.

Tighten each jaw slightly in turn, after necessary adjustments of the two sets of opposite jaws are made.

Recheck the centre dot or guide circle with the surface gauge.

Realign the jaws, if required.

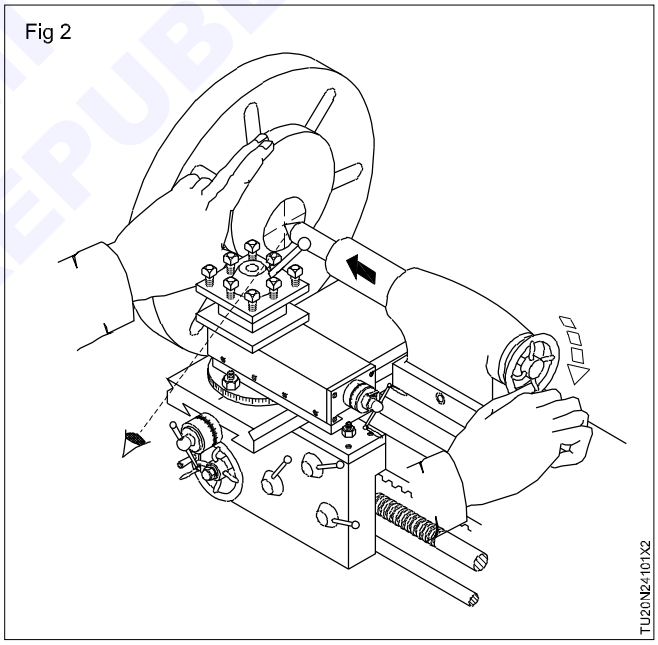
Tighten the jaws fully.

Truing the eccentric job held on a face plate

Hold the workpiece on to the face plate.

Bring up the tailstock, locate the tailstock centre to the eccentric centre dot, apply pressure until the workpiece is held in position. (Fig 2)

Fig 2



If the eccentric bore in the workpiece is to be through then the parallel bars must be placed behind the workpiece to clear off the face plate during drilling and boring.

Select suitable clamps, Tee bolts, nuts, washers and packings as required.

Clamp the workpiece to the face plate, clamp in positions which will give support ie., as near to the holes as possible but diametrically opposite. (Fig 3)

Remove the tailstock.

Recheck the tightness of the clamps.

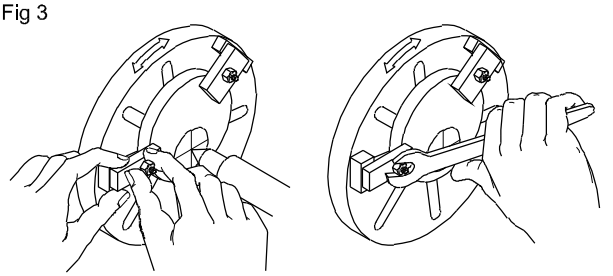
Rotate the face plate by hand and check the concentricity of the marked off bore with the help of the surface gauge.

Realign the job, if required.

Recheck the tightness of the clamps.

If the amount of eccentricity is great, then the counter-balance weights must be clamped or bolted to the face plate to give a balanced condition.

Fig 3



Eccentric turning and boring

Objectives: This shall help you to
- **turn external eccentric diameter**
- **bore internal eccentric diameter.**

Turning external eccentric diameter, work held in a 4 jaw chuck

The procedure of setting the work to turn the eccentric shaft has already been dealt with.

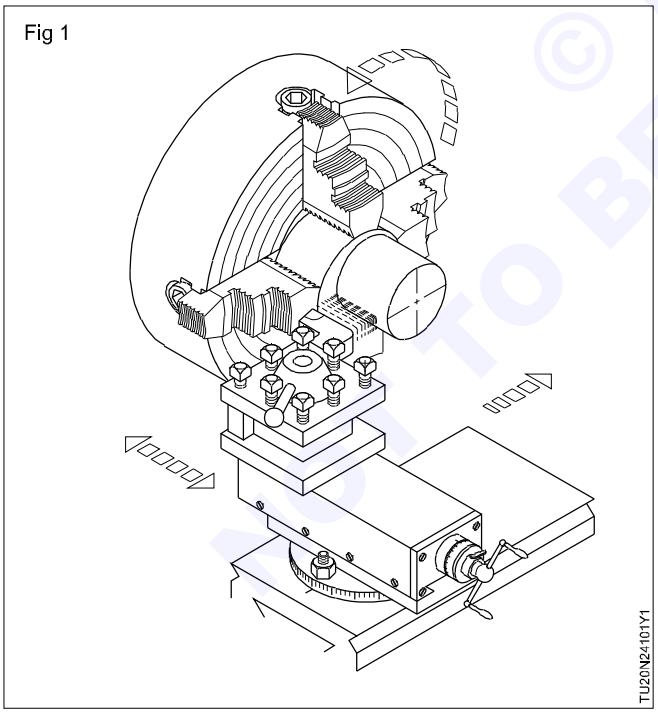
Further steps for turning are given below.

Set the tool to centre height with a minimum overhang.

Ensure that the tool tip is clear off the eccentric throw at the commencement of the operation as shown in Fig 1.

Rough turn by successive cuts the eccentric diameter leaving approximately 0.8 mm in the diameter for finish turning. (Fig 1)

Fig 1



Set the finishing tool and finish turn to the diameter.

Face to length.

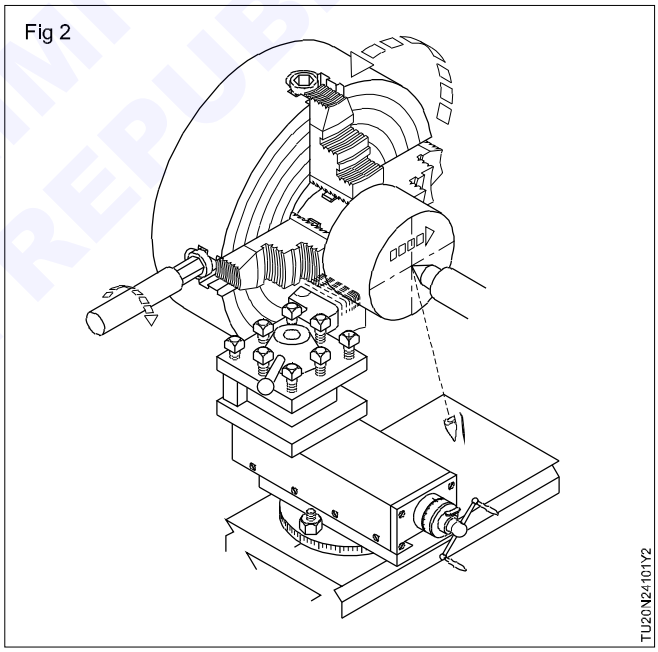
Remove the workpiece from the chuck.

Reverse and reset the job for turning concentric diameter. Use packing strips to protect the turned diameter held in chuck. (Fig 2)

Bring up the tailstock centre to the workpiece.

Release the chuck jaws, rotate the workpiece until the centre dot of the concentric centre is in line with the tailstock centre. (Fig 2)

Fig 2



Check that all packing strips are in position.

Tighten the jaws.

Remove the tailstock.

Check the running of the concentric centre dot with the help of the surface gauge.

Realign the jaws, if required.

Give each jaw a final tightening.

Make sure that the running of the concentric centre dot is in line with the lathe axis.

Set the tool for rough turn.

Rough turn the concentric diameter.

Set the tool for finish turn, and finish turn the concentric diameter.

Face to length.

Remove the eccentric turned job from the chuck.

Eccentric boring work held in a face plate

Truing has already been dealt with.

Rotate the face plate by hand to check that the clamps etc. are clear of the cross-slide and saddle.

Centre drill.

Drill a hole for boring. Use a suitable coolant, and drill the hole on slow r.p.m. Keep an even pressure on the tailstock hand wheel. Take care when the drill is breaking through.

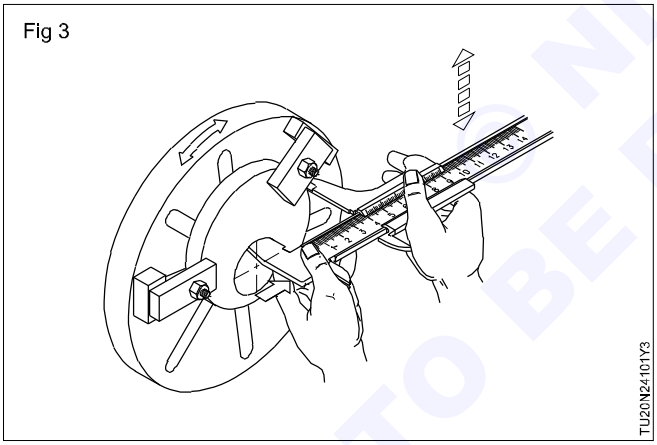
Set the boring bar/tool in the tool post.

Check and ensure that the length of the boring bar will clear the bore and clamping bolts.

Rough cut bore.

Check the position of the hole from the outside diameter of the workpiece. Measurement should be taken on the line where maximum eccentricity occurs. (Fig 3)

Fig 3



Adjust the work position, if needed.

Finish bore to size.

Check the diameter.

Unlock the tool post and swing the boring tool clear.

Bring up the tailstock to support the workpiece.

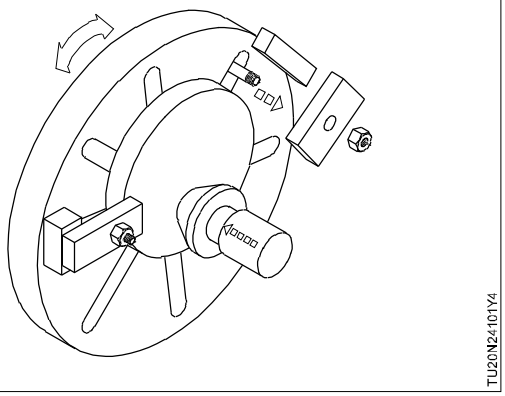
Remove the clamps, retract the tailstock and remove the workpiece from the machine. (Fig 4)

Return the tailstock to the extreme right hand end.

Remove all the counter-balance weights from the face plate.

Eccentric turning on work held between centres.

Fig 4



Centre drill the marked dots at both ends. (Concentric and eccentric centres)

Mount the catch plate to the lathe spindle.

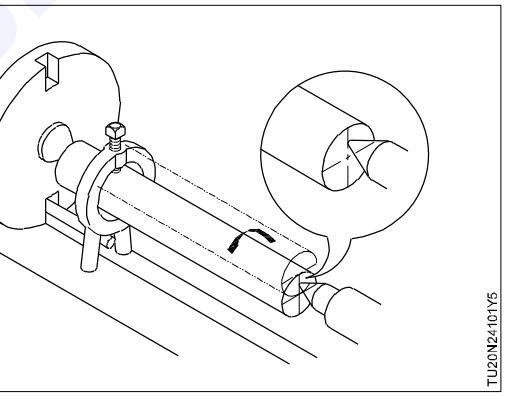
Clamp the carrier to the workpiece.

Grease the dead centre.

Support the workpiece between the eccentric centres for eccentric turning.

Ensure a positive drive from the catch plate to the carrier, and a minimum overhang of the tailstock barrel. Rotate the catch plate by hand to check that the workpiece is in the correct plane. (Fig 5)

Fig 5



Clamp the tool in the tool post to correct centre height with a minimum overhang.

Rough turn eccentric diameter.

Take successive cuts until over half the diameter is machined.

Check and determine the amount of metal to be removed.

Rotate the workpiece by hand and touch the tool to the workpiece at the highest point of the throw and set the cross-slide graduated collar to zero.

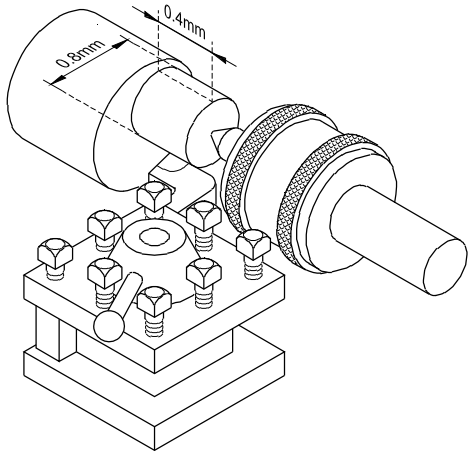
Rough turn the eccentric diameter leaving 0.8 mm for finish turning. (Fig 6)

Turn length, leaving 0.4 mm for finish turning. (Fig 6)

Remove the workpiece and reset the job for concentric turning.

Clamp a suitable carrier to the eccentric diameter. Use soft packing strips between the finished diameter and the carrier.

Fig 6



Hold the workpiece between the concentric centres.

Check that the workpiece is located in the correct plane.

Rough turn the diameter.

Finish turn the diameter to size.

Remove the workpiece and reset the job for finish turning the eccentric diameter.

Clamp the carrier to the finished concentric diameter. Use soft packings to protect the finished diameter.

Finish turn the eccentric diameter to size.

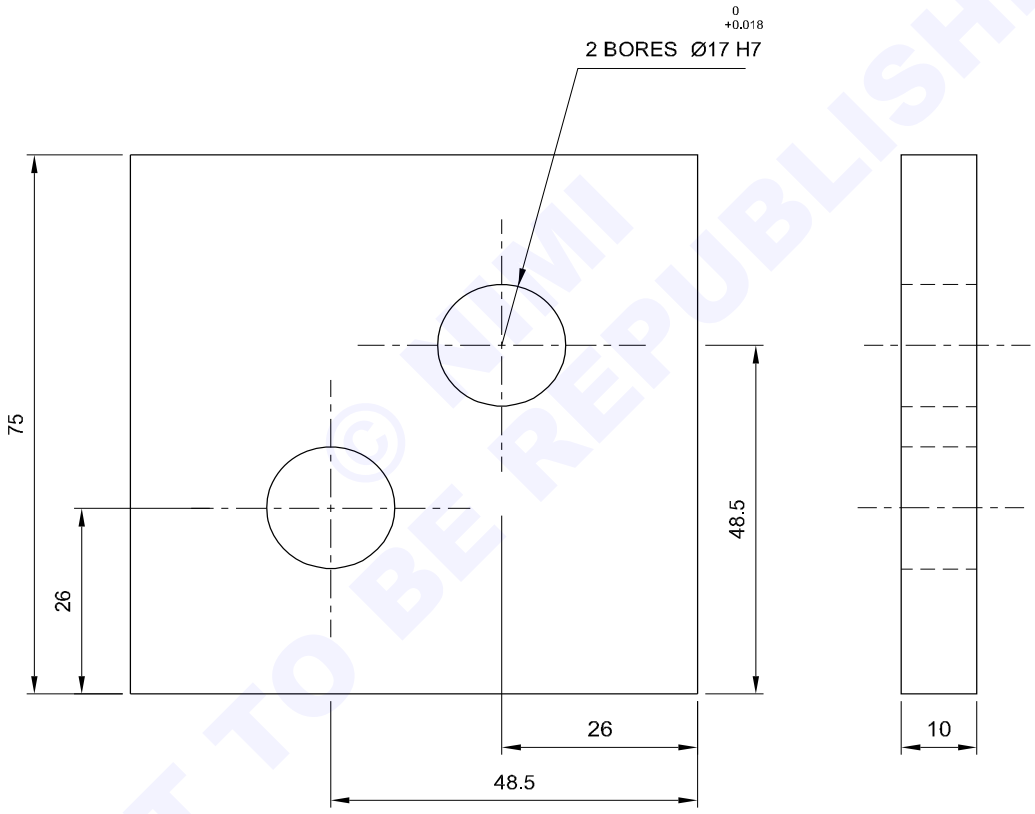
Finish turn the length to size.

This procedure applies only to jobs which will have both the centre holes available for roughing and finishing operations.

Position boring using tool makers button

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

- **make use of tool maker's button**
- **set and mark bore offsets**
- **set on four jaw to bore offset bores.**



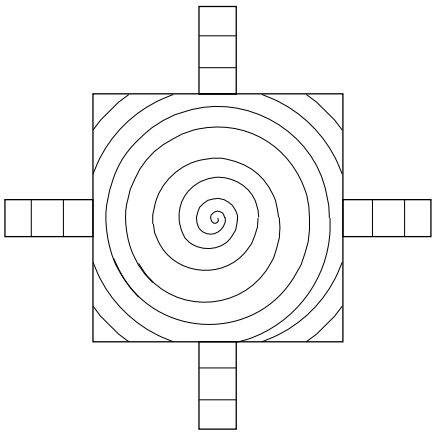
1	80ISF 12x80	-	Fe 310	-	-	2.4.102
NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.

SCALE NTS	POSITION BORING USING TOOL MAKER'S BUTTON	TOLERANCE.	TIME:
		CODE NO. TU20N24102E1	

Job Sequence

- Check raw material and deburr.
- In 4 jaw chuck make blank of 10 x 75 x 75. (Fig 1to4)

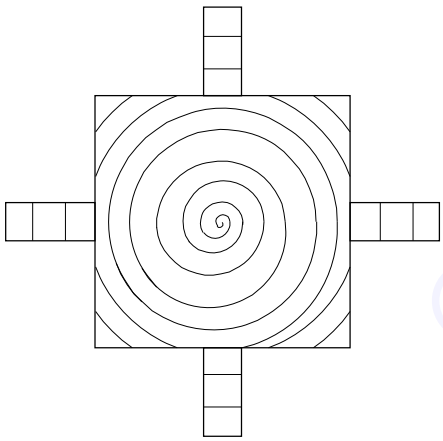
Fig 1



SET UP FOR FACING THICKNESS

TU20N24102H1

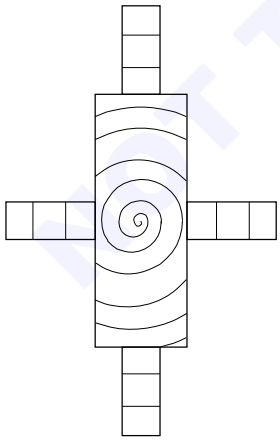
Fig 2



SET UP TO MAINTAIN THICKNESS

TU20N24102H2

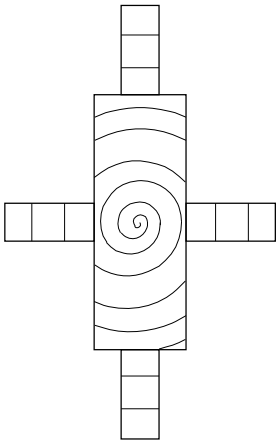
Fig 3



SET UP FOR FACING WIDTH

TU20N24102H3

Fig 4

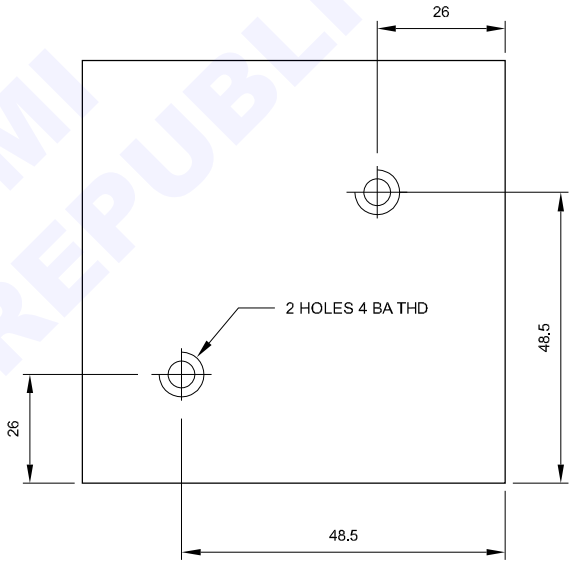


SET UP TO MAINTAIN WIDTH

TU20N24102H4

- Mark two centre points of 26, 48.5 two points as per drawing. (Fig 5)

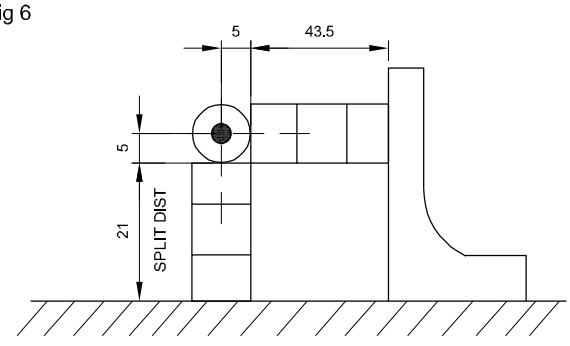
Fig 5



TU20N24102H5

- Drill to Tap to suit 4BA.
- Fix tool maker's button Ø 10.
- Set the centre to 25.88, 48.5 co-ordinate. (Fig 6)

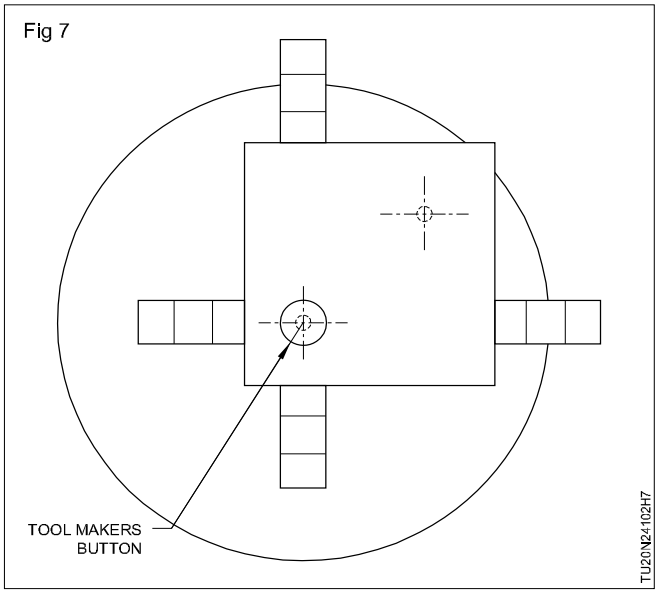
Fig 6



TU20N24102H6

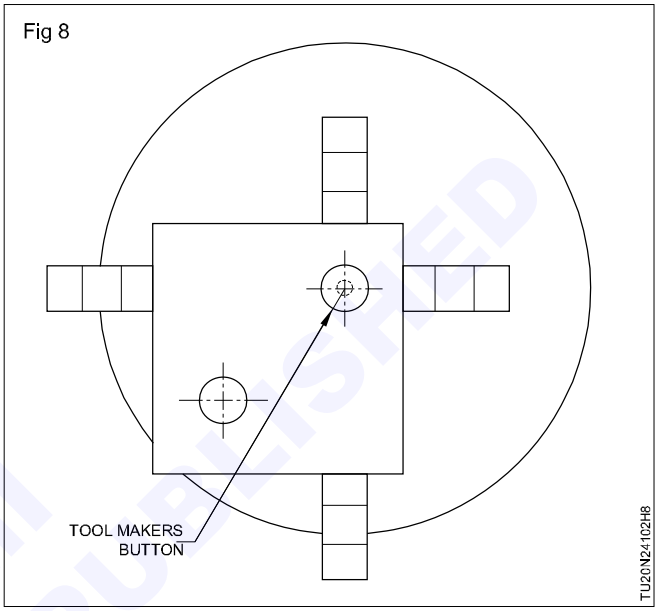
- Fix job in 4 jaw chuck and tool maker's button Ø 10 to lathe axis.
- After aligning remove tool maker's button. (Fig 7)

Fig 7



- Remove tool makers button drill and bore to dia Ø 17 $H_7^{+0.018}$
- Remove job and fix tool maker's button in the other centre.
- But the job against angle block build slips 26 - 5 = 21 mm, adjust the roller against slip. Set the dial test indicator 0' on surface plate and raise height to 26 + 5 = 31 mm and compare with slips. Then check and correct the tool maker's button to 26 of co-ordinate centre. Similarly the distance 48.5 is set.
- Fix job in 4 jaw chuck and align tool maker's button to axis of lathe. (Fig 8)
- Remove tool maker's button.
- Drill and bore to dia 17 $H_7^{+0.018}$
- Remove job and deburr.

Fig 8



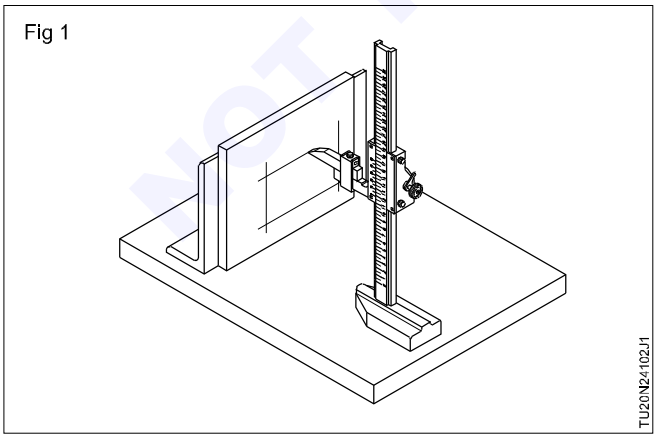
Skill Sequence

Setting job for eccentric boring

Objective: This shall help you to
- **setting job for eccentric boring using tool maker's button.**

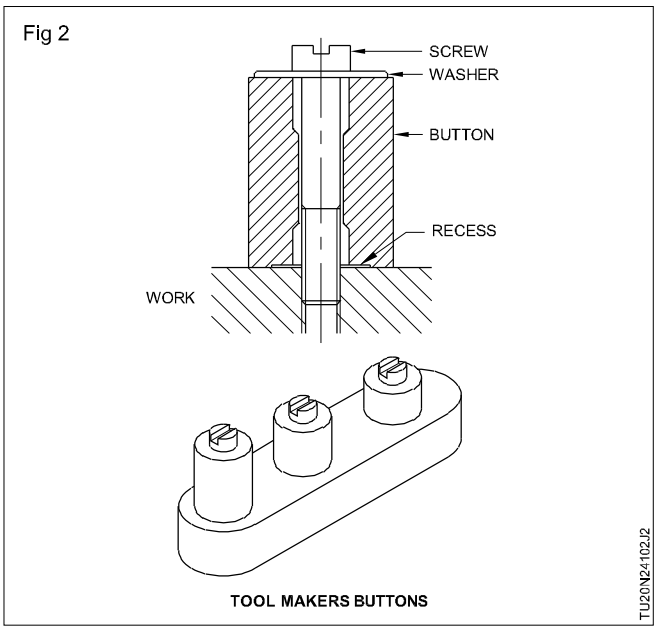
- Mark the centres as per rawing vernier height gauge. (Fig 1)
- Punch the centre marked & drill Ø 8 mm.

Fig 1



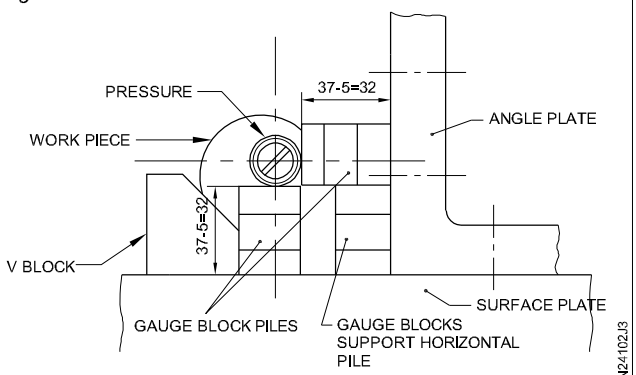
- Tap M10 threaded hole.
- Fit a tool maker's button over this threaded hole. (Do not tighten fully). (Fig 2)

Fig 2



- Adjust the button position by building up by slip gauge block. (Fig 3)

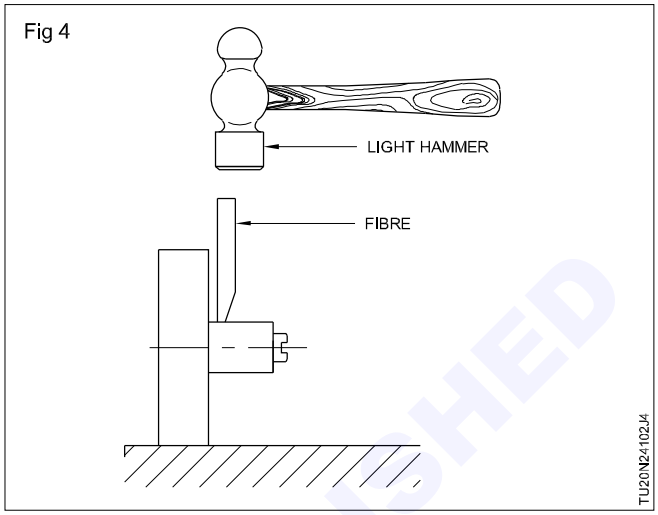
Fig 3



SETTING THE BUTTON

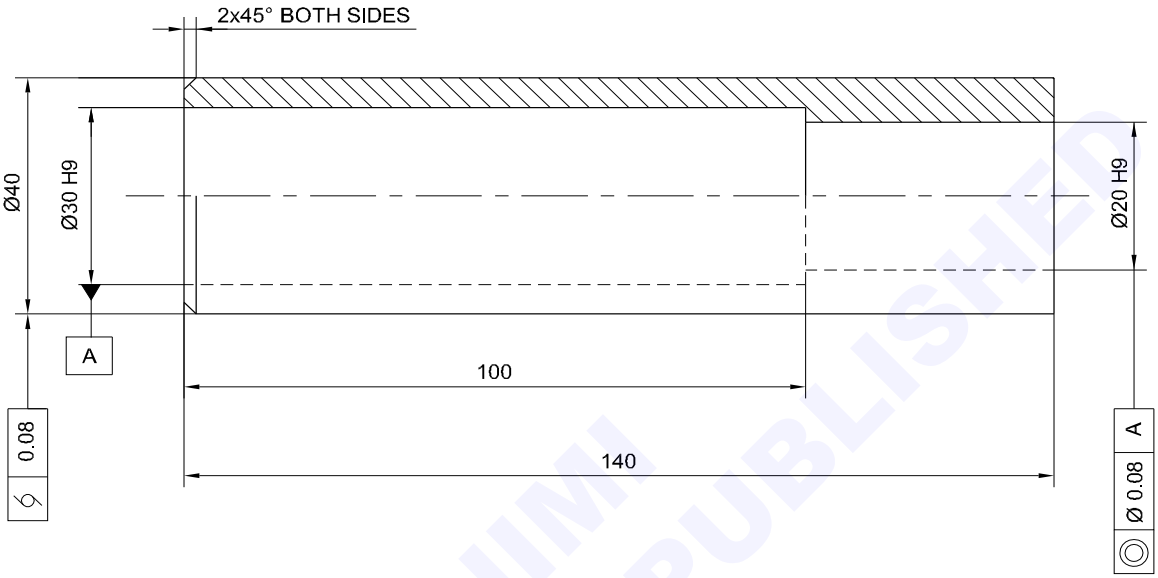
- While positioning the button, tap gently as shown in Fig 4
- After the button positioned as per the required dimension tighten the screw fully.
- Finally mount the work in four jaw chuck & true it by using dial test indicator.

Fig 4



Boring and stepped boring

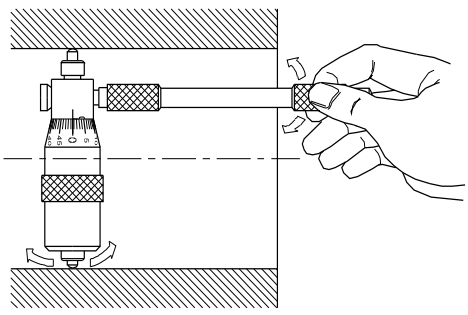
Objectives: At the end of this exercise you shall be able to
- **bore deep holes within close limits**
- **check the bore dia using inside micrometer.**



Job Sequence

- Check the raw material size.
- Hold the job in three jaw self centring chuck. Projecting 110 mm outside the chuck.
- Face the end.
- Turn to Ø 40 mm for a maximum possible length.
- Reverse the job & face to maintain a length of 140 mm.
- Centre drill the end.
- Turn to dia 40 mm for a remaining length.
- Drilling step by step (first pilot drill of Ø 10 mm, Ø 16 mm & Ø 22 mm) for full length.
- Set the boring tool & bore to dia 20 H_9 through hole.
- Check the hole by using inside micrometer.
- Step bore to Ø 30 H_9 for a length of 100 mm (By giving several rough cuts & finally by finish cut)
- Check the stepped bore using inside micrometer. (Fig 1)
- Deburr sharp corners if any.

Fig 1



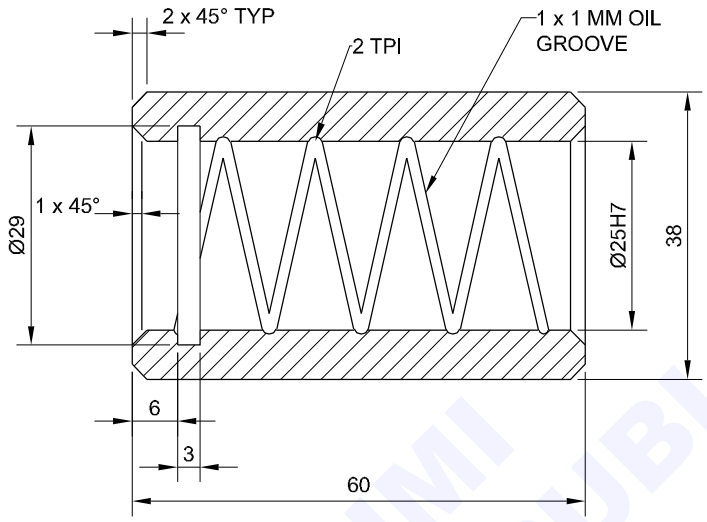
USING AN EXTENDED HANDLE WHEN MEASURING A DEEP BORE

TU20N24103H1

1	Ø45 - 150	-	Fe 310 - O	-	-	2.4.103
NO.OFF	STOCK SIZE	SEMI PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.
SCALE 1:1	BORING AND STEPPED BORING				DEVIATIONS IS:2102 (M)	TIME
					CODE NO. TU20N24103E1	

Cutting helical grooves (in bearing/bushes)

Objectives: At the end of this exercise you shall be able to
- **finish the bore within close limits**
- **cut the oil groove**
- **chamfer the edges.**



Job Sequence

- Hold the job in a four jaw chuck and true it.
- Face one end.
- Turn dia 38 mm for the maximum possible length.
- Chamfer the end 2 x 45º.
- Reverse the job and reset it.
- Face the other end and maintain to 60 mm length.
- Turn to ∅38 mm.
- Chamfer the end 2 x 45º.
- Centre drill the job with 60° type 1
- Drill the job with a pilot drill ∅12 mm
- Drill the job with a ∅20 mm drill bit.
- Bore the drilled hole to ∅24.75 mm and chamfer 1 x 45º.
- Ream the job with a ∅25H7 machine reamer.
- Form the recess ∅29 mm x 3 mm width in after 6 mm length. (as per drawing)
- Chamfer the other end of the hole after reversing and resetting.
- Check the dimensions after deburing.
- Hold the job in the chuck, select minimum speed.
- Hold the selected 1 mm grooving tool in the tool post.
- Change gear setting for 2 TPI.
- Cut the oil groove 1x1 on bore ∅25 to 50 mm length.
- Deburr the chips in the grooving area.

1	Ø42 - 65	–	Fe 310	–	–	2.4.104
NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.

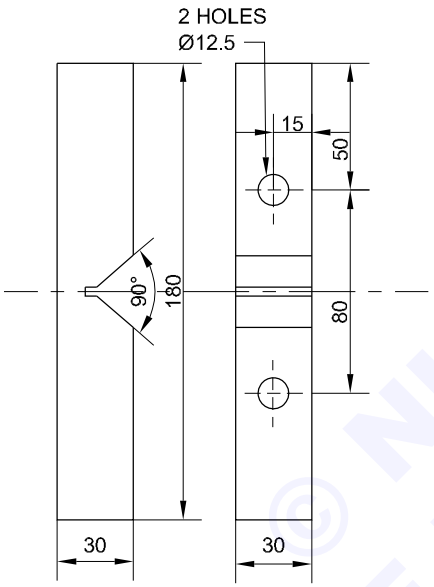
SCALE 1:1	CUTTING HELICAL GROOVES (IN BEARING/BUSHES)	TOLERANCE ± 0.5 on dia and length	TIME
		CODE NO. TU20N24104E1	

Turning & boring of split bearing (using boring bar & fixture)

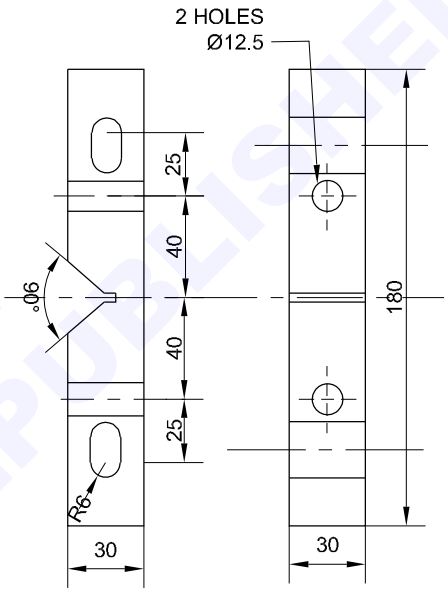
Objectives : At the end of this exercise you shall be able to

- **face the work squareness to all the sides**
- **mark for 90º V groove**
- **drill holes to required size**
- **file and form V groove**
- **drill hole at right angles to the axis of work.**
- **split the bush in vertical position by hacksawing**
- **set the job in fixtures.**

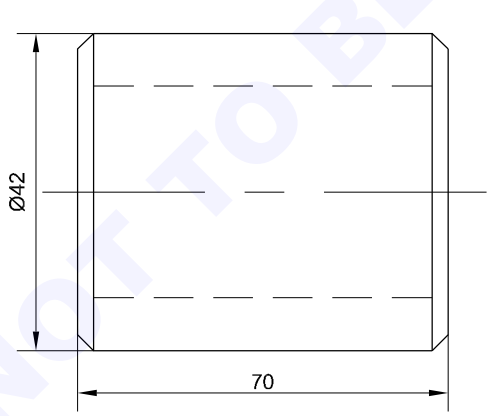
TASK-1



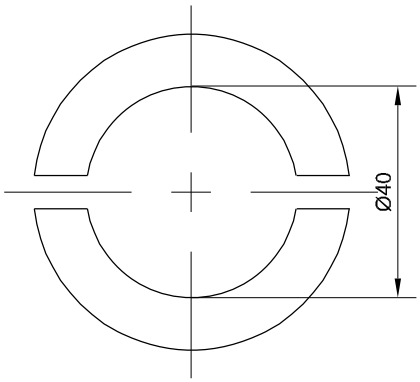
TASK-2



TASK-3



BEFORE SPLITTING



AFTER SPLITTING

1	32SQ-185	–	Fe 310	–	TASK- 1 AND 2	–
1	Ø50 - 75	–	Fe 310	–	TASK - 3	2.4.105
NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.
SCALE 1:1	**TURNING & BORING OF SPLIT BEARING (USING BORING BAR & FIXTURE)**				DEVIATIONS ± 0.1	TIME
					CODE NO. TU20N24105E1	

Job Sequence

TASK 1&2 : **Fixture**

- Check the raw material for its size.
- Hold the raw material about 10 mm outside the chuck and true.
- Face one end.
- Reverse and hold work and true.
- Face and maintain a total length of 180 mm.
- Remove and hold work in vertical position and true it. (By reversing the two opposite jaws on 180 mm side)
- Face the job on all four sides maintaining squarness to 30 mm size.
- Mark and punch for ∅12.5 mm drill hole location.
- Centre drill and drill ∅12.5 mm hole by holding the work on a vice.
- Mark for the 90º V groove as per drawing. (By reversing 'V' block, Vernier height gauge & Vernier bevel protractor)
- Remove the extra material with a hacksaw leaving sufficient metal for finish filling the 'V' groove.
- File and finish the 'V' groove and check with universal version bevel protractor.

Note: Repeat the same sequence to complete task 2 component and made slot of 22 mm as per drawing.

TASK 3 : **Split bush**

- Check the raw material for its size.
- Hold the job in a four jaw independent chuck and true it.
- Face one end and turn ∅50 mm for 25 mm long.
- Finish turn ∅48 for 25 mm long.
- Reverse the job and true it.
- Face other end and maintain a total length of 70 mm.
- Turn remaining portion to ∅48 mm.
- Drill dia 10 mm through hole.
- Enlarge the through hole to ∅25 mm.
- Fix the boring bar in the tool post and bore ∅28 hole for the full length by giving successive cuts.
- Split the bush by hacksawing.
- File the saw cut surface and finish the surface flat.
- Hold the job in a fixture (Task 1&2)
- Finish turn ∅45 mm for maximum length.
- Reverse the job and hold in a fixture.
- Turn dia 45 mm in remaining length.
- Finish the bore dia 40 mm for the full length.
- Check the bore with help of inside micrometer.

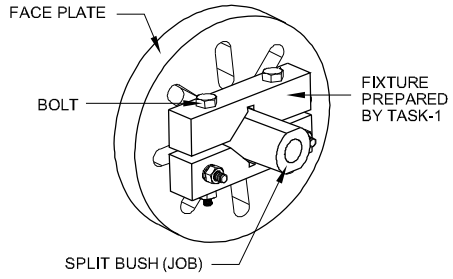
Skill Sequence

Set the job in fixture

Objective: This shall help you to
- **set the job in fixture.**

Set the Job in fixture and set the fixture in the face plate. (Fig 1)

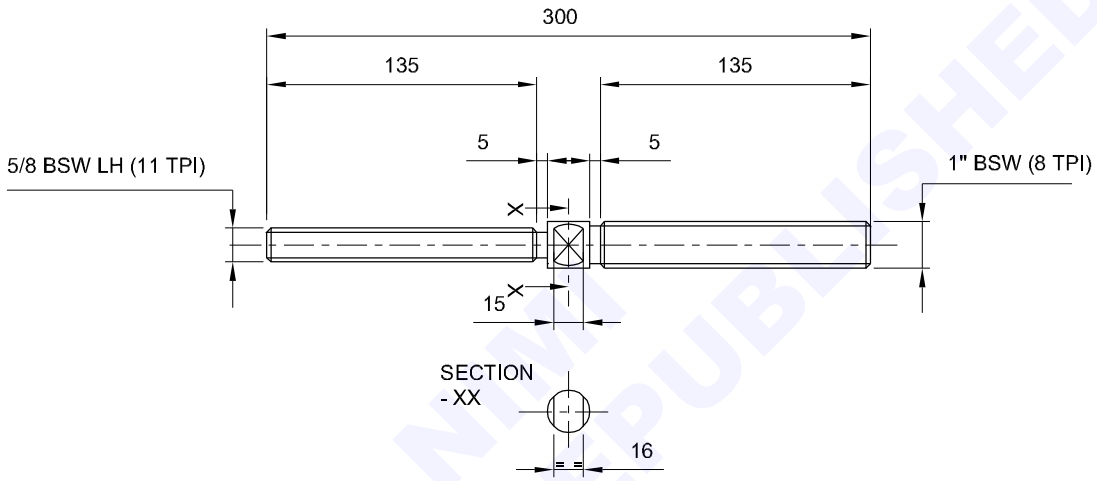
Fig 1



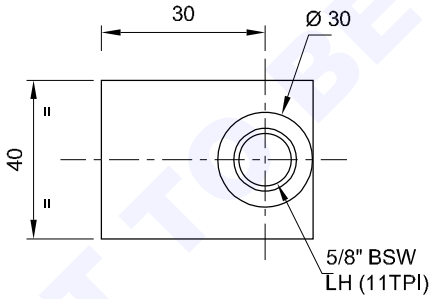
Cutting thread of 8 & 11 TPI

Objectives: At the end of this exercise you shall be able to
- **inch thread RH & LH on long shaft**
- **turn rectangular block on lathe**
- **drill & thread RH & LH**

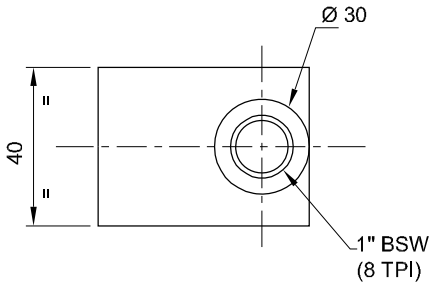
TASK-1



TASK-2



TASK-3



1	20x40x50	-	30C8	-	TASK-3	2.5.106
1	20x40x50	-	30C8	-	TASK-2	2.5.106
1	Ø30x305	-	30C8	-	TASK-1	2.5.106
NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.

SCALE NTS	**CUTTING THREAD OF 8 & 11 TPI**	DEVIATIONS	TIME:
		CODE NO. TU20N25106E1	

Job Sequence

TASK 1 : **Turning & Extrnal threading**

- Check the raw material.
- Face and centre drill.
- In between centre turn Ø 25.4 mm, make undercut 5 mm width through cut threads 1” BSW (11 TPI) to 135 length.
- Similarly on other end turn turn Ø 15.375 mm, make undercut 5 mm width, cut threads 5/8” BSW LH 11 TPI)
- File flat 16 mm across flat to 15 mm width.

– – – – – – –

TASK 2 & 3 : **Turning bush and Internal threading**

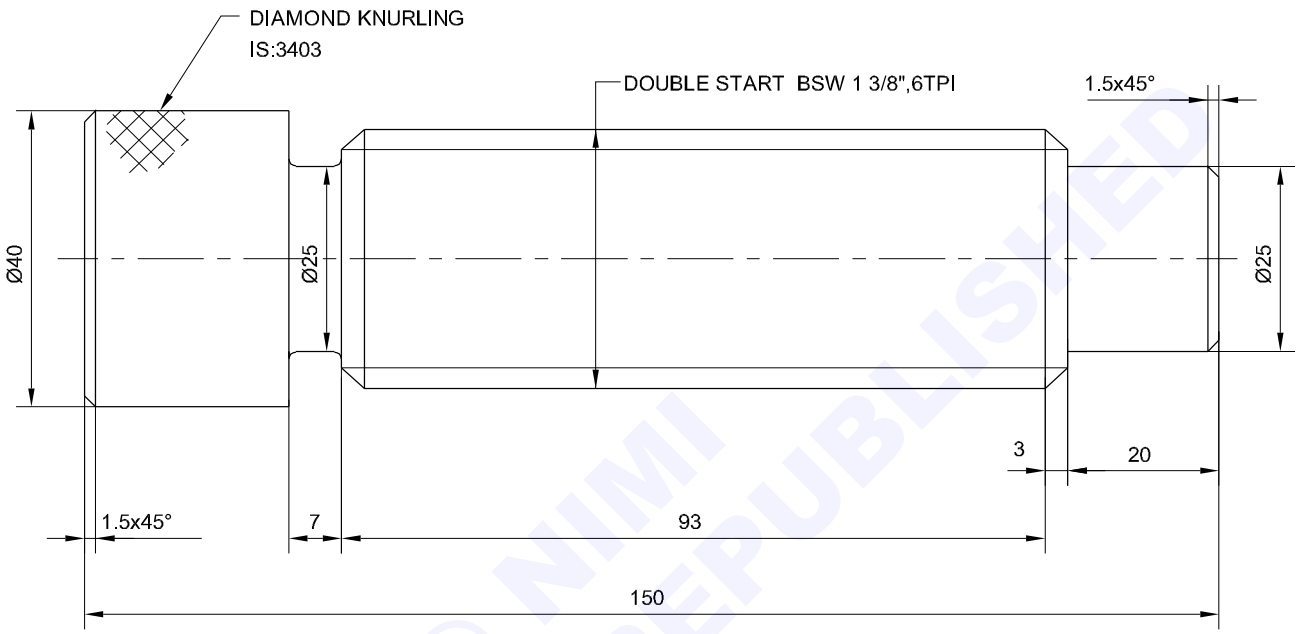
- Check the raw material.
- On 4 jaw chuck in different setting make the blank to size as per drawing.
- Mark the centre of bush and circle and punch.
- Align bush circle to lathe axis.
- Drill and bush to dia 13.1 mm for threading 5/8 BSW (11 TP1) LH of Task 3 and dia 22.85 mm for threading & BSW (8 TPI)
- Make threads and suit male part (Task 1)
- Remove, deburr and assemble part (Task 1,2,3) as per drawing.

– – – – – – –

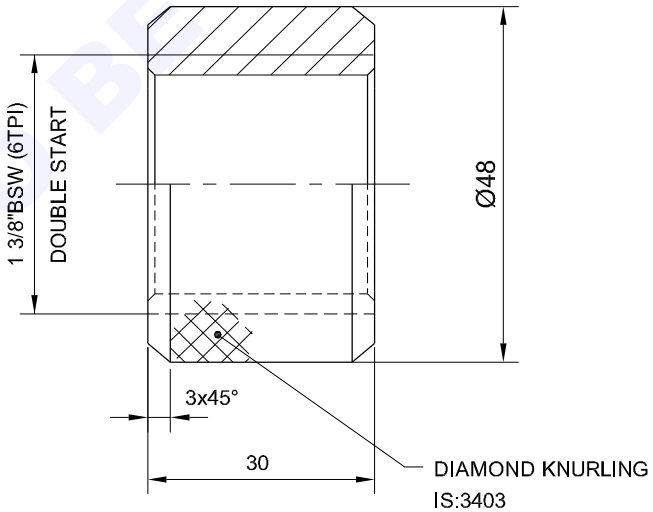
Multi start thread cutting BSW (external & internal)

Objectives: At the end of this exercise you shall be able to
- **cut an external double start acme thread**
- **cut an internal double start acme thread and match.**

TASK-1



TASK-2



1	Ø50 - 55	-	Fe 310	-	TASK-2	2.5.107
1	Ø45 - 160	-	Fe 310 - O	-	TASK-1	2.5.107
NO.OFF	STOCK SIZE	SEMI PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.

SCALE 1:1	**MULTI START THREAD CUTTING BSW (EXTERNAL & INTERNAL)**	DEVIATIONS IS:2102 (M)	TIME
		CODE NO. TU20N25107E1	

Job Sequence

TASK 1 : **External thread**

- Hold the job in a 4-jaw chuck and face the ends, and maintain a total length of 150 mm.
- Centre drill the ends.
- Hold the job between the chuck and the centre.
- Turn Ø $40^{+0.5}$ mm for possible length, and step turn 1 3/8" + 1 mm for 120 mm length.
- Form the radius groove for 7 mm width and maintain Ø25 mm.
- Step turn Ø 25 mm for 20 mm length.
- Chemfer the ends.
- Reverse and hold the job on 1 3/8" + 1 mm and true.
- Finish turn Ø 40 mm, with the work supported with dead centre.
- Knurl on Ø 40 mm and chamfer the edge 1.5 x 45°.
- Reverse and hold on knurled dia. true and support with dead centre.
- Finish turn Ø 1 3/8 mm to cut the thread.
- Make chamfer of 3 x 45°.
- Arrange the gear train for 4 TP1 pitch thread.
- Arrange the thread chasing dial.
- Hold the BSW thread cutting tool in the tool post to centre height.
- Swivel the compound slide 27.5° to the right from the vertical position, and set the tool at right angles to the work axis.
- Cut the external BSW thread by the half included angle method for first start.
- Index the tool for second start & finish the thread.
- Deburr sharp corners.
- Check the formation with screw pitch gauge & dimension with screw thread micrometer.

TASK 2 : **Internal thread**

- Hold the job in 4 jaw chuck projecting 35 mm.
- True it by using surface gauge.
- Set the facing tool & face the end.
- Set RH Turning tool & turn to dia $48^{-0.2}$ for a maximum length.
- Chamfer the end to 3 x 45°.
- Form knurling & maintain dia 48 mm for maximum possible length.
- Centre drill the end.
- By drilling & boring maintain internal dia equal to the root dia of thread.
- Chamfer the end to 3 x 45°.
- Set the thread chasing dial.
- Cut first start of thread.
- Index the thread chasing dial & cut second start.
- Check for the formation by assembling with Part A (Task 1)
- If any variation, again give cuts by indexing.
- Finally match wtih Task 1 job.
- Set the parting tool & part off to a length of $30^{+0.2}$ mm.
- Set the joh & face it to maintain 30 mm length.
- Chamfer the end to 3 x 45° (Both external & internal)
- Deburr all sharp corners.

Skill Sequence

Cutting external 'V' thread by half angle method

Objective: This shall help you to

- **cut external 'V' thread by the half angle method.**

The plunge cut method of cutting 'V' thread is limited for fine pitch threads where the depth of cut is less, and the successive cuts to be given are comparatively less. Further in the plunge cut method, the tip of the tool as well as both the flanks will be removing material from the workpiece. This causes heavier load on the cutting tool's cutting face which may result in the breaking of the tool tip, damaged thread and reduction in the tool life. To overcome these defects, the half angle method is adopted to cut threads.

The steps required are given here.

Check the diameter of the workpiece to be threaded by referring to the drawing.

To provide thread clearance, it is good practice to turn the diameter of the workpiece undersize depending upon the pitch.

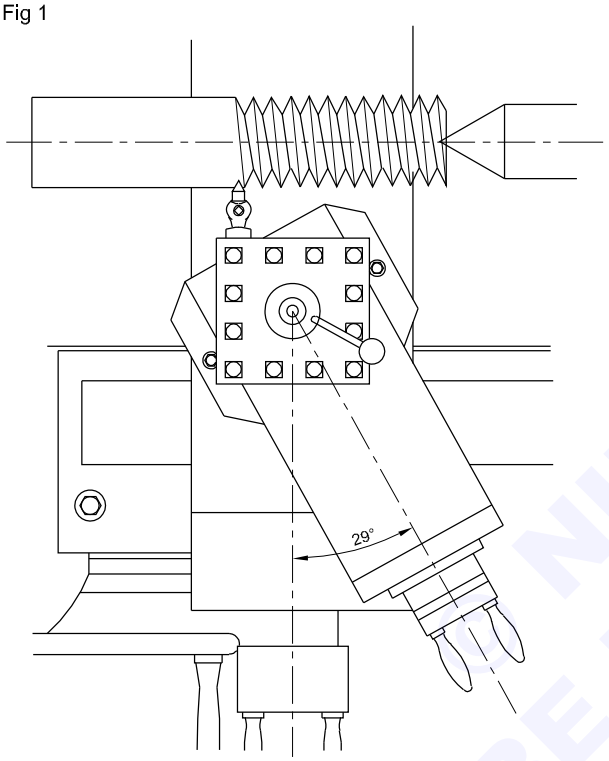
Set the lathe spindle speed to about one fourth of the turning speed.

Set the gearbox according to the pitch of thread to be cut.

Swivel the compound slide to 90° from the horizontal position to bring it in line with the cross-slide.

Swivel to the right 1° less than the half included angle of the thread if it is a right hand thread. (Fig 1)

Fig 1



COMPOUND REST SET AT 29° TO THE RIGHT TO CUT A 60° INCLUDED ANGLE RIGHT HAND THREAD

The angle to which the compound rest is set affects the cutting action of the cutting tool by producing a shearing action on the trailing edge of the tool. This produces a smooth cut.

Set the tool in the tool post with a minimum overhang perpendicular to the axis and also set with a centre gauge. (Fig 2)

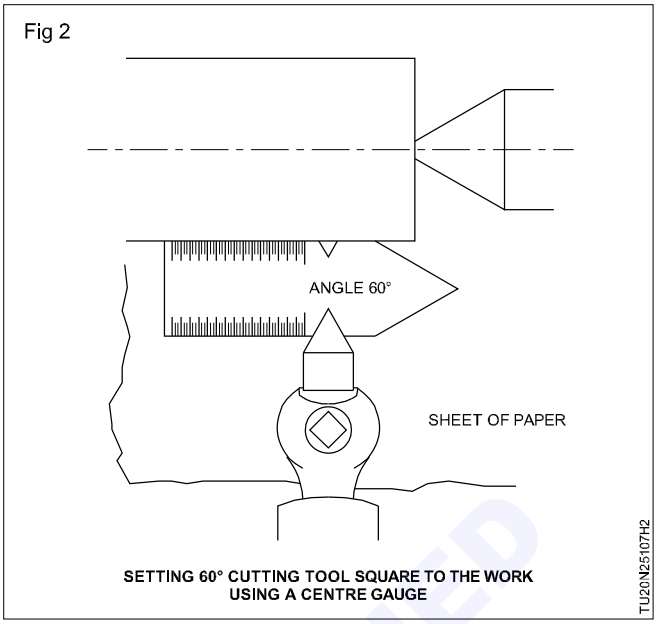
Mark out the length of the workpiece to be threaded.

Chamfer the end of the workpiece surface with the leading edge of the cutting tool to a depth, just greater than the minor diameter of the thread to be cut.

Advance the cutting tool to the work surface by operating the cross-slide hand wheel.

When the tip of the tool just touches the work surface, stop further advancement and set the cross-slide and compound slide graduated collars to zero.

Fig 2



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

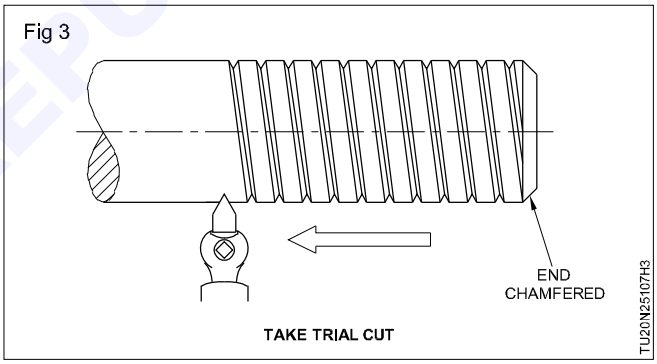
Move the carriage to the right until the end of the tool clears the work.

Feed the tool in about 0.1 mm using the top slide hand wheel.

Engage the half nut referring to the chasing dial.

Take a trial cut along the workpiece to be threaded. (Fig 3)

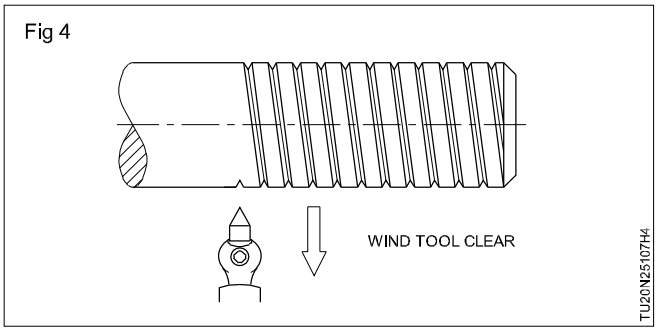
Fig 3



TAKE TRIAL CUT

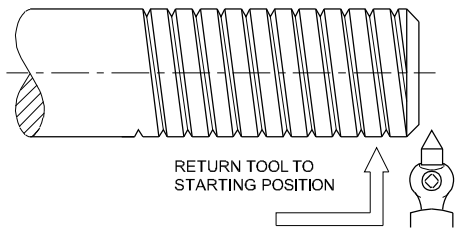
At the end of the trial cut, withdraw the tool immediately, winding it clear off the workpiece by operating the cross-slide hand wheel and simultaneously reversing the machine. (Fig 4)

Fig 4



Allow the carriage to move to the right till it is cleared from the end of the work, and stop the machine. (Fig 5)

Fig 5



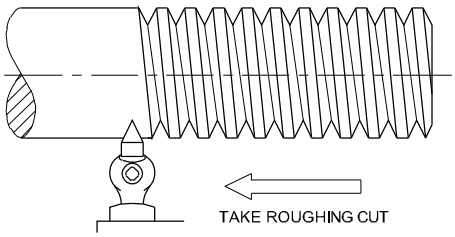
Check the thread formation with a pitch gauge.

Advance the tool by the cross-slide hand wheel till zero position.

Give depth of cut with the top slide handle.

Start the machine and allow the tool to cut the thread. (Fig 6)

Fig 6

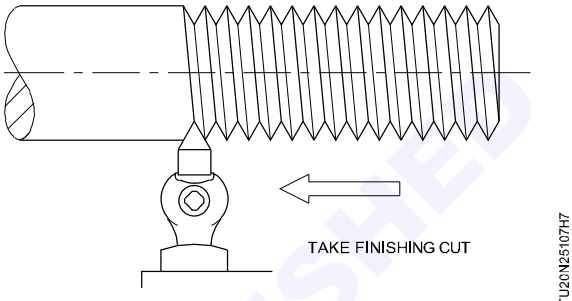


Use plenty of coolant during threading.

Repeat the steps till the required depth is reached. (Fig 7)

At the end of each cut, the tool is withdrawn from the work by the cross-slide hand wheel, and the carriage is brought to the starting point. The cross-slide hand wheel is brought to zero position and a depth of cut is given by the top slide.

Fig 7



Cutting internal 'V' thread by half angle method

Objective: This shall help you to

- **cut internal metric R.H. 'V' thread by the half angle method.**

Internal 'V' threads are generally cut by the plunge cut method even for coarse pitch threads. As in the case of external 'V' thread cutting, the internal 'V' thread tool also has to undergo heavy cutting load during the internal threading operation. But as the tendency of the tool is to spring, the breakage of the tool is greatly avoided. To prolong the tool life and to have a better thread finish even for internal threading, the half angle method is followed.

Following is the procedure for internal threading in a blind hole.

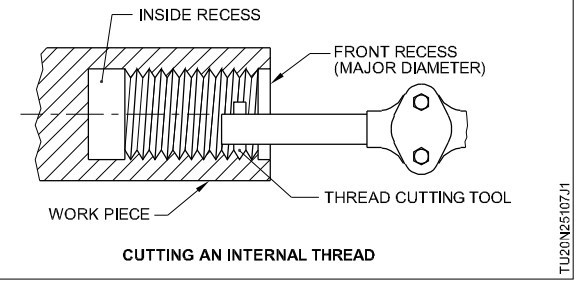
The work is drilled and bored to the core diameter of the thread.

An internal recess is formed at the end of the blind hole. The hole itself must be sufficiently long enough to clear off the boring bar. The width of the recess formed must be wide enough to allow the cutting tool to clear off the thread and the diameter of the recess must be larger than the major diameter of the thread.

The width and diameter of the recess will generally be given in the drawings, but in case it is not, it is left to the discretion of the operator himself to determine them. (Fig 1)

Counterbore the outer end of the hole for a length of 2 mm and a diameter equal to the major diameter of the thread.

Fig 1



CUTTING AN INTERNAL THREAD

This is possible only if the total length of the work is sufficiently more than the finished length.

Swivel the compound slide 29° to the vertical towards left for cutting a right hand thread. (Fig 2)

Arrange the quick change gearbox levers for the pitch of the thread to be cut.

Mount the boring bar with the 'V' thread tool in the tool post, parallel to the lathe axis, and set the point of the cutting tool to the centre and with the centre gauge. (Fig 3)

Mark on the boring bar an indication for the required entry into the bore to cover up the thread length.

Fig 2

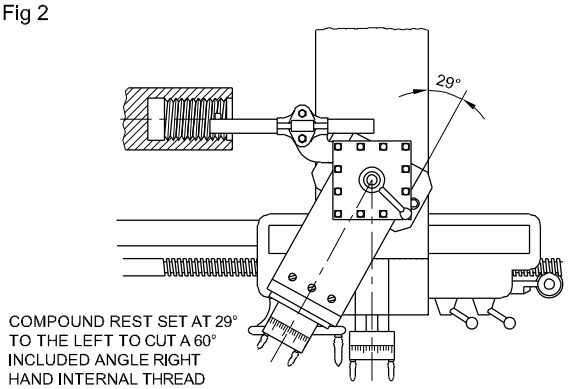
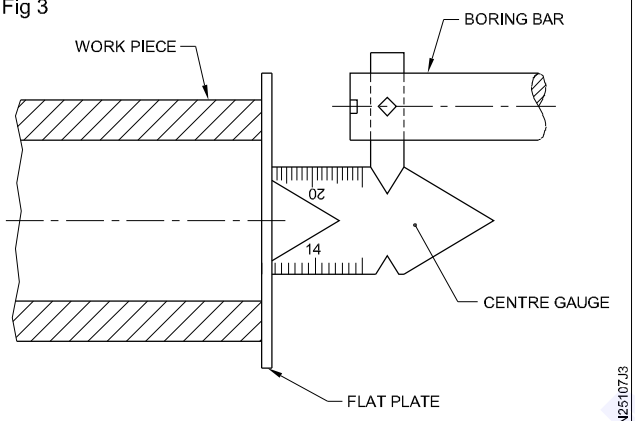


Fig 3



SQUARING THE CUTTING TOOL WITH A CENTRE GAUGE

Add half the recess width to the length of the thread to be cut, and then mark this distance on the boring bar back from the centre line of the cutting tool.

In-feed the point of the cutting tool until it lightly touches the work surface at the outside end of the bore.

Set the cross-slide and the top slide graduated collars to zero.

Position the tool to clear off from the work.

Give a depth of cut by the top slide hand wheel by rotating the handle anticlockwise.

Engage the half nut and allow the tool to have the trial cut.

When the tool has entered up to the scribed mark on the bar, release the tool by rotating the cross-slide hand wheel clockwise, simultaneously reversing the machine to make the carriage move to the starting point.

Rotate the cross-slide hand wheel in the anticlockwise direction to bring to zero mark.

Give a further depth of cut by the top slide hand wheel rotating in the anticlockwise direction.

Give successive cuts and finish the thread.

The last depth of cut should make the tool tip to just touch the counterbored hole at the mouth of the hole.

Repeat the same last cut twice or thrice.

Check with the mating part for the fit.

Face off the counterbored end, if sufficient material is there and chamfer to match.

Double start thread by thread chasing dial

Objective: This shall help you to

- **cut multi-start threads using thread chasing dials.**

This may be used to produce 2, 4 and 8 starts mostly with odd numbered leads such as 1,3,5,7 etc. or fractional leads such as 1/1.5. (Fig 1)

The steps for cutting double start threads of 1/7" pitch on a lathe having 4 TPI lead screw are as follows.

Set the lathe for cutting a screw thread of the given lead.

Mount the workpiece.

Engage the carriage half nut on a numbered line on the thread chasing dial.

Make the first cut of the first start threads and return the cutting tool to the starting position of the threads.

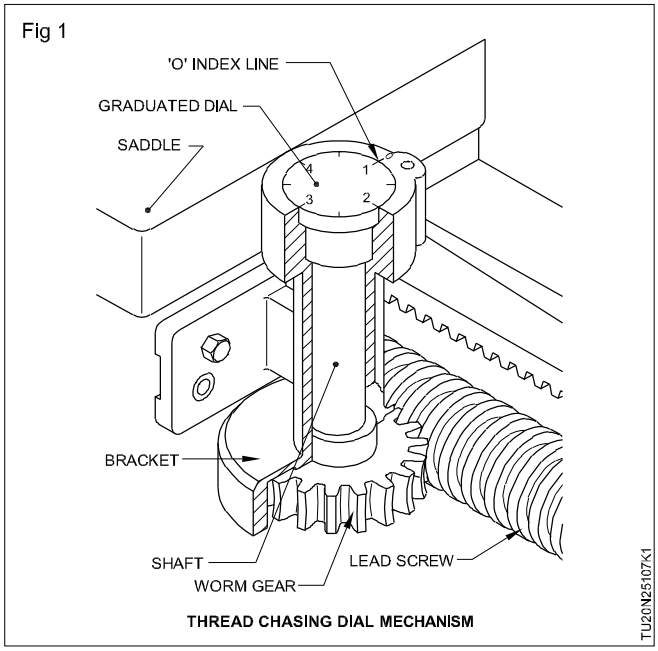
Set the cross-slide graduated collar to zero.

Engage the carriage half nut on an unnumbered line on the thread chasing dial and take the first cut of the second start threads.

Engagement of the half nut at this position automatically centres the cutting tool half way between the path of the first start threads.

Give the same depth of cut for both the starts alternatively till the full depth is reached.

Fig 1

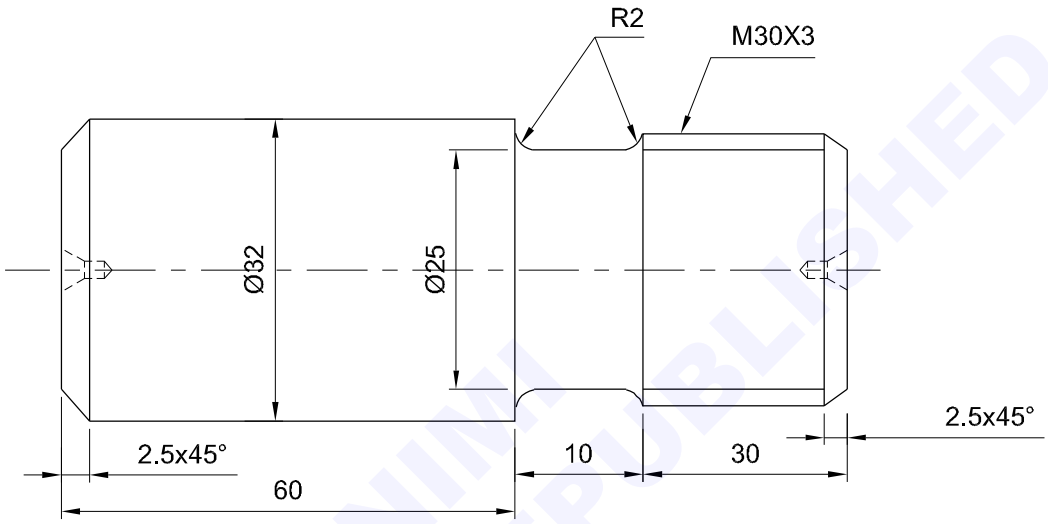


THREAD CHASING DIAL MECHANISM

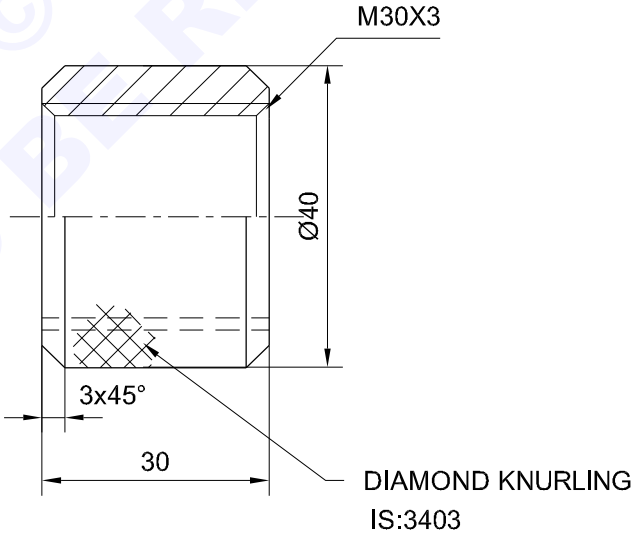
Multi start thread cutting (metric) (external & internal)

Objectives: At the end of this exercise you shall be able to
- **holding and by using the job on 4 jaw chuck**
- **set the job in between centre**
- **set the face plate for cutting multi start thread.**

TASK-1



TASK-2



NO.OFF	STOCK SIZE	SEMI PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.
1	Ø45 - 35	-	Fe 310	-	TASK-2	2.5.108
1	Ø45 - 105	-	Fe 310	-	TASK-1	2.5.108

SCALE NTS	MULTI START THREAD CUTTING (METRIC) (EXTERNAL & INTERNAL)	TOLERANCE	TIME
		CODE NO. TU20N25108E1	

Job Sequence

Part A - External thread

- Hold the job in a four jaw chuck and face both the ends maintain total length of 100 mm.
- Centre drill at the ends and hold the job between centres using a drilling plate having slots milled at 180° and a bent tail carrier.
- Turn the outer dia 32 mm to full length and chamfer the ends.
- Form the groove Ø 25 x 7 mm leaving 30 mm distance from the end and form the radius 2 mm.
- Arrange the gear train to cut 6 mm lead threads on the job.
- Cut the first start thread to half the depth.
- Change the setting the bent tail of the dog carries with opposite slot of the face plate.
- Cut the 2nd start thread to full depth by giving successive cuts.
- Reset the job for completing the 1st start threads to full depth.
- Cut the first start threads to full depth by giving successive cuts.

Part B - Internal thread

- Hold the job in four jaw chuck projecting 35 mm and true it.
- Face the end and turn to Ø $40^{-0.2}$ for maximum length.
- Make centre drill.
- By drilling and boring maintain internal dia equal to the root dia of thread.
- Chamfer the ends 3 x 45° for both external and internal.
- Set the internal threading tool and set the change gears to cut a feed of 6 mm (3mm pitch of double start)
- Cut the first start thread to half the depth.
- Second start thread is to be cut by indexing the thread chasing dial.
- Cut the thread to full depth by giving successive cuts.
- Again reset by indexing to the first start finish the thread full depth.
- Match the job with Part A (Task 1).
- Deburr all the sharp corner.
- Set the knurling tool and for knurling to maintain dia 40 mm for 30 mm length.
- Set the parting tool and part off to a length of $30^{+0.2}$ mm.
- Hold the job and face to maintain a length of 30 mm.
- Remove the sharp corner if any.
- Assemble both external and internal.

Skill Sequence

Multi - start thread cutting using slotted face plate

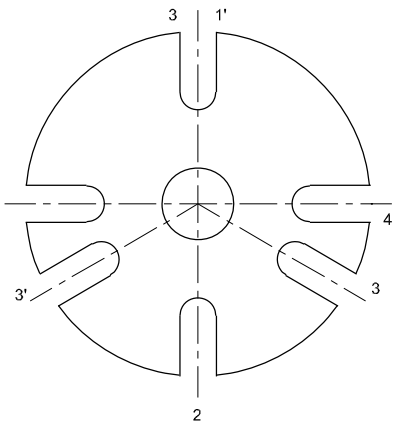
Objective: This shall help you to
- **cut multi-start thread using slottted face plate.**

Using slotted face plate (Fig 1)

A slotted face plate (illustrated) is used to cut threads of 2 starts, 3 starts, 4 strats etc.

Slots are provided on the face plate at convenient distances. Two opposite slots to cut double start thread, 3 slots 120 degree apart to cut 3 start thread and 4 slots 90 degree apart to cut 4 start thread, and so on and so forth, are provided.

Fig 1



FACE PLATE WITH SLOTS

Multi - start thread cutting by dividing 1st driver of change gear train

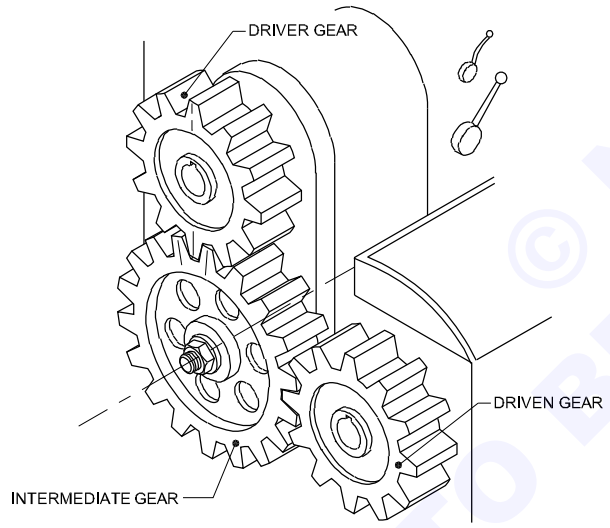
Objective: This shall help you to
- **arrange to gear for multi start thread**
- **cut multi-start thread by dividing the first driver method.**

Dividing the first driver method

As regards the gear train it becomes necessary to arrange the layout so that the first driver is a multiple of the number of starts required. Thus for a double start thread, the gear teeth must be divisible by two.

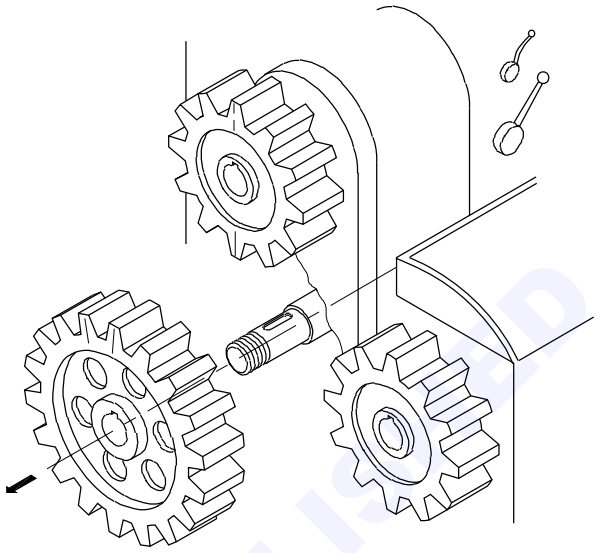
After finishing the first start, the lathe is stopped. One tooth of the 1st driver and the space of the first driven gear in which it is seating are marked. By counting the number of teeth from the marked tooth of the 1st driver, make another mark on the tooth which is exactly 180° away. Loosen the swing plate and disengage the idler gear from the 1st drier. Rotate the spindle by hand to bring the second mark of the first driver to mesh in the previously marked space of the 1st driven gear. The lathe is now ready for cutting the 2nd start. This procedure is applicable to cut threads of more than two starts also. Figs 1&2 illustrate marking on change gears.

Fig 1



GEAR TEETH MARKED FOR INDEXING

Fig 2

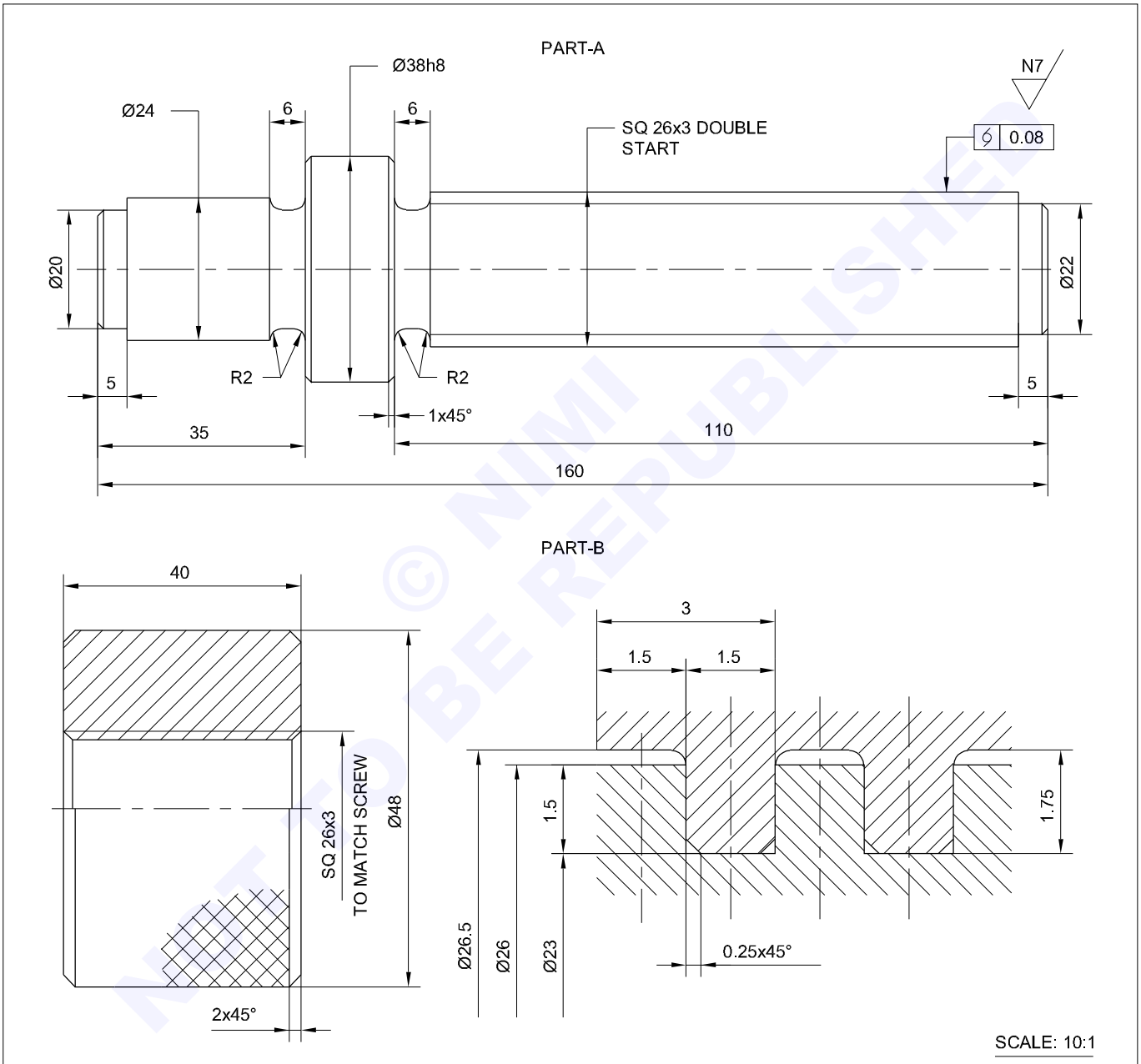


DISENGAGE THE INTERMEDIATE GEAR

Multi start square thread cutting (male & female)

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

- **set and index the driving gear for cutting double start square threads**
- **grind the correct width of tool required as per the pitch for cutting double start square threads, external and internal**
- **cut internal double start square threads and match the male part.**



1	Ø63 - 45		Cu SnP7 IS:28		TASK-2	2.5.109
1	Ø40 - 165		Cu6 IS:2073		TASK-1	2.5.109
NO.OFF	STOCK SIZE	SEMI PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.

SCALE 1:1	**MULTI START SQUARE THREAD CUTTING (MALE & FEMALE)**	DEVIATIONS IS:2102 (M)	TIME
		CODE NO. TU20N25109E1	

Job Sequence

TASK 1: **Plain turning and double start square threading (External)**

Part - A

- Hold the job in a four-jaw chuck, true and face both ends and maintain a total length of 160 mm.
- Centre drill both ends.
- Remove the four-jaw chuck and mount a driving plate.
- Insert the headstock centre with the spindle sleeve and the tailstock centre.
- Hold the job in a suitable carrier in between centres.
- Turn dia. 38 h8 to a length of 125 mm.
- Turn Ø 24 mm to a length of 35 mm.
- Turn Ø 20 mm to a length of 5 mm.
- Finish radius groove R2 to a length of 6 mm at 35 mm length.
- Finish chamfer 1 x 45° at the end of 20 mm dia.
- Reverse the job and hold in a suitable carrier with soft metal packing.
- Turn Ø26 – 0.08 to a length of 110 mm.
- Turn Ø 22 mm to a length of 5 mm.
- Chamfer 1 x 45° at the end of Ø 22 mm. Chamfer 1 x 45° at the collar on Ø 38 h8 on both sides.
- Undercut to core dia. for 6 mm width.
- Set the square threading tool, to correct centre height.
- Cut the first start to a length of 99 mm.
- Index the driving gear exactly for cutting the 2nd start.
- Cut the second start to a length of 99 mm.

TASK 2: **Bush turning and Internal square threading**

Part - B

- Hold the job in a four-jaw chuck, true and face both ends maintaining a total length of 40 mm.
- Drill Ø18 mm through hole.
- Bore the drilled hole to the size of the core diameter.
- Grind the internal square threading tool to a width of ½ x pitch + 0.02 mm.
- Set the internal square threading tool to correct centre height.
- Cut the first start through.
- Cut the second start through.
- Match the job with the male piece.
- Hold the male and female pieces assembled together in between centres.
- Turn Ø 60 mm to a length of 40 mm.
- Diamond knurl to a length of 40 mm.
- Chamfer 2 x 45° at both ends.

Skill Sequence

Square thread

Objective: This shall help you to
- **cut a square thread.**

Check the diameter of the portion to be threaded as indicated in the drawing.

Chamfer the edge of the job to 2 x 45 degree.

Set up the gearbox to cut the required pitch of thread, and hand of thread.

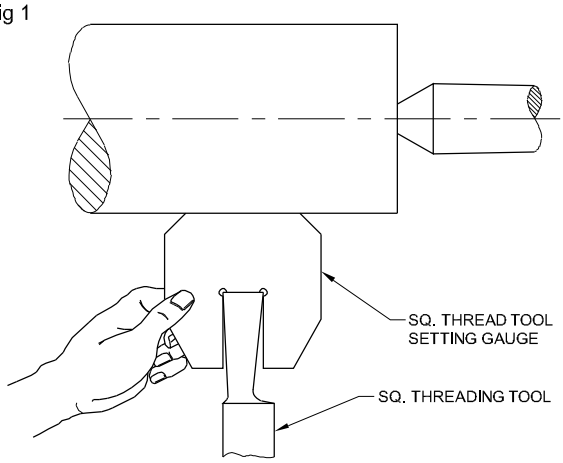
Set the square threading (roughing) tool in the tool post with its height to the centre height of the lathe.

Set the tool square to the axis of the job with the square thread gauge. (Figs 1&2)

Care is to be taken to prevent overhanging of tool from the holder to avoid chatter.

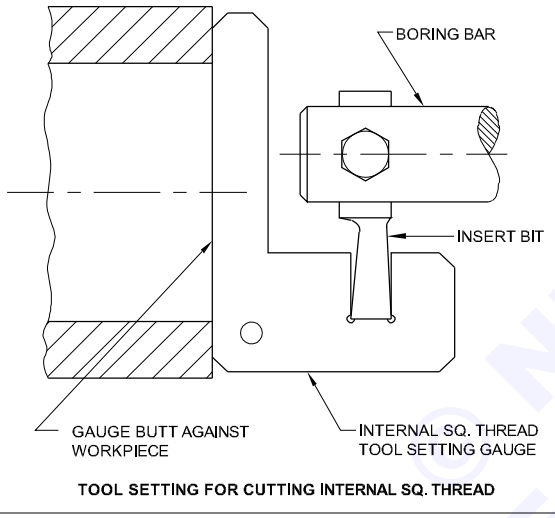
Set the spindle speed about 1/4th of normal turning.

Fig 1



TOOL SETTING FOR
CUTTING EXTERNAL SQUARE THREAD

Fig 2



TOOL SETTING FOR CUTTING INTERNAL SQ. THREAD

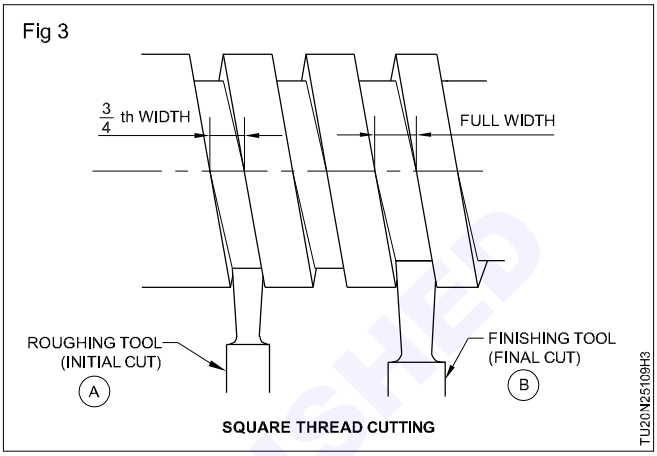
Cut the square thread, by repeated successive depth of cuts to reach 3/4th width and depth of thread. (Fig 3)

Set the finishing square thread tool for finishing cuts.

Cut the square thread to full depth and full width by taking successive cuts to complete the required thread form.

Deburr and check the square thread using a square thread gauge.

Fig 3



SQUARE THREAD CUTTING

Grind an external threading tool

Objective: This shall be able to
- **grind an external square threading tool.**

Determine width and angles required for grinding the external square threading tool.

The side clearance of the square threading tool is of prime importance to prevent the tool from interfering or rubbing against the vertical flank of the thread. As a rule, the forward side clearance angle (a1) is determined by adding 1° to the helix angle of the thread and trailing side clearance angle is obtained by subtracting 1° from the helix angle. (Fig 1)

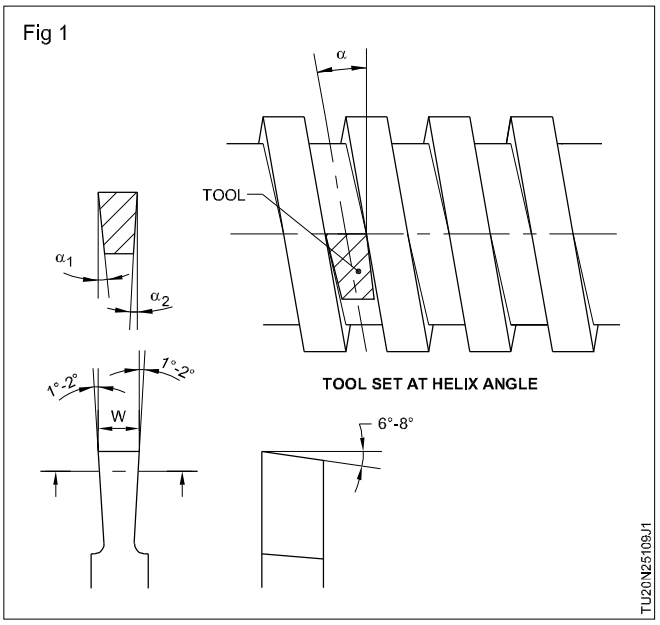
$\alpha 1 = 1° +$ Helix angle of thread

and

$\alpha 2$ = Helix angle of thread - 1° where, helix angle (α)

$$= \frac{\text{Lead of thread}}{\text{axcoredia of thread}}$$

Fig 1

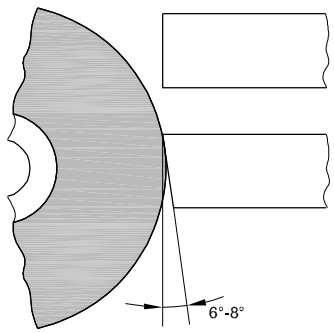


TOOL SET AT HELIX ANGLE

The width of the nose of the square threading tool should be equal to half of the pitch of the square thread to be cut.
W = 0.5 x p

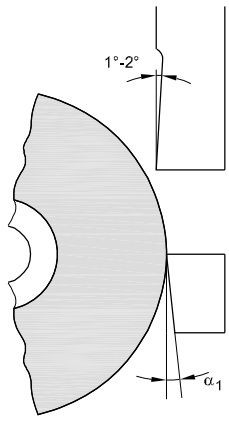
Grind the front flank of the tool to get the front clearance angle 6° to 8°. (Fig 2)

Fig 2



Grind the forward side flank of the tool to obtain side relief angle 1° to 2° and forward side clearance angle α1. (Fig 3)

Fig 3

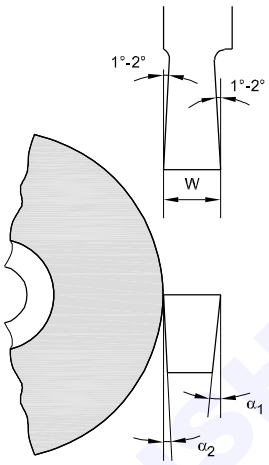


$\alpha 1 = \alpha + (1° \text{ to } 2°)$

Grind the trailing side flank of the tool to maintain the side relief angle 1°-2°, trailing side clearance angle a 2 (a2 = a-1°) and the width W (W=0.5 x p). (Fig 4)

Deburr and check the width of the angles using a vernier caliper and bevel protractor.

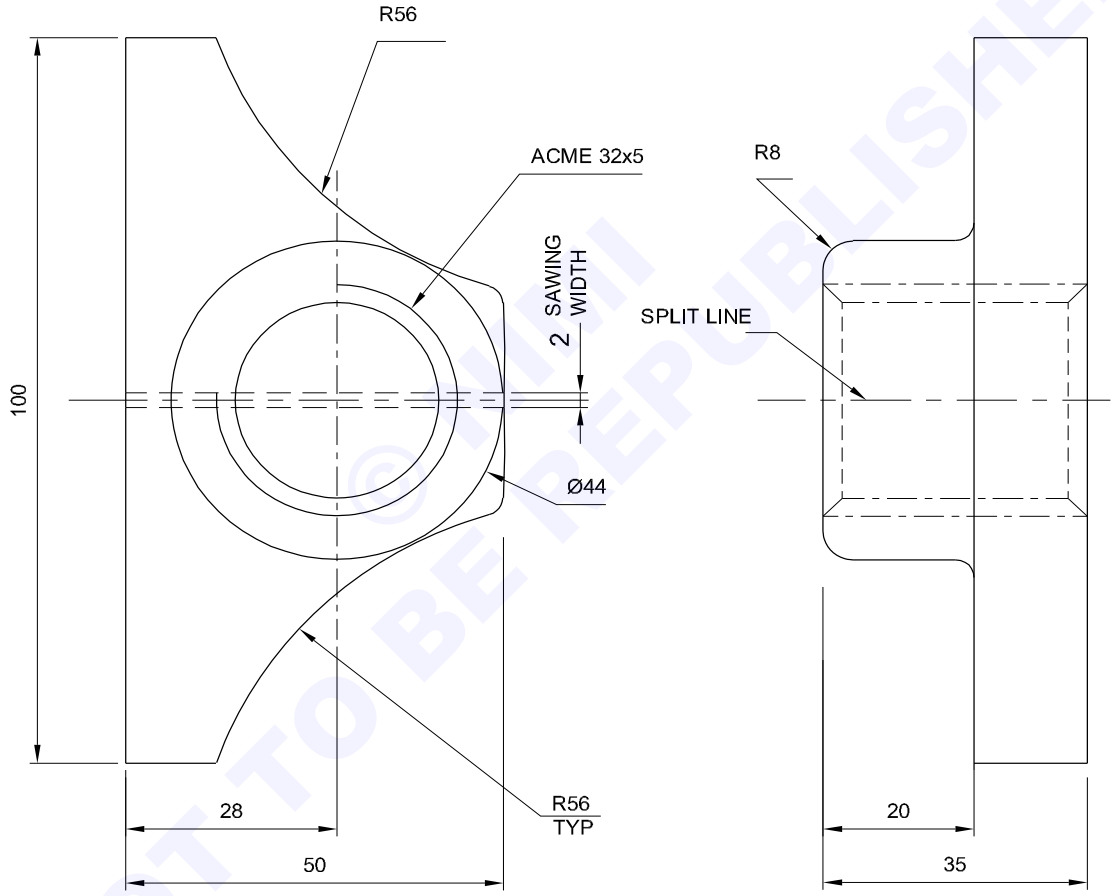
Fig 4



Making half nut as per standard lead screw

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

- **turn eccentric stepped boss and boring**
- **spliting twohalfs and setting on fixture**
- **boring and threading of ACME 32 x 5 internal thread**
- **setting and forming R 56 on both sides.**



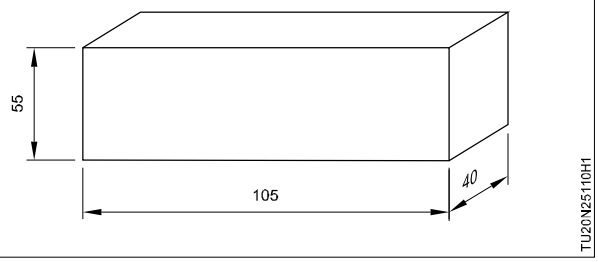
1	40x55x105	-	Fe 310	-	-	2.5.110
NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.

SCALE NTS	MAKE HALF NUT AS PER STANDARD LEAD SCREW	DEVIATIONS IS:2102(M)	TIME:
		CODE NO. TU20N25110E1	

Job Sequence

- Check the raw material to the size of 40 x 55 x 105 and deburr. (Fig 1)

Fig 1



- Fix on 4 jaw chuck, in different setting face all faces one by one and maintain size 35 x 50 x 100 (Fig 2,3,4)

Fig 2

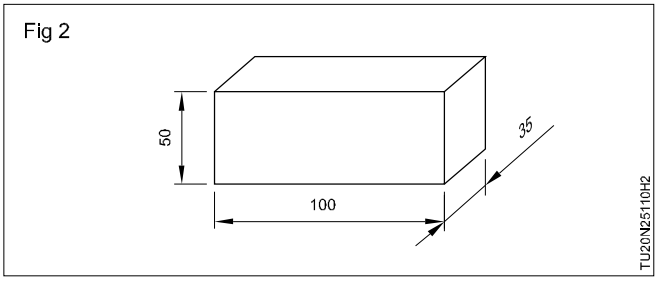


Fig 3

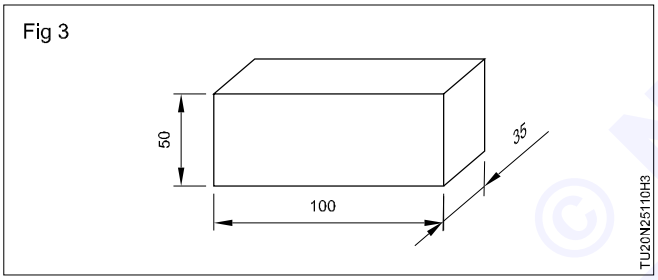
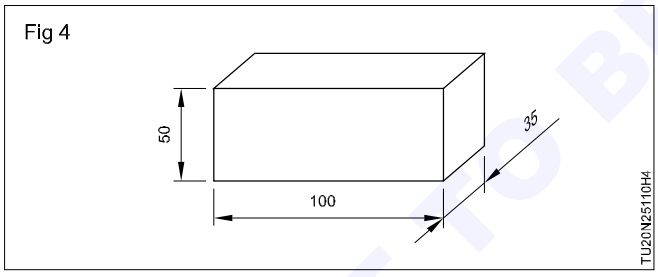


Fig 4



- Deburr corners.
- Draw the shape on paper.
- Paste paper over the blank punch dots over the marked lines of arcs and circles and axis line of bore. Punch dots on Ø 22 circle and axis lines for setting and truing of eccentric bore. (Fig 5)
- Set the job blank on 4 jaw chuck and align circle centre to lathe.
- Turn Ø 44 x 20 and form R 8.
- Drill and bore to dia 22. (Fig 6)
- Remove the job from chuck.

Fig 5

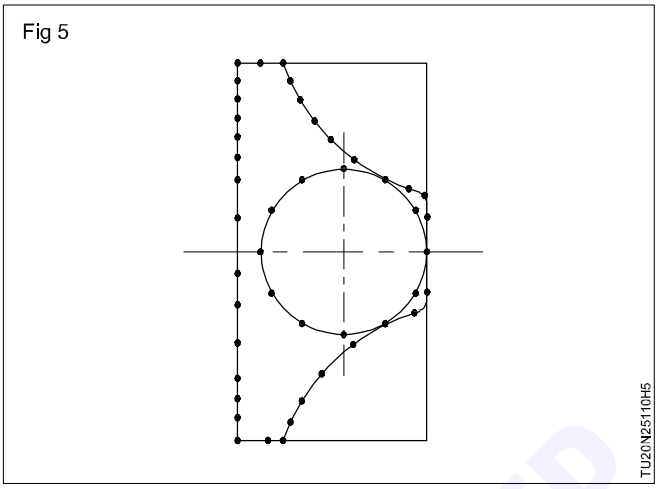
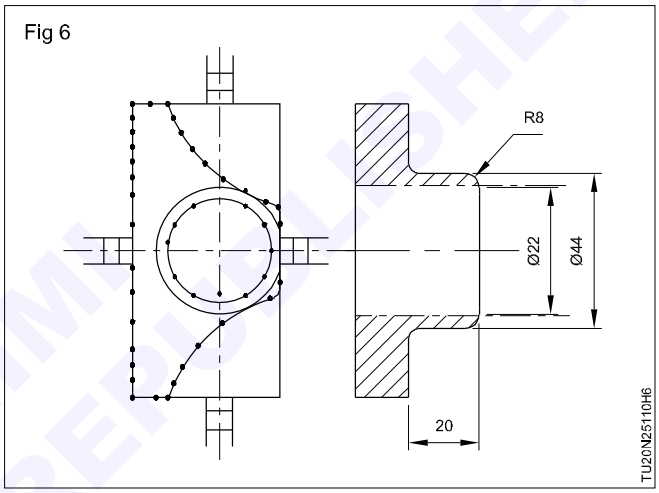
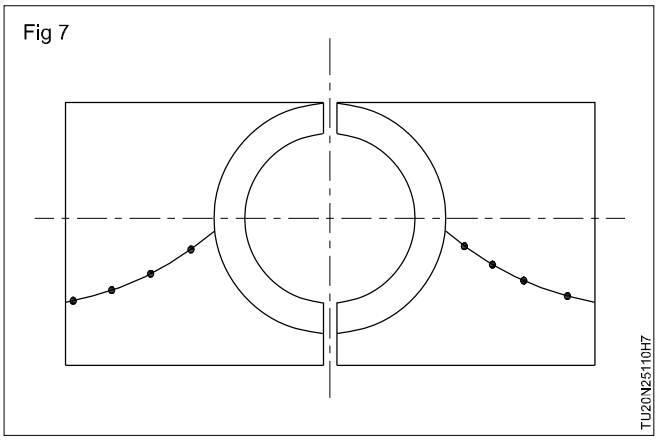


Fig 6



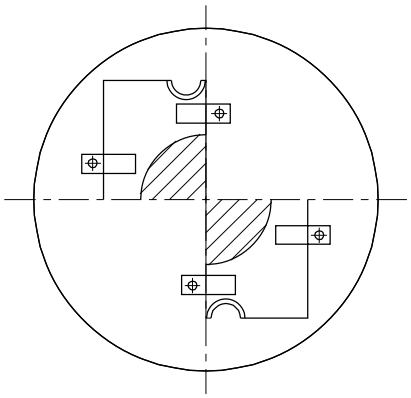
- Slit Saw/Hacksaw into two pieces at bore axis line. (Fig 7)

Fig 7



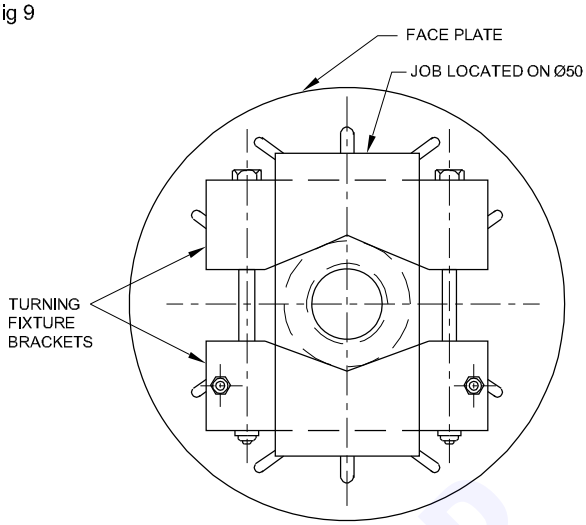
- Fix face plate and clamp two holes opposite to each other and align R 56 radial dot punch mark to lathe axis.
- Bore the form R 56 radius. (Fig 8)
- File flat the mating surface.
- Fix the turning fixture bracket in face plate slot and align 'V' centre to lathe axis.

Fig 8



TU20N25110H8

Fig 9



TU20N25110H9

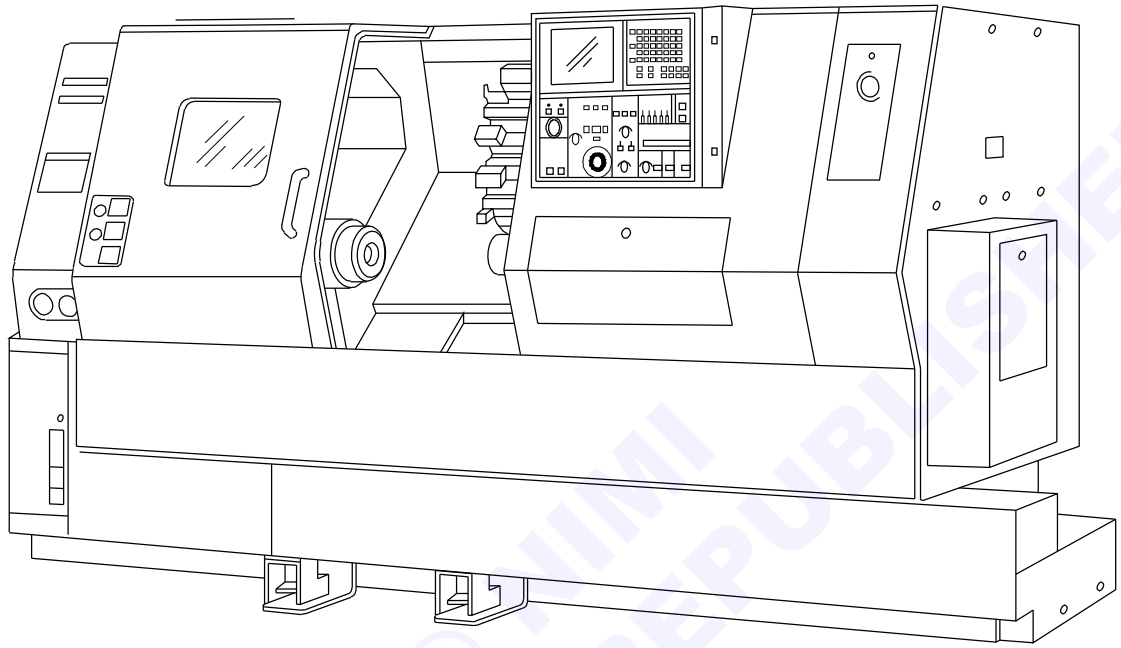
- Clamp, true it to lathe axis. (Fig 9)
- Depth of thread = 1/2 P + 0.01

  = 1/2 x 5 + 0.01 = 2.51
- Bore to diameter = 26.50.
- Set threading tool for ACME thread.
- Cut threads to 2.75 deep.
- Finish threads remove two havlves and deburr.

Personnal and CNC machine safety

Objectives: At the end of this exercise you shall be able to
- **adopt personnal safety in CNC workshop**
- **maintain safety of CNC machine**
- **adopt safety procedures while handling tools, equipments & CNC machine.**

Fig 1



Job Sequence

Safety measures for handling tools and equipments

- Wear appropriate PPE
- Inspect tools and equipments before use
- Dispose old and blunt tools
- Use tools and equipments for their intended prupose only
- Use proper technique and posture while handling tools and equipments
- Unplug tools when not in use
- Ensure cleanlines of all tools and equipments before and after using
- Apply recommeded antirusting agents on tools and equipments beofe storing properly

Safety measures of CNC machines

- A well trained operator should operate the 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.
- Do not operte 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.
- Perform preventive maintenance as per company recommendations.
- Immediately pass the "EMERGENCY STOP" button when ever emergency situation is observed.

Answer the following

1 What will you do, when a malfunction of machine is observed?

a ……

b ……..

2 List the PPE required while operting CNC machine?

a ……

b ……

c ……

d ……

e ……

3 What will you do, if observe a broken tool in turret?

a ……

b ……

4 How do you check the lubrication and hydraulic oil level?

a ……

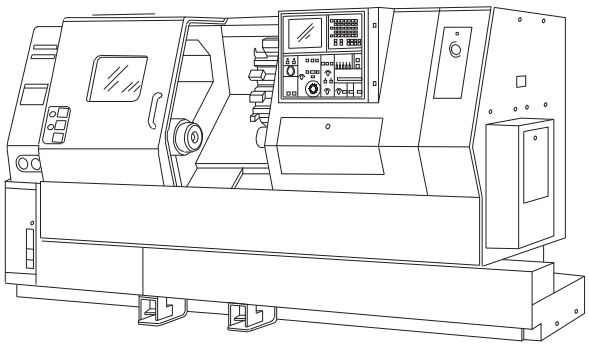
b ……

Identify the CNC machine and CNC console

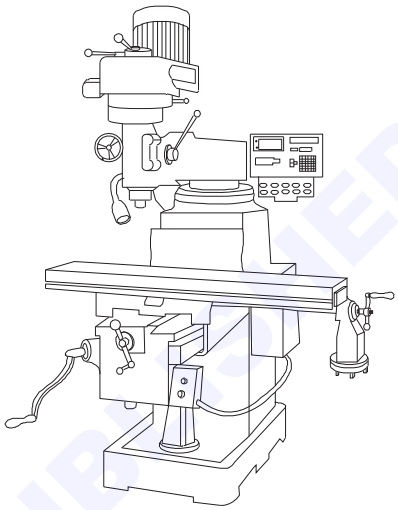
Objectives: At the end of this exercise you shall be able to
- **identify the CNC machine**
- **identify the controllers of CNC machine.**

Fig 1

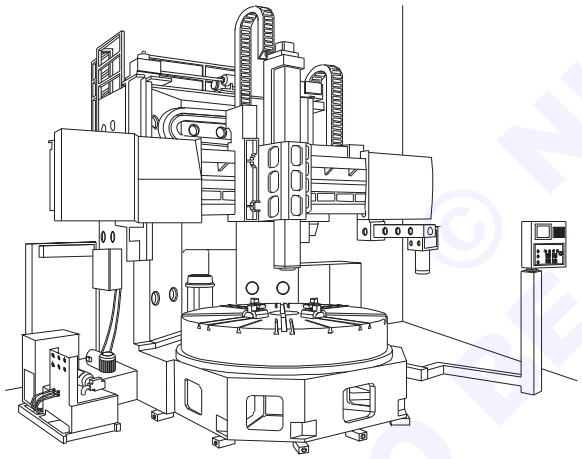
1)



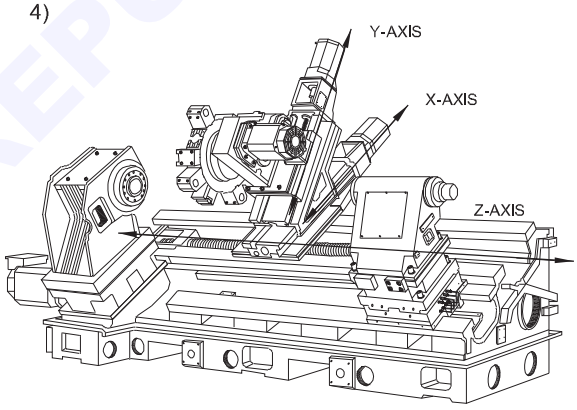
2)



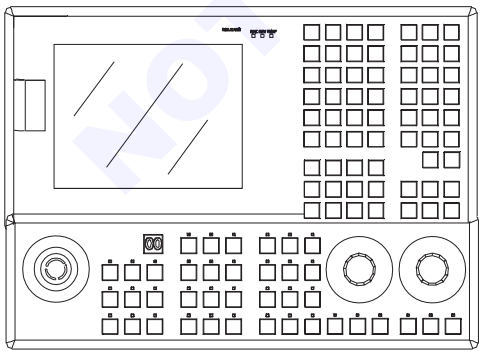
3)



4)



5)



6)

TU20N26112H1

Fanuc controller

A: Key description

Control keyborad, digitizer overlay

Key function

Reset ..... Cacel an alarm, reset the CNC (e.g. Interupt a program), etc

Help ..... Helping menue

Cursor ..... Search function, line updown

Page..... Page updown

Alter ..... Alter word (replace)

Insert ..... Insert word, create new program

Delete ..... Delete (progra, block, word)

EOB ..... End of block

Can ..... Delete input

Input ..... Word input, data input

POS ..... Indicates the current position

Prog ..... Program functions

Offset setting.. Setting and display of offset values, tool and wear data, variables.

System ..... Setting and display of parameter and display of diagnostic data

Messages ..... Alarm and message display

Graph ..... Graphic display.

Instructor to guide the trainees in major identification of CNC machine and in axis control.

Job Sequence

- Identify your CNC machine and its console of axis.
- List the name of CNC machine in Table 1.

Table 1

Sl.No M/C NO	Name of the CNC machine with major identification (Type & axis)
1	
2	
3	
4	

- List the name and number of axis in Table 2.

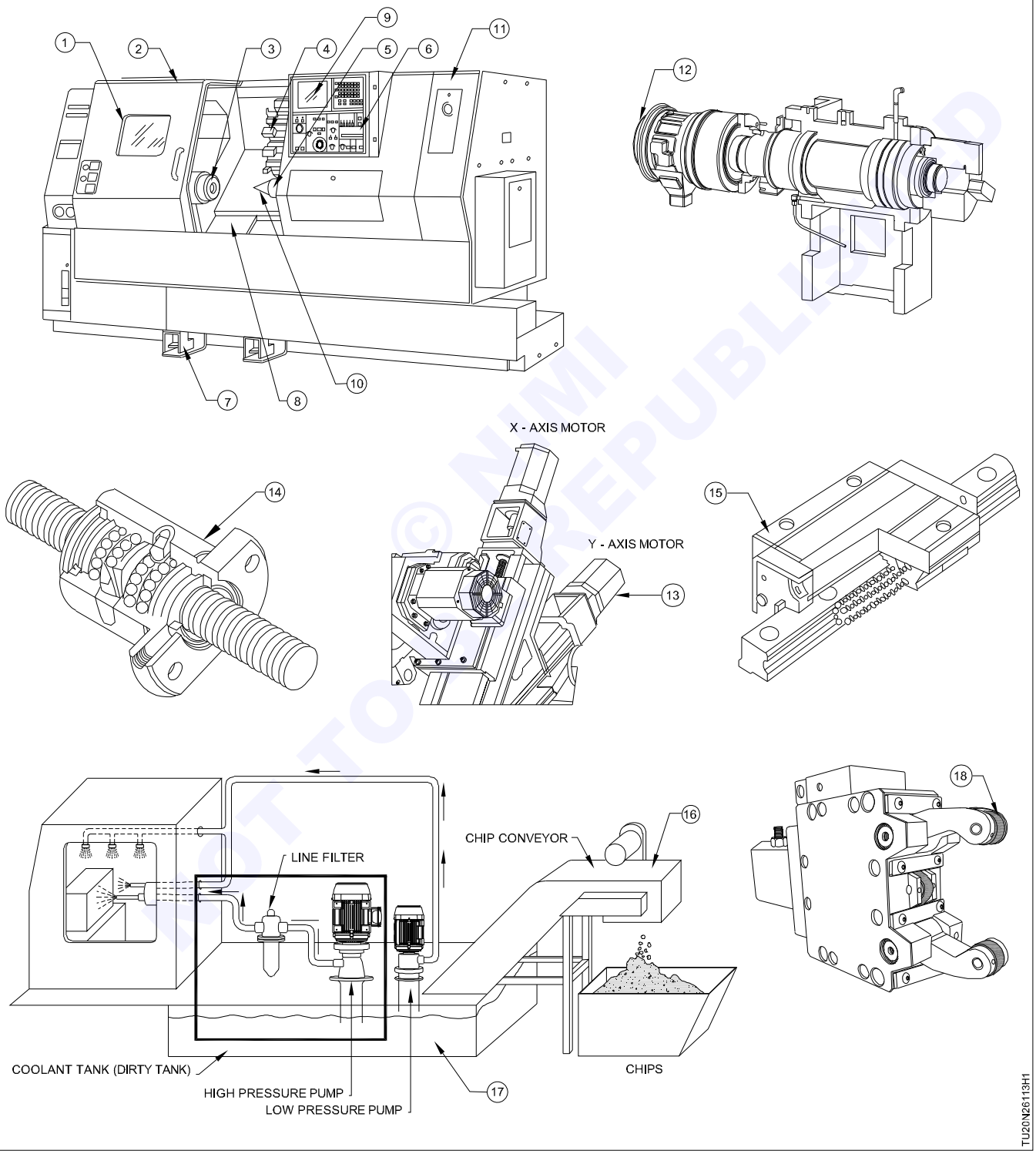
Table 1

Sl.No	Name of the CNC machine	Console and the number of axis
1		
2		
3		
4		

Demonstration of CNC lathe machine and its parts

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
- List out the name of the parts and functions shown in figure in the given table.
- Record it in table.
- Instructor shall demonstrate the parts

Note: Instructor shall demonstrate parts and its function of CNC lathe.

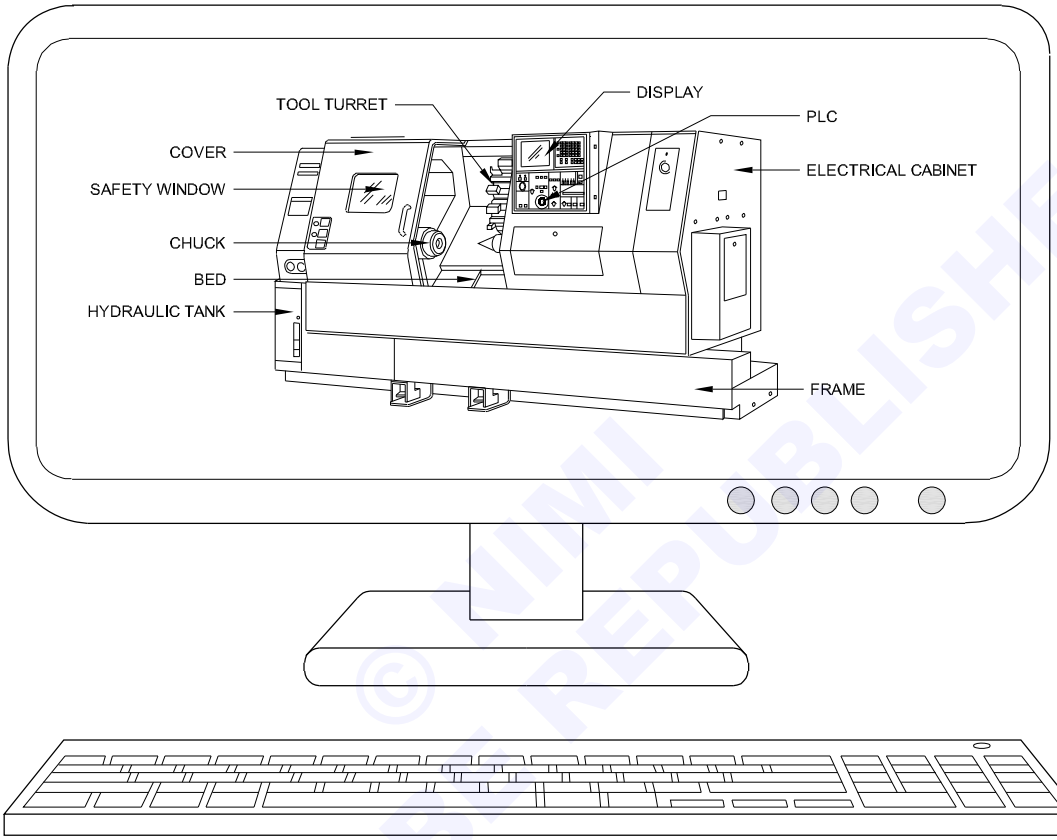
Table 1

Sl.No	Name of the CNC lathe parts
1	
2	
3	
4	
5	
6	
7	
8	
9	
10	
11	
12	
13	
14	
15	
16	
17	
18	

Working of CNC machine parts of 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 simulator.**

Fig 1



Job Sequence

Note: Use appropriate simulator softwares for demonstrating CNC machine parts viz. Sinumeric or Fanuc.

1. **Instructor shall demonstrate the CNC machine and it's function's.**
2. **Trainees should note down the parts and its functions and record it in Table 1.**

Table 1

Sl.No	CNC Parts	Function of part
1		
2		
3		
4		
5		
6		
7		
8		
9		
10		
11		
12		
13		
14		
15		
16		
17		

Get it verified by the Instructor.

Identify machine over travel limits and emergency stop

Objectives: At the end of this exercise you shall be able to
- **identify the over travel limit switches and x and y 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 axes**

There are two types of over travel limit

1 Soft over travel

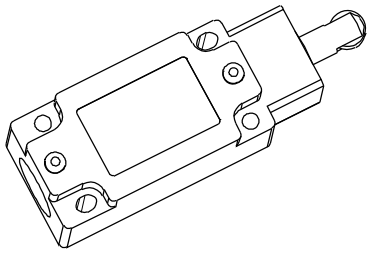
2 Hardware over travel

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

TU20N26115H1

- Position the X axis in the middle of the x and 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.
- Figure 2 shows the x axis dog and limit switch
- Figure 3 shows the z axis dog and limit switch

Fig 2

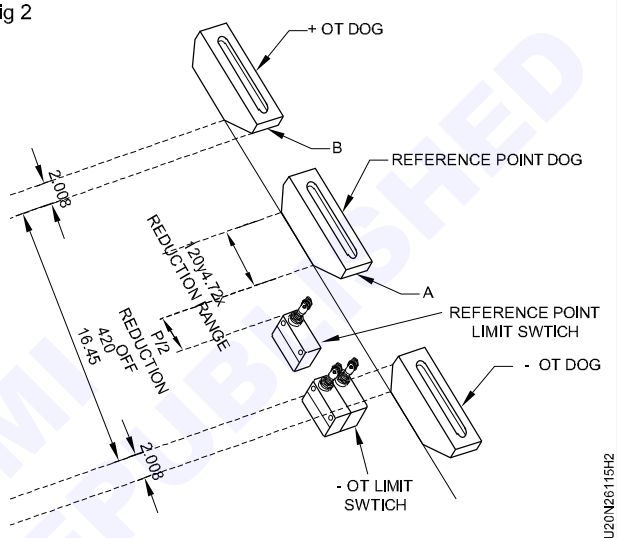
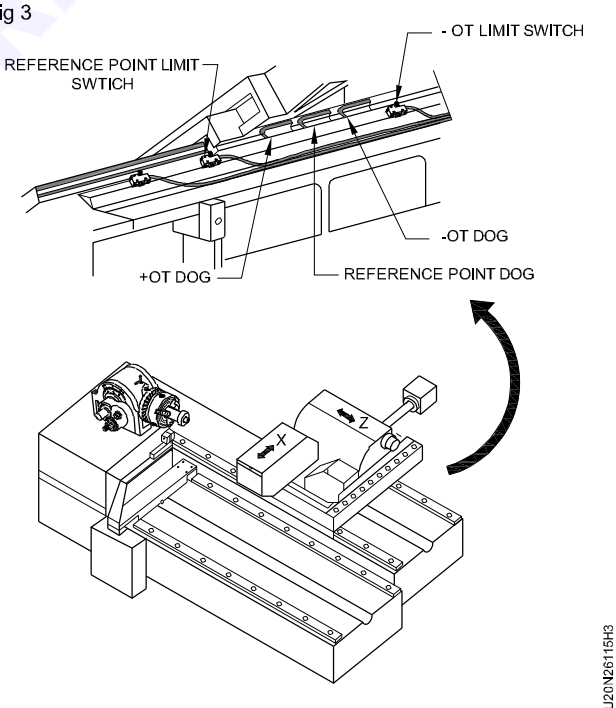


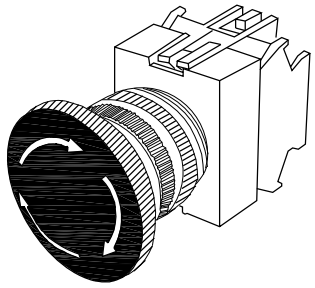
Fig 3



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.

Fig 1



TU20N26115J1

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

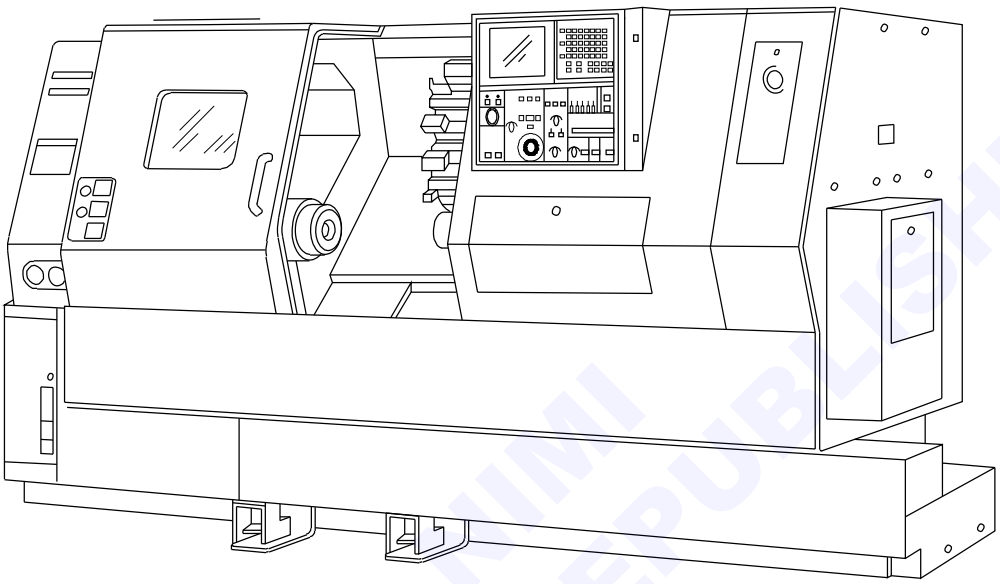
Caution: Do not try to rotate in anticlockwise direction.

— — — — — — — —

Conduct a premilinary check of readiness of the CNC

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



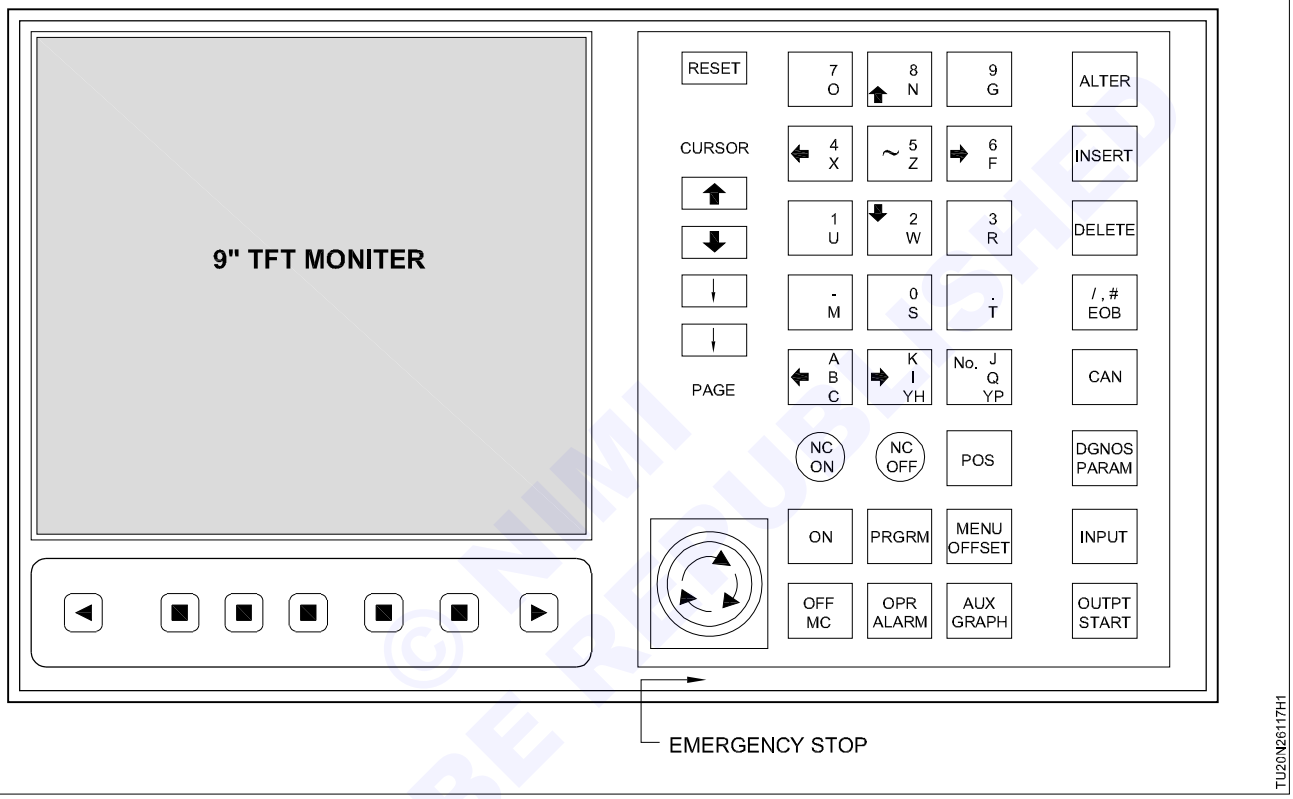
Job Sequence

- 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.

Identification of safety switches and interlocking of DIH (Direct input hand wheel) 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



Job Sequence

1 **Trainer shall demonstate 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/	Start the auto cycle to machine job blocks)	.....

Get it verified by the trainer.

Machine starting & operating in reference point, JOG and incremental mode

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

- **switch ON CNC turning centre**
- **start the machine and operate in reference point**
- **start the machine and operate in JOG mode**
- **start the machine and operate in incremental mode**
- **switch OFF CNC turning centre.**

Job Sequence

CNC machine starting and operating

"Machine ON" procedure (in sinumeric)

Switch ON (main power supply)

↓

Stabilizer ON

↓

Machine main switch ON (Located at the side of the machine)

↓

Release emergency button (By rotating clockwise direction)

↓

N.C. ON (press the green button)

↓

Machine 'ON' (press the green buttin)

To switch 'ON' Display panel

To switch 'ON' Hydraulic & NC

Reference point or homing procedure

Press control key

↓

Select refrence point (By using softkeys and extension key)

↓

Press X + (wait till it reaches to set value)

↓

Press Z + (wait till it reaches to set value)

Procedure the JOG feed

- Press the JOG mode switch.
- Select the axis to be moved. (Say z-or x-)
- Keep the feed rate switch open.
- Keep pressing the direction switch until the tool reaches the desired position.

Procedure for incremental feed

- Select the INC mode
- Keep the feed rate switch open
- Select the distance to be moved in each step with the magnification dial.
- Select the axis.
- Press the distance switch.
- Note the movement of the axis.

Machine off procedure

Press emergency button (Red colour)

↓

Switch "OFF" NC (Red button)

↓

Switch "OFF" machine (located at the side of the machine)

↓

Switch "OFF" stabilizer

↓

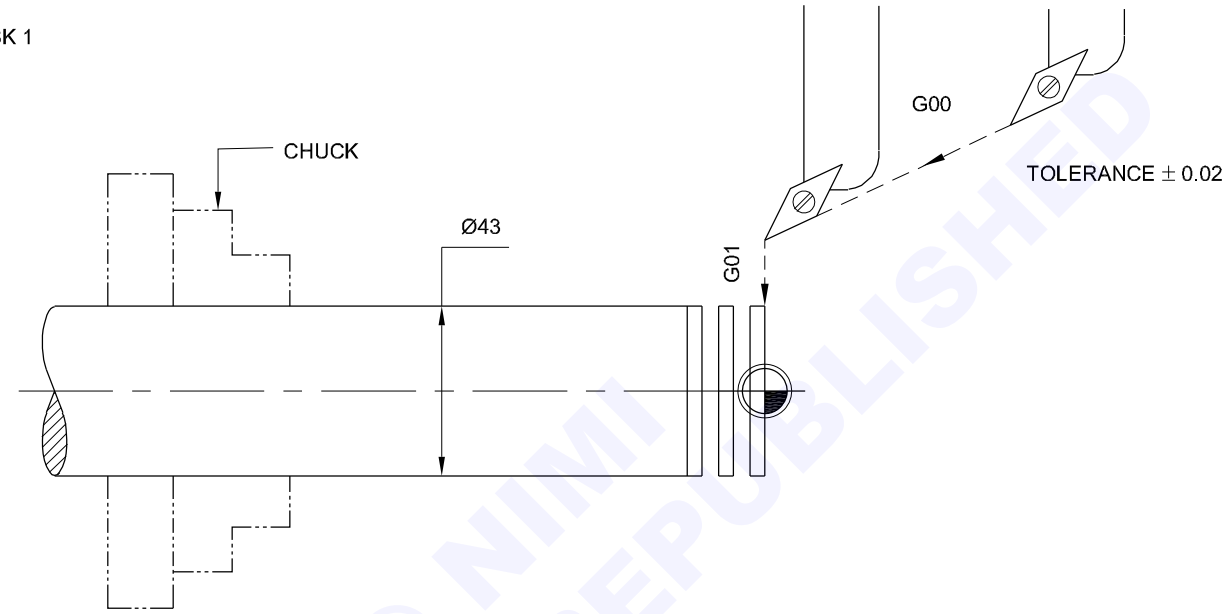
Switch "OFF" main power supply.

Note: Follow the above procedure to switch 'OFF' the machine. If the main is switched OFF without following shutdown procedure, the MCU will get collapsed, just like in a computer system.

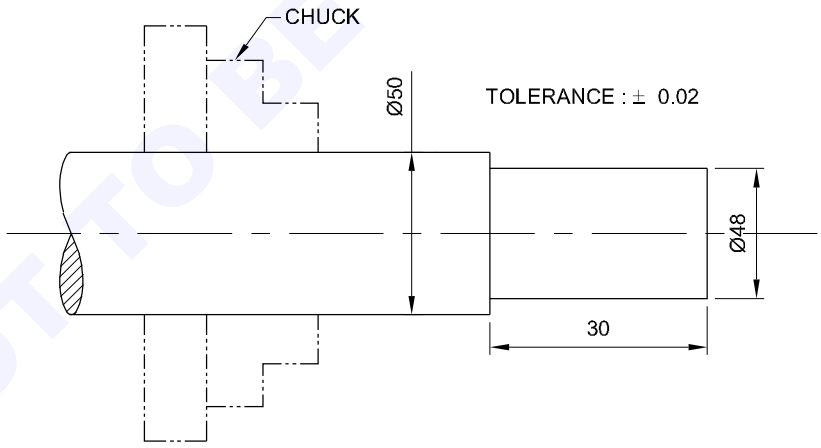
Check CNC part programming simple exercises using various programming codes

Objectives: At the end of this exercise you shall be able to
- **prepare the CNC part programme for facing**
- **prepare the CNC part programme for turning.**

TASK 1



TASK 2



ALL DIMENSIONS ARE IN mm

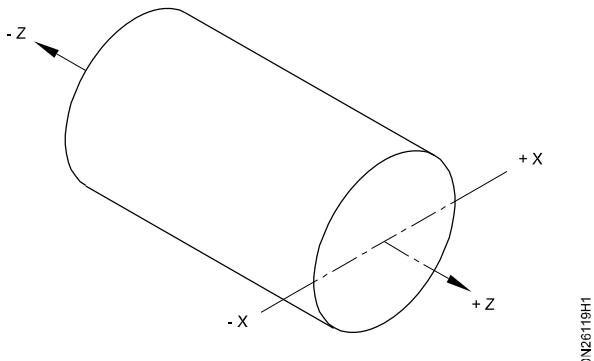
-	Ø50 x 100	-	Fe310	-	TASK 2	2.6.119
-	Ø43 x 100	-	Fe310	-	TASK 1	2.6.119
NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.

SCALE 1:1	**PROGRAM FOR FACING AND TURNING OPERATION**	DEVIATIONS ± 0.1	TIME :
		CODE NO. TU20N26119E1	

Job Sequence

- Study the drawing dimension
- Check the raw material for the required size
- List out the G code & M code needed for the programme
- Fix the axes for turning operation (Fig 1)
- Prepare & write down the part program
- Ensure the tool T03 in place.

Fig 1



ROTATIONAL WORKPIECES

- Hold the job on machine chuck & clamp securely
- Get the program checked for its correctness (Avail instructor's help)
- Try to make few similar program for some other components made earlier.

Axis movement

Part program for facing in sinumeric (Task 1)

```
%15
N5    G72   G90   G95
N10   G00   G53   X180  Z0;
N15   D05   M06   T03;
N20   M03   S900  F0.1;
N25   G00   X50   Z5;
N30   G01   Z-0.5
N35   X-1;
N40   Z2    X50;
N45   Z-1;
N50   X-1;
N55   Z2    X50;
N60   Z-1.5;
N65   X-1;
N70   Z2    X50;
N75   D0    M05   M30;
```

Part program for facing in fanuc (Task 1)

```
:0001
N5 G28 U0    W0;
N10   G50   S1500  T0101;
N15   G96   S900   M03;
N20   G00   X50.0  Z5.0;
N25   Z-0.5;
N30   G01   X-1.0  F0.1;
N35   G00   Z2.0   X50.0;
N40   Z-1.5;
N45   G01   X-1.0  F0.1;
N50   G00   Z2.0   Z50.0;
N55   G28   U0     W0;
N60   M05;
N65   M30;
```

Part program for turning in sinumeric (Task 2)

```
%20
N05   G71   G90   G95;
N10   G00   G53   X180  Z0;
N15   D04   M06   T03;
N20   M03   S900  F0.1;
N25   G00   X50   Z2;
N30   G01   X49;
N35   Z-30;
N40   G00   X50   Z5;
N45   G01   X48;
N50   Z-30;
N55   G00   X50   Z5;
N60   M05   D0    M30;
```

Part program for turning in fanuc (Task 2)

```
0000;
N1;
G28   U0    W0;
G50   S1500 T0101;
G96   S900  M03;
G00   X50.0 Z2.0;
X49.0;
G01   Z-30.0 F0.1;
G00   X50.0 Z5.0;
X48.0;
F01   Z-30.0 F0.1;
X50.0;
G00   X52.0 Z5.0;
G28   U0    W0;
M05;
M30;
```

Check the programme simulation on machine OR practice in simulation software in respective control system

Objectives: At the end of this exercise you shall be able to
- **write the programme in edit mode**
- **run the programme in simulation mode on machine**
- **run the programme in simulation software.**

Job Sequence

Machine simulation

- Switch ON N.C on machine
- Select edit mode
- Write the programme
- Select appropriate tools
- Select simulation mode
- Observe the tool path and programme sequence (Fig 1)

Fig 1

- Correct the programme in edit mode if any error is observed
- See the simulation again
- If the programme is correct, exit the simulation mode
- Switch OFF NC control

Job sequence for software simulation (Fig 2)

Fig 2

- Open simulation software on P.C or multimedia system
- Enter the programme
- Select appropriate tools
- Run the programme
- Observe the tool path and programme sequence (Fig 2)
- Correct the programme in edit mode if any error is observed.
- See the programme again

If the programme is correct, EXIT

Close the software.

Note: Instructor to guide the trainee to write various programme and see the simulation on the machine (or) in the software control sytem.

Alsolute and incremental programming assignments and simulations

Objective: At the end of this exercise you shall be able to
- **learn absolute and incremental programming of job.**

Job Sequence

TASK 1 : **(Fig 1)**

Method of Programming

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.

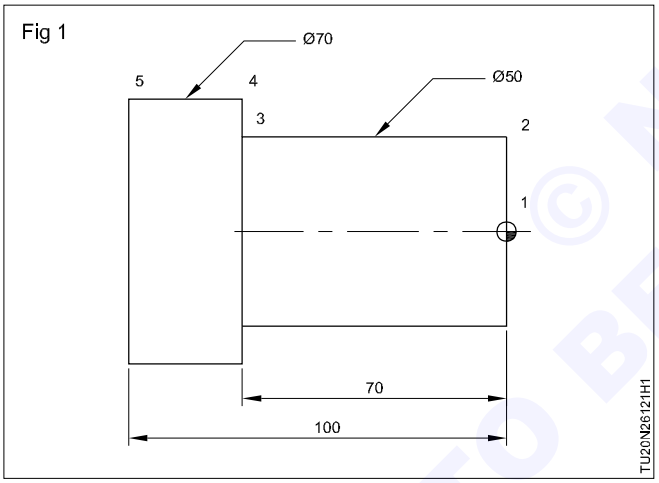
Absolute Programming

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

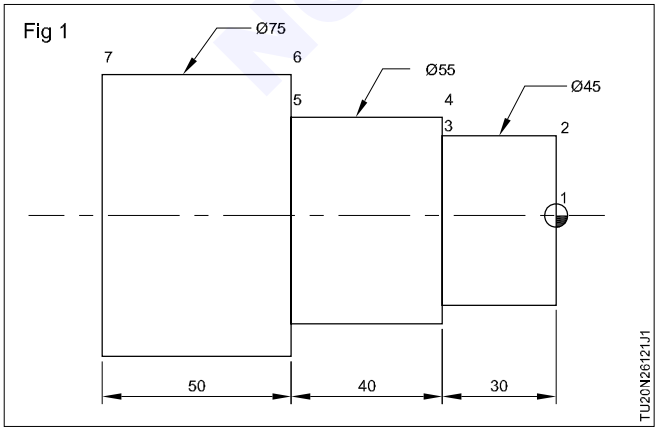
Incremental		
Position	**U**	**W**
1	0.0	0.0
2	50.0	0.0
3	0.00	- 70.0
4	20.0	0.0
5	0.0	- 30.0

Exercise for Absolute & Incremental Methods

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

TASK 2 : **(Fig 1)**

Fig 1



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

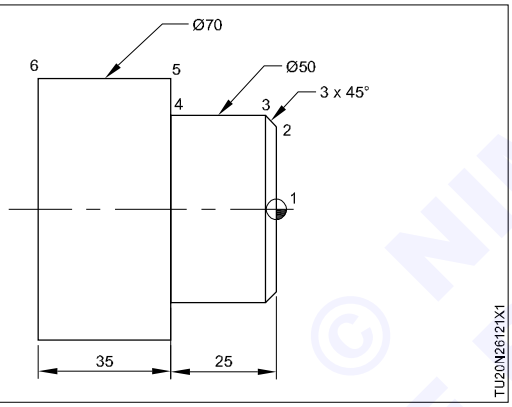
Incremental		
Position	**U**	**W**
1		
2		
3		
4		
5		
6		
7		

— — — — — — — —

TASK 3 **: (Fig 1)**

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

Fig 1



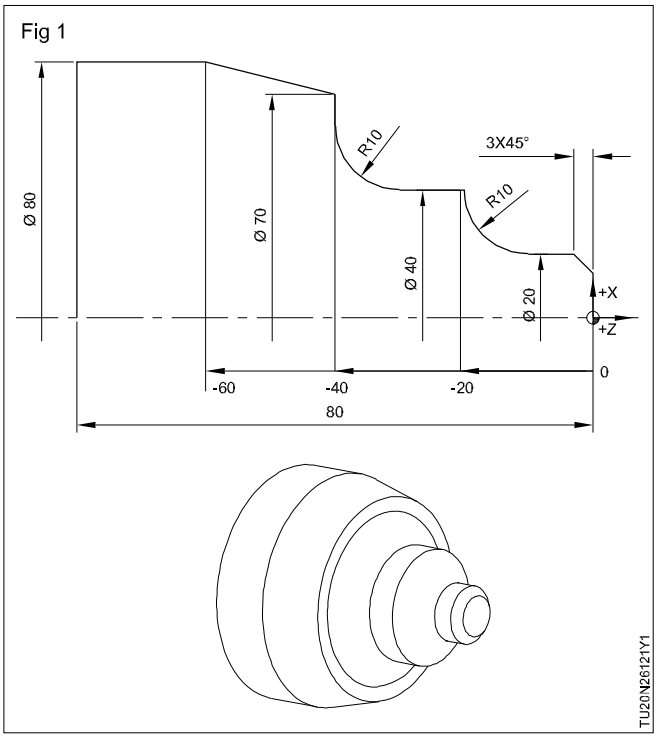
Absolute		
Position	**X**	**Z**
1		
2		
3		
4		
5		
6		

Incremental		
Position	**U**	**W**
1		
2		
3		
4		
5		
6		

— — — — — — — —

TASK 4 **: (Fig 1)**

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



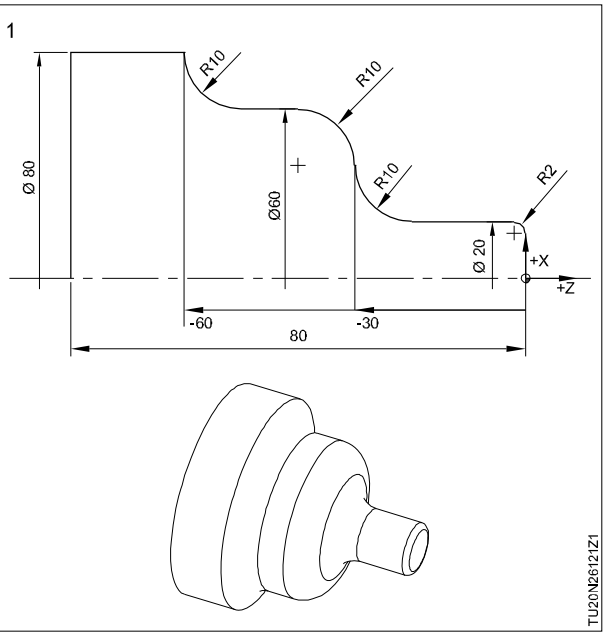
G90					
N	**G**	**X**	**Z**	**I**	**K**
N1					
N2					
N3					
N4					
N5					
N6					
N7					
N8					
N9					
N10					
N11					
N12					
N13					

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

TASK 5 **: (Fig 1)**

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

Fig 1



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

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

Linear interpolation and circular interpolation assignments and simulations on software

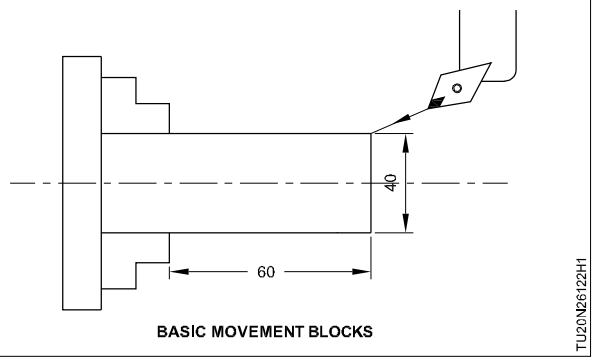
Objectives: At the end of this exercise you shall be able to
- **prepare the various programme in linear, circular interpolation**
- **check the prgramme in simulation software.**

Job Sequence

TASK 1: **Rapid positioning**

Rapid positioning G00 X….Z

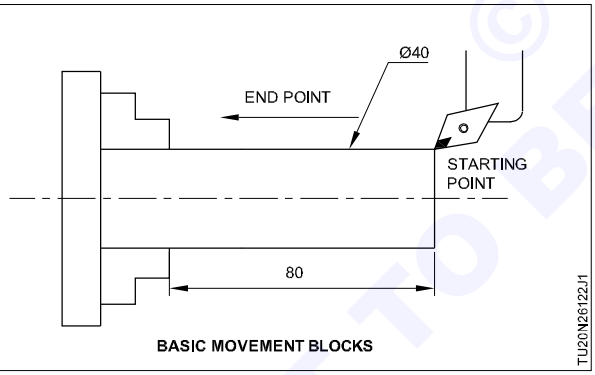
Fig 1



TASK 2: **Linear interpolation**

Linear interpolation G01 X….Z

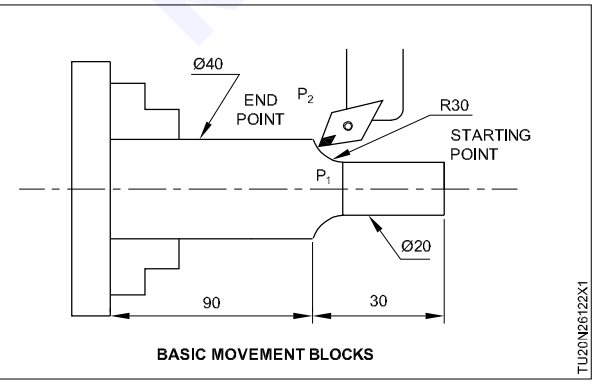
Fig 1



TASK 3: **Circular interpolation (CW)**

Circular interpolation (CW) G02 G90 X…Z..

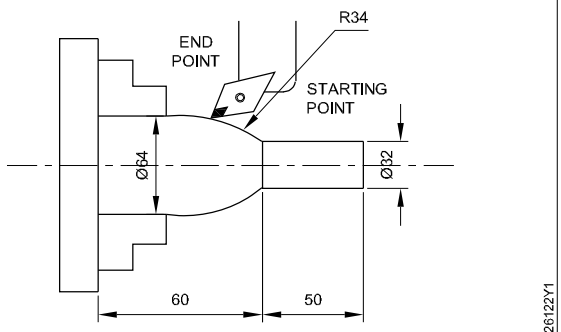
Fig 1



TASK 4: **Circular interpolation (CCW)**

Circular interpolation (CCW) G03 G90 X…Z..

Fig 1



BASIC MOVEMENT BLOCKS

Rapid positioning

Write and simulate the program
TASK 1

Linear inerpolation

Write and simulate the program
TASK 2

Circular inter polation (CW)

Write and simulate the program
TASK 3

Circular inter polation (CW)

Write and simulate the program
TASK 4

Note: Instructor to scrutinze the programme writtern by trainees and guide them to simulate the programme in machine or simulation software.

Perform work and tool setting & tool setup and live tool setup

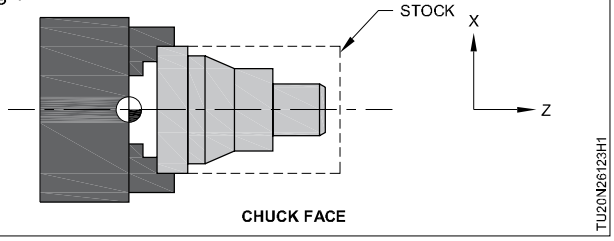
Objectives: At the end of this exercise you shall be able to
- **set the work co-ordinate system**
- **set the tool with respect to the work co-ordinate system**
- **set the live tool datum.**

PROCEDURE

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

- Set the work piece zero at the chuck face

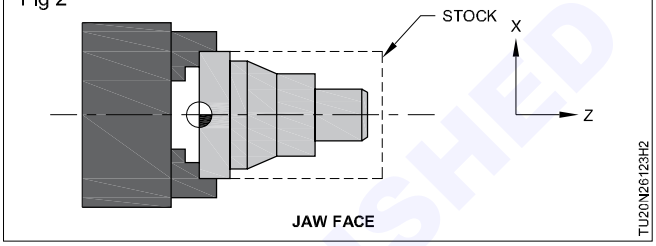
Fig 1



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.

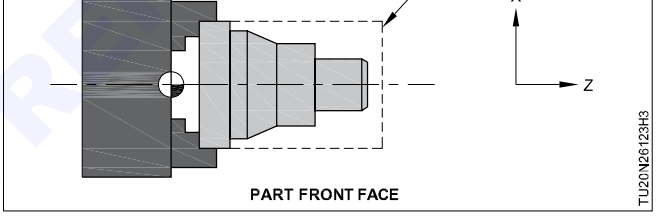
- Work co-ordinate on face of jaw (Fig 2)
- Set the work piece on the face of the jaw as shown in Fig 2.
- In this case the x zero and z zero is work piece seating side is/or in the face of jaw

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

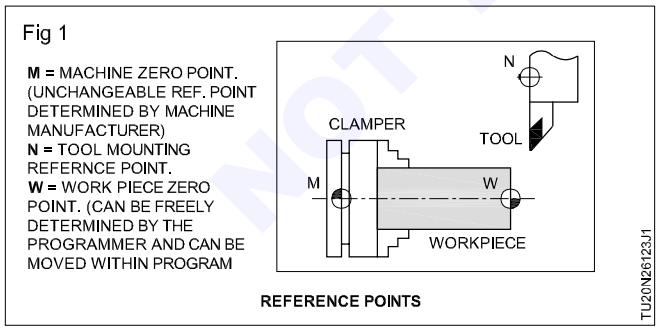
Fig 3



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

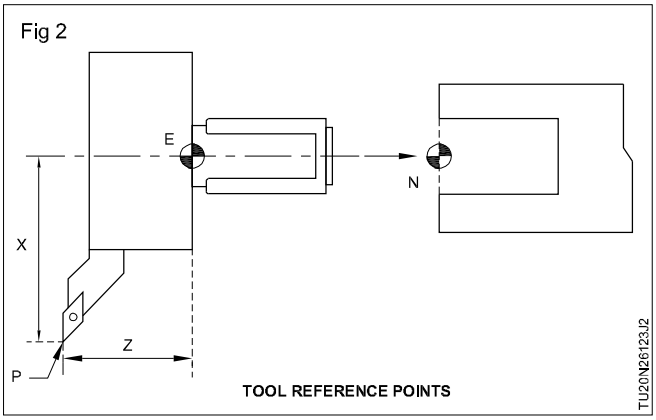
- Turret reference point in relation to the work co-ordinate system is shown in Fig 1

Fig 1



- 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

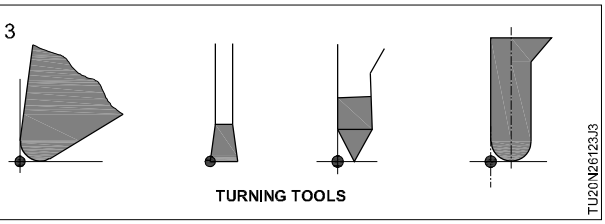


- Measure x and z distance and enter in tool offset page, then 'P' will become commanding point

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

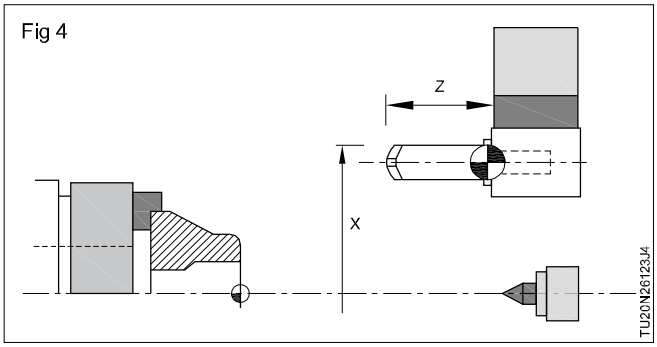
- For various tool the commanding point are shown in Fig 3

Fig 3



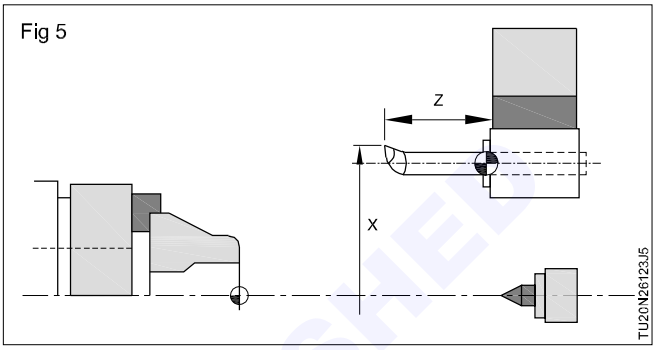
- For tools such as drill and other point to point tool used in milling or turning the reference point is always the extreme tip of the tool measured along z axis . x will always zero (Fig 4)

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



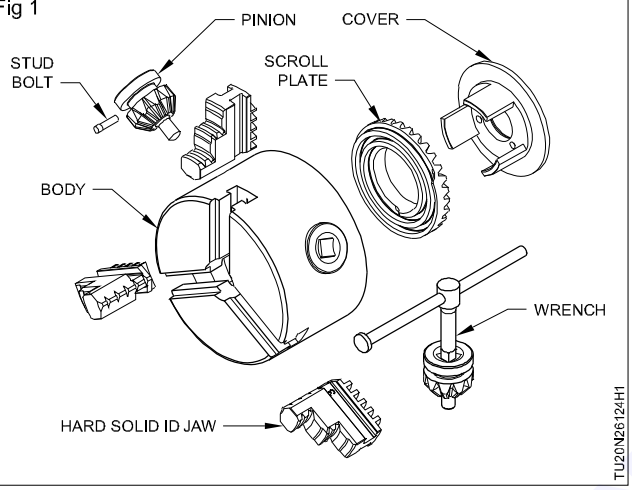
Carryout jaw adjustment according to diameter and tooling setup on turret

Objectives: At the end of this exercise you shall be able to
- **set the jaw of chuck according to the diameter of job**
- **reverse the jaw of chuck for larger diameter job**
- **set the tool in the respective slot of turret.**

Job Sequence

- **Parts of 3 jaw chuck (Fig 1)**

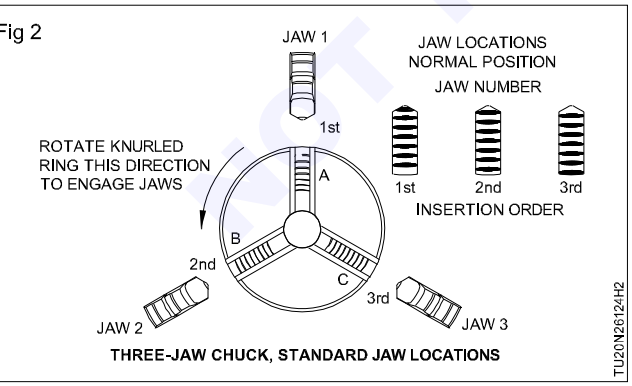
Fig 1



- Loosen allen screw of all the three jaws.
- Count the number of serration radially outwards
- Leave equal number of serration (Racks) outwards and hold jaws firmly
- Ensure the space slightly higher than the diameter of job.
- Tighten the allen screws.
- Check concentricity of job using dial test indicator.

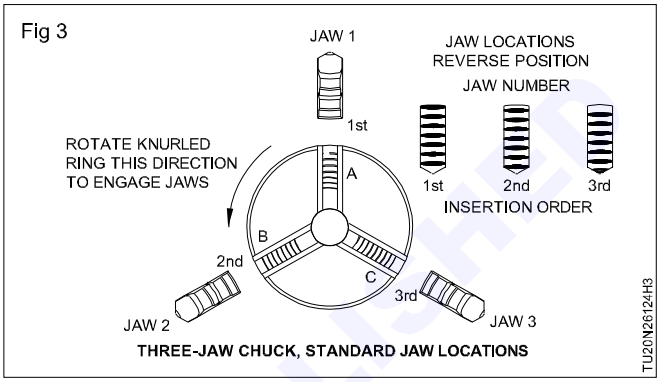
Reversing the jaws (Fig 2)

Fig 2



- Rotate the knurled scroll until the jaws can be removed.
- Insert the jaws in the order and location as per Fig 3

Fig 3



- Turn the scroll counter clockwise until the outside start of scroll thread is just ready to pass the slot for the jaw.
- Slide the first jaw as for as possible in the slot.
- Turn scroll until the first jaw is engaged
- Engage second and third jaws following aboe step.

Note: Do not turn the lathe spindle on until the chuck is tightened. The acceleration of the spindle can cause the scroll to open the chuck jaws if not tightened!

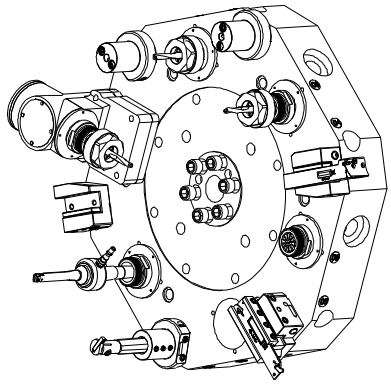
Do not overtighten the chuck. Use only moderate pressure with the supplied tommy bars.

Tooling setup on turret (Fig 4&5)

- Select the tool numer on turret by JOG or MDI mode
- Loosen a pair of allen screw near the tool slot on turret insert shank of tool.
- Insert shank of tool
- Tighten the allen screws
- Take tool OFFSET for that tool before using
- Add tool OFFSET number in the respective programme.

Note: Distribute tool weight evenly across the turret.

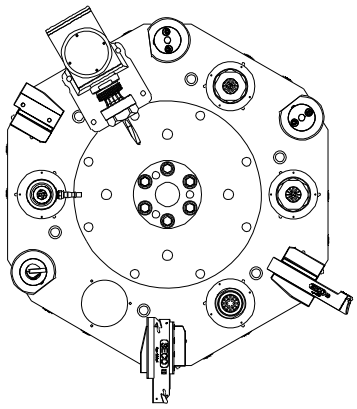
Fig 4



TOOL TURRET

TU20N26124H4

Fig 5



TOOL TURRET

TU20N26124H5

CNC turning centre operation in various modes (JOG, EDIT, MDI, SBL & AUTO MODE)

Objectives: At the end of this exercise you shall be able to
- **operate with JOG mode**
- **operate with EDIT & MDI mode**
- **operate with SBL & AUTO mode.**

Fig 1

Job Sequence

JOG mode

1 For axis movement
- Select 'JOG' mode.
- Press X+, X-, Z+ & Z- for moving axis.
- Keep pressing direction switch until the tool reaches desired position
- To move rapidly, press "JOG", then press "RAPID + (X+, X-, Z+ OR Z-)

2 For indexing tool turret
- Press "JOG" key
- Press "Tool" key

3 For spindle rotation
- Press "JOG" key
- Press SPINDLE CW/CCW

EIDT mode

1 Set "EDIT" mode

2 Enter new programme in block by block.

3 Use Input/Alter/Delete/EOB keys whenver required.

4 To alter a block or programme, correct a block or programme or deleting a block or programme, go to edit mode.

MDI mode (Fanuc)/MDA mode (Sinumeric)

For home position

This function is used for homing XZ axis

1 Press "MDI" key

2 Press "PROG" key., keep pressing that key constantly until the "Prgram MDI" is appeared on screen.

3 Type code "G28 U0.00 W0.00;"

4 Press "INSERT" key

5 Press "CYCLE START" key.

For tool indexing

1 Press "MDI" key.

2 Long press "PROG" key until the "Program MDI" appears on the screen

3 Type code " T0100".

4 Press "INSERT" key.

5 Press "CYCLE START" key

For spindle rotation

1 Press "MDI" key

2 Long press "PROG" key, until the "Program MDI" appears on the screen.

3 Type code " G97 S2000 M03;.

4 Press "INSERT" key.

5 Press "CYCLE START" key.

Note: G97 - constant surface speed control.

Single block mode

1 Enter the programme

2 Press "SINGLE BLOCK" key

3 Press " CYCLE START" key (machine will pause-after exicuting the first line of code)

4 Repeat step 3, until all lines of programme are exicuted.

5 Turn OFF "SINGLE BLOCK" key, to exit "SINGLE BLOCK" mode.

Note: "SINGLE BLOCK" mode allows an operator to check the programme by executing.

Auto mode

1 Enter the programme

2 Press "AUTO" mode key.

3 Press "CYCLE START" button to start the programme

(Machine will execute the pogramme from start to finish)

4 If any errors in the programme, machine will stop at the error.

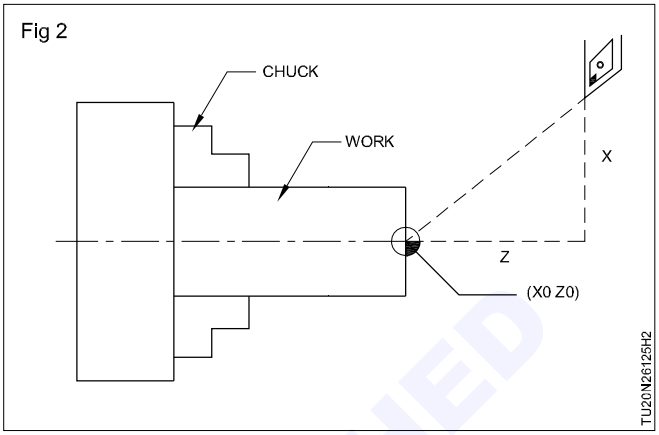
5 Correct the error, press "CYCLE START" key to continue.

6 Turn OFF "AUTO" mode key to exit "AUTO" mode.

Note: While machine is running this mode, don't change to any other mode on function.

Instruct the trainee to enter the below programme and execute in SBL and AUTO mode. (Fig 2)

Fig 2



%20

N1 G01 X50 Z5 F0.1;

N2 S1200 M03;

N4 M08;

N5 G01 X48 F0.1;

N6 G01 Z-25 F0.1;

N7 G00 X60 Z10;

N8 M09;

N9 M05;

N10 M30;

Material: M.S rod ϕ50x80

Tolerance: ±0.02

All dimenstion are in "mm"

Programme entry in CNC turning centre

Objectives: At the end of this exercise you shall be able to
- **enter the programme in Fanuc control**
- **prepare the CNC programme and enter in CNC control insert job drawing.**

Job Sequence

Fanuc control

↓

Go to JOG mode

↓

Press Part Program

↓

Press Edit

↓

Enter % and new program number of job

↓

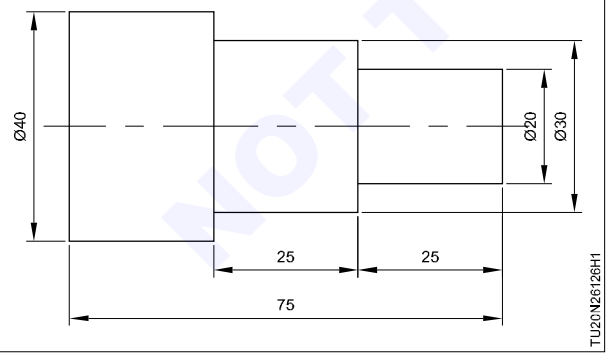
Press Select Program

(Blank space open for entry of program)

↓

Write program using numerical key and edit key

Fig 1



Material: M.S rod ϕ40x75

Tolerance: ±0.02

All dimenstion are in "mm"

```
%
O 0001
N1
G28   U0     W0;
G97   S1000  M03;
T0101;
G00   X45.0  Z5.0   M08;
Z0.0;
G01   X0.0   F0.1;
G01   X40.0  F0.1;
G01   Z-75.0 F0.1;
G00   X45.0  Z5.0;
G01   X30.0  F0.1;
G01   Z-50.0 F0.1;
G00   X45.0  Z5.0;
G01   X20.0  F0.1;
G01   Z-25.0 F0.1;
G00   X50.0  Z10.0  M09;
G28   U0     W0;
M05;
M01;
M30;
```

Write the sinumeric programme for above Fig 1, and enter in sinumeric control

.....

.....

.....

.....

.....

.....

.....

Set the tool offsets, entry of tool nose radius and orientation

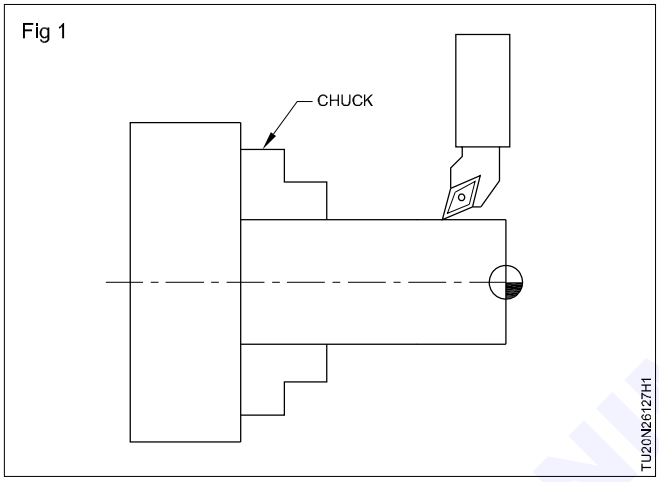
Objectives: At the end of this exercise you shall be able to
- **set and enter the tool offset in X and Z direction**
- **set and enter tool nose radius**
- **orient different tools according to work.**

Job Sequence

Turning tool offsetting

X-Axis tool offsetting (Fig 1)

Fig 1



Material: M.S rod ϕ40x80

All dimenstion are in "mm"

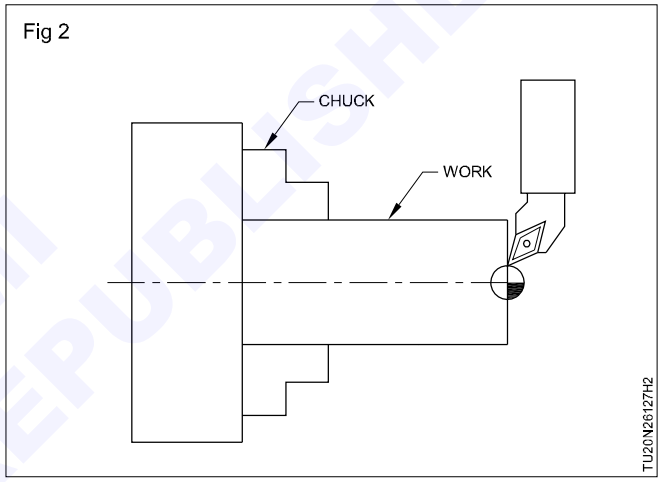
Reference tool is T01 and OFFSET is zero in X and Z axis

- Clamp the job in chuck
- Select "MDI" mode.
- Press "PROG" key
- Enter tool number: T0202 (Turning tool)
- Press "INSERT" key
- Press "CYCLE START" key.
- Enter "M03 S1500;
- Press "INSERT" key
- Press "CYCLE START" key
- Select "JOG" or "MPG" mode to move X-axis and Z-axis.
- Touch the job in X-axis, just clean OD (for ensure no disturbance)
- Then stop spindle
- Measure OD diameter, using vernier or micrometer
- Select "OFFSET" key displayed in geometry mode
- Using cursor in geometry screen, select tool no:2 and X axis.
- Enter job diameter (OD)
- Press "measure" in soft key

Now tool offset in X-axis is saved.

Z axis tool offsetting (Fig 2)

Fig 2



- Spindle ON
- Select "JOG" or "MPG" mode.
- Manually face the job (for ensure no disturbance)
- Select "OFFSET" key in geometry mode
- Select tool no 2 and Z axis
- Enter Z0.0
- Press "MEASURE" in soft key
- Now tool offset in Z axis is saved.

To measure and enter the tool nose radus in a CNC lathe in Fanuc controller

- Mount the tool in the tool holder and secure it with the tool clamping screw.
- Set the tool offset registers to zero.
- Use a tool presetter to measure the nose radius.
- Enter the measured tool nose radius in the R column of the tool offet table.
- Enter the tool tip orientation number in the T column of the tool offset table.
- Save the tool offset table

Here are the detailed steps

- Mount the tool in the tool holder and secure it with the tool clamping screw. Make sure that the tool is mounted in the correct orientation.
- Set the tool offset registers to zero. This can be done by entering the following commands in the fanuc controller

Code snippet

G52 X0 Z0

- Use a tool presetter to measure the tool nose radius. The tool presetter will measure the radius of the tool tip and display the result on the screen.
- Enter the measured tool nose radius in the R column of the tool offset table. The R column is used to store the tool nose radius. The value that you enter should be the same as the value that was displayed on the tool presetter.
- Enter the tool tip orientation number in the T column of the tool offset table. The T column is used to store the tool tip orientation number. This number is used to determine the direction of the tool nose radius compensation.
- Save the tool offset table. This can be done by entering the following command in the Fanuc controller.

Code snippet/ Save

- One you have completed these steps, the tool nose radius will be strored in the Fanuc controller. You can then use tool nose radius compensation in your CNC programs.

Here are some additional tips for measuring and entering the tool nose radius.

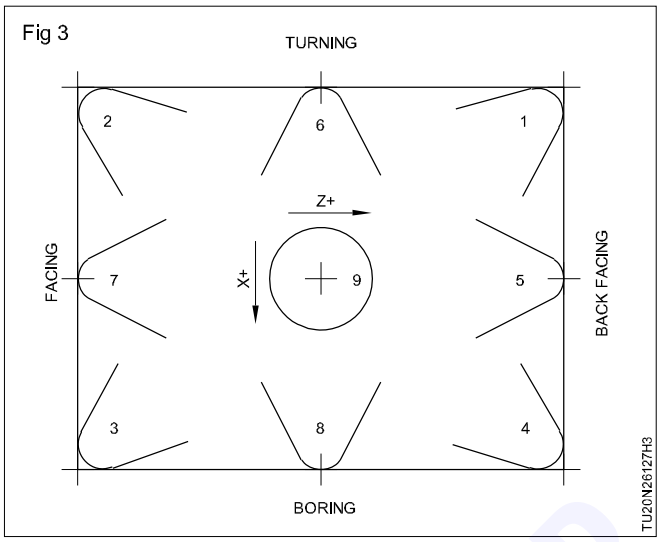
- Make sure that the tool is clean and free of chips before you measure the tool nose radius.
- Use a sharp tool presetter to get accurate measurements
- Double check the values that you enter in the tool offset table.

Tool orientation (Fig 3)

1 Set the tool tip orientation

- Manually
  - a Select the tool that you want to set the orientation for.
  - b Use the machine's handwheel to rotate the tool until the cutting edge is pointing in the desired direction.
  - c Lock the tool in place.
- Automatically
  - a Use a program to rotate the tool to the desired orientation.

Fig 3



  - b The program will need to know the tool's orientation offset, which can be found in the machine's manual.

2 Set the tool orientation offset

- Manually
  - a Enter the tool orientation offset value into the machine's control panel.
  - b The offset value will need to be in the same units as the machine's coordinate system.
- Automatically
  - a The tool orientation offset can be set automatically by the machine's control system.
  - b The machine's manual will have instructions on how to do this.

3 Run a test program

- Create a simple program that will use the tool that you have just set the orientation for.
- Run the program and check to see if the tool is cutting correctly.
- If the tool is not cutting correctly, you will need to troubleshoot the problem and correct it.

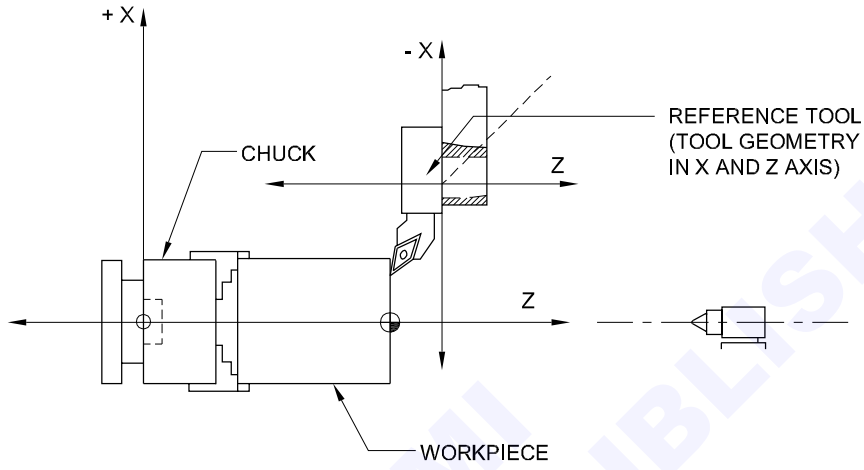
Here are some additional tips for setting tool orientation in CNC turning center

- Make sure that the machine is properly calibrated before you start.
- Use the correct tool for the job.
- Be careful not to damage the tool or the machine.
- Follow the manufacturer's instructions carefully.

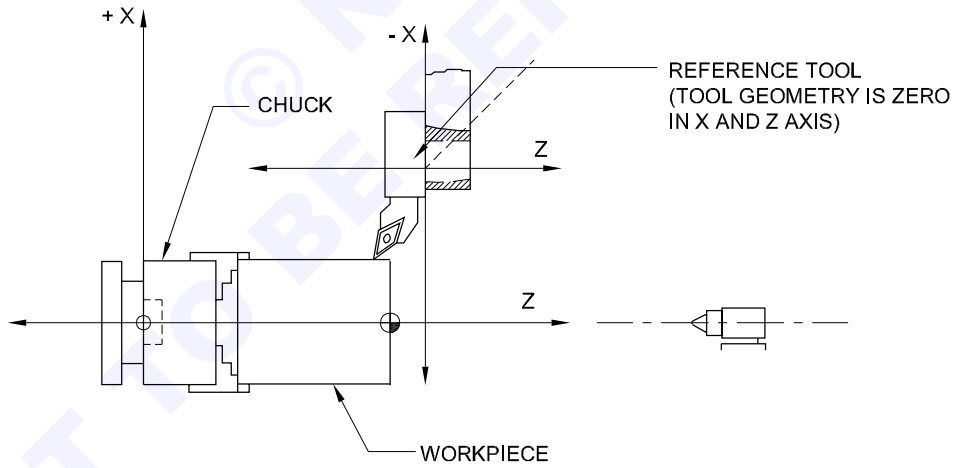
Conduct work offset measurment, tool offset measurement and entry in CNC control

Objectives: At the end of this exercise you shall be able to
- **measure and enter work offest in X & Z axis**
- **measure and enter tool offset in sinumeric.**

Fig 1



WORK OFFSET MEASUREMENT IN Z AXIS



WORK OFFSET MEASUREMENT IN X AXIS

TU20N26128H1

Job Sequence

- Select tool no 1: T0101

Step 1 - First of all you have to reference the machine

a Press "REF" key. (Reference)

b Press "+x: key.

c Press "+z" key

(Tool reach at reference position)

Step 2 - Press "MDI" key.

Step 3 - Press "PROG" key.

(In program mode we give the command for tool T0101)

a Type command T0101 in the system

b Press "EOB" key (to complete block)

c Press "INSERT" key.

Step 4 - Press "CYCLE START" key.

(If your tool no.1 is not on work position it will become on working position)

Step 5 - Now we have to touch the edge of the tool by touching the face and diameter of the workpiece for set offset. But here we take offset of z-axis.

a Press "z or x-axis" key (as you need)

b Press "100 - micron" key

c Then move the tool turret by handwheel for touch the tool edge to workpiee face.

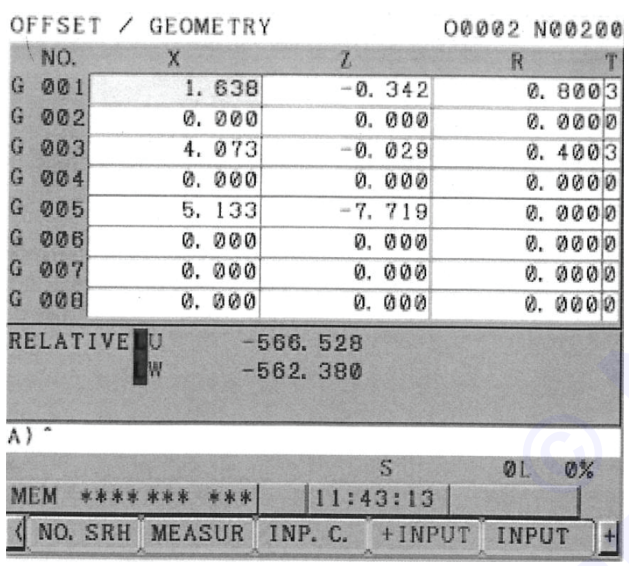
Note: Fanuc CNC have many models there are more changes in MID panel. Some model there is no handwheel at time u can use an alternative method to move tool turret

Step 6 - After touching tool we move on the system screen.

Press "OFFSET" KEY.

Step 7 - for entering an offset data (Fig 2)

Fig 2



a Select "GEOM" (geometry) option which is on screen by using just below soft key.

b Set the cursor on a tool no G001 (Tool no 1)

c The take cursor on z-axis in front of tool no G001.

Step 8

a Type code Z0.0 by using keys. Then

b Select "MEASUR" option which is on screen by using just below soft key.

(here we saw the z-axis offset value save in z column)

Step 9 - Now we want to take the offset of the x-axis. We want to do the same here as we did in the z -axis.

a Press "X or Z-axis" key. (as you need touch the tool at the workpiece diameter).

b Then move the tool turret by handwheel for touch the tool edge to workpiece diameter.

Step 10 - Go on screen.

a Set cursor on x-axis column front of tool no G001.

b Type code X50 (actual diameter of the workpiece) by using keys. Then

c Select "MEASUR" option which is on screen by using just below soft key.

(here we saw the X-axis offset value save in X column)

Step 11 - For tool take home position by using "ref" mode.

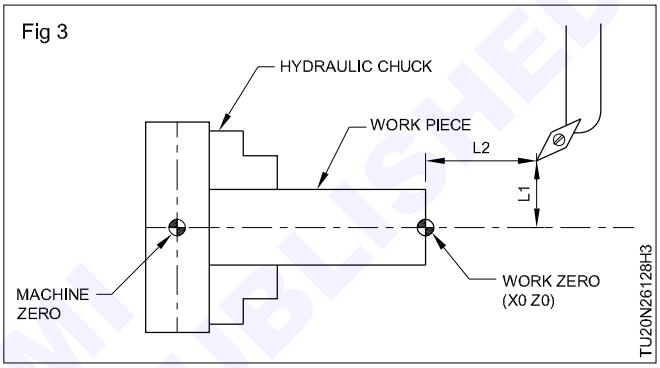
a Press "REF" key

b Press "+X" key

c Press "+z" KEY.

Tool offset (sinumeric) (Fig 3)

Fig 3



- Tool offset is done to find out work zero.
- To do the tool offset we have to find the tool geometry (i.e L1 and L2) for each tool.
- **L2 Geometry:** Run the spindle take the tool towards the face of the job and face it, if length permits note down the value of Z ie. The value of L2.

  Example L2 = -294.754 (Directly taken from the system)

L1 Geometry

- In JOG mode run the spindle, take the tool towards diameter of the job and touch it and do plain turning (if material allows) and note down the value of x and measure the diameter of the work then find out L1 by using the following formula.

$$L1 = \frac{\text{The value of X} - \text{Actual diameter of Job}}{2}$$

$$L1 = \frac{\text{The value of} \times - \text{Actual diameter of Job}}{2}$$

$$L1 = \frac{28.68 - 31.62}{2}$$

$$L1 = \frac{-2.94}{2}$$

L1 - 1.47

Eample: The value of x = 28.68 (from control panel)

Acutal of work = 31.62 mm

Now select the tool off set

(Follow the procedure select JOG mode)

↓

Select tool offset (By using soft keys)

↓

Now enter tool offset number E.g. D5 (D1 to 99)

↓

Enter tool number Example T3 (T1 to T8)

↓

Enter tool type (1 to 8) please refer tool type

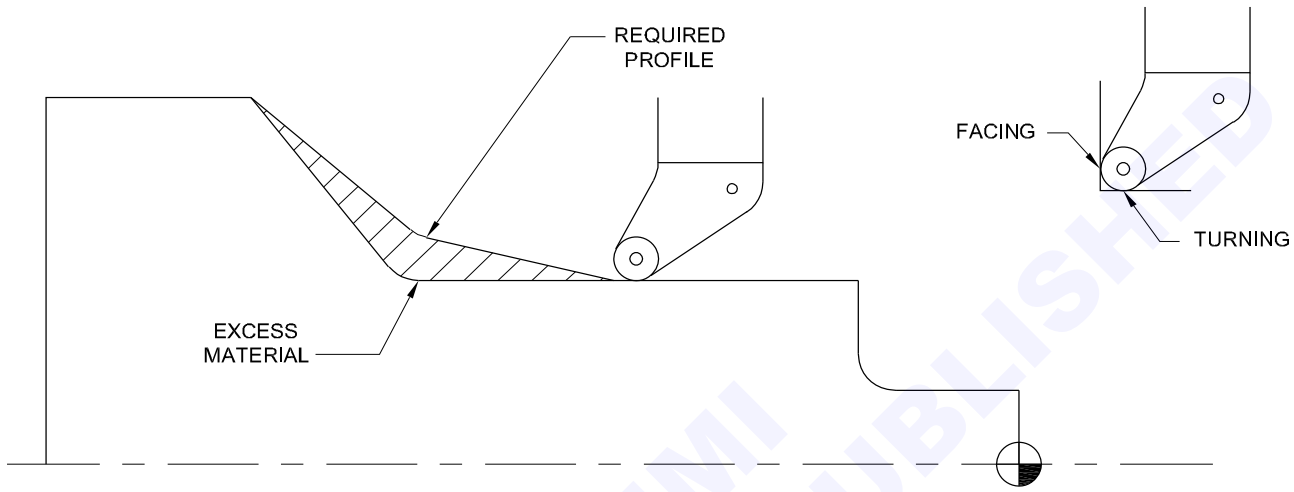
↓

Enter L1 = -1.47

L2 = -294.754 (from the system)

Make tool nose radius and tool orientation entry in CNC control

Objectives: At the end of this exercise you shall be able to
- **set the tool nose radius compensation**
- **set the tool imaginary nose position to 'T' in tool geomentry offset.**



Job Sequence

- Study the drawing
- Select the suitable tool for the operation
- Identify the tool type
- Study the tool nose radius
- Input the value of tool nose radius in G41 & G42 in the programme.

-	-	-	-	-	-	2.6.129
NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.
SCALE 1:1	MAKE TOOL NOSE RADIUS AND TOOL ORIENTATION ENTRY IN CNC CONTROL				DEVIATIONS ± 0.1	TIME :
					CODE NO. TU20N26129E1	

Skill Sequence

Tool nose radius and its imaginary nose positions

Objectives: This shall help you to
- **interpret G41, G42 and G40 codes**
- **list tool types of various operation.**

The following data's must be specified to carryout automatic tool nose radius compensation to obtain required profile exactly

- For external turning and to move the tool in left with respect to job program as G41 (Tool left) and to move tool right with respect to job program as G42 (Tool right)
- For internal turning like boring and to move left with respect to the job the tool program as G41 (Tool Left) and to move the tool right with respect to job, program as G42 (Tool right)
- Input the radius of tool to R in geometry offset.
- Input the imaginary nose position in 'T' in geometry offset.

Fig 1

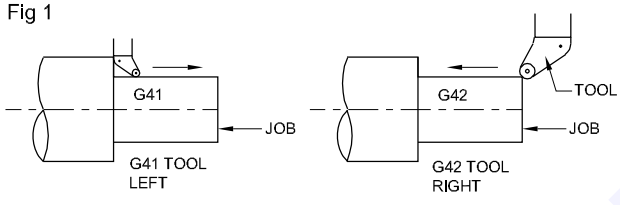
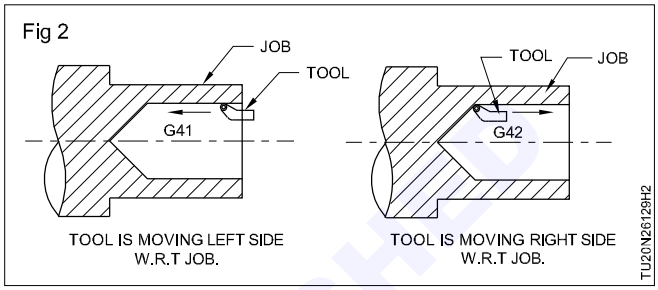


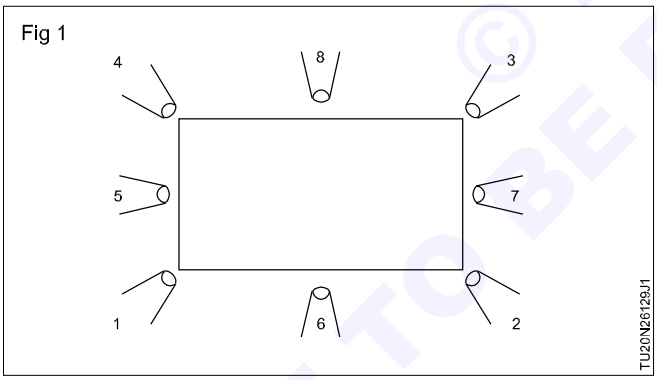
Fig 2



Imaginary nose position to Tool offset in Geometry offset (Tool type)

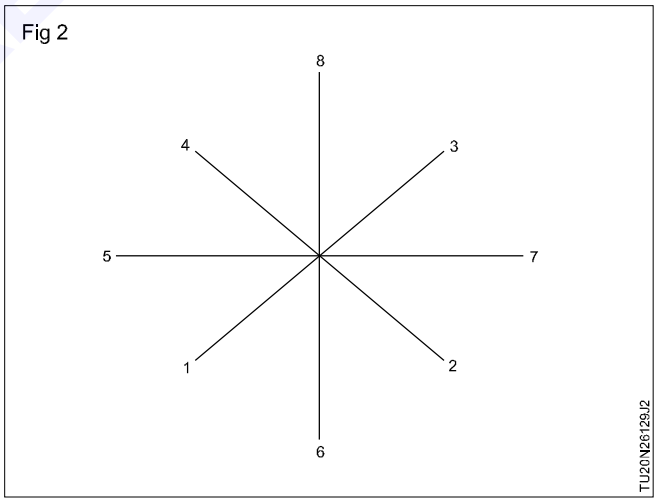
Method - I

Fig 1



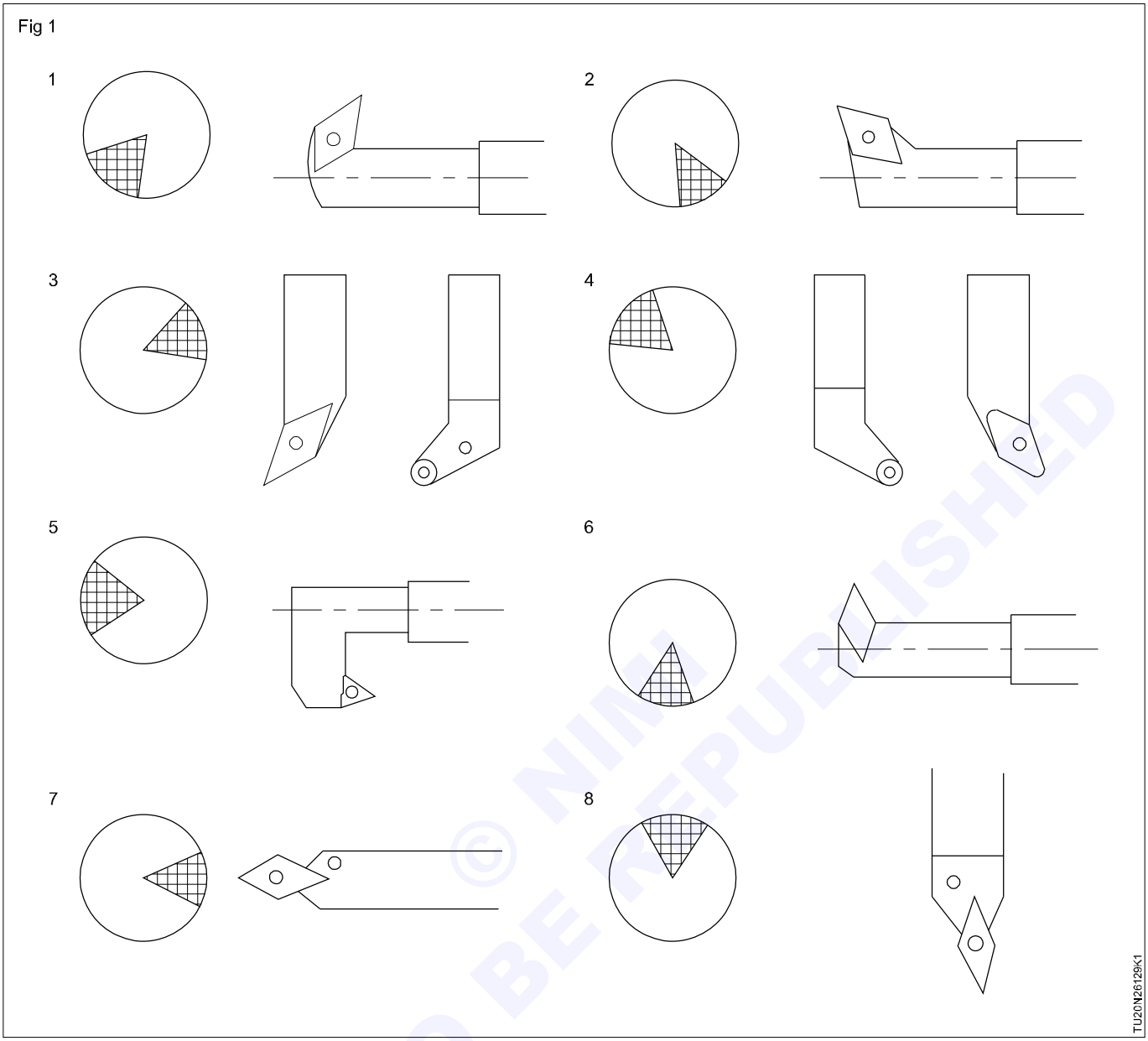
Method - II

Fig 2



Method - III

Fig 1

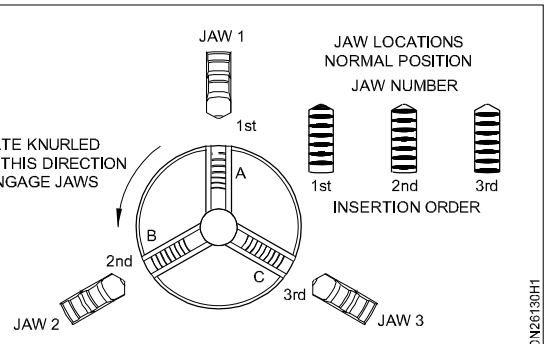


Jaw removal and mounting on CNC lathe

Objectives: At the end of this exercise you shall be able to
- **remove the jaws and assemble**
- **reverse the jaws for larger diameter jobs.**

Job Sequence

Fig 1



THREE-JAW CHUCK, STANDARD JAW LOCATIONS

Trainee will write job sequence for dismantling and assembling of chuck for given job.

1 .....

2 .....

3 .....

4 .....

5 .....

6 .....

7 .....

8 .....

9 .....

10 .....

11 .....

12 .....

13 .....

14 .....

15 .....

16 .....

17 .....

18 .....

19 .....

20 .....

Manual data input (MDI), manual pulse generator (MPG) mode operations & checking zero offsets and tool offsets

Objectives: At the end of this exercise you shall be able to
- **data input and opertion of MDI & MPG mode**
- **checking the zero offsets and tool offsets in MDI mode**
- **checking of zero offsets and tool offsets in MPG mode.**

Job Sequence

MDI operation (Fig 1)

Fig 1

Steps

1 Press the MDI button on the control panel.

2 The MDI prompt will appear on the display.

3 Enter your commands one at a time.

4 To execute a command, press the CYCLE START button.

5 To move the machine manually, use the JOG buttons on the control panel.

6 To exit MDI mode, press the MDI button again.

Example

1 G0 x 100 Y200 - Move to the position X100, Y200.

2 G1 Z-10 F100 - Move down 10mm at a feed rate of 100 mm/min

3 M03 - Start the spindle clockwise

4 M05 - Stop the spindle

Keyboard shortcuts in MDI mode

1 Up and down arrow keys - scroll through the MDI prompt

2 Backspace key - Delete characters from the MDI prompt

3 Cancel key - Clear the MDI prompt and start over.

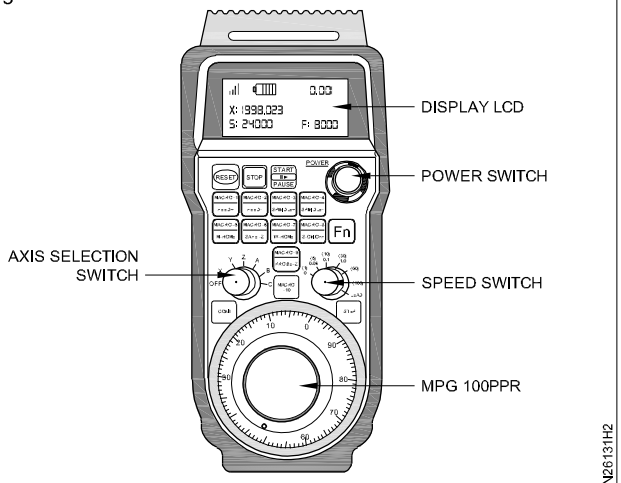
4 Help key - Get help on the MDI commands

Steps for using MPG mode (Fig 2)

1 Enter MPG mode

- Press the MPG mode button on your machine's control panel
- The MPG mode indicator light will turn on, indicating that you are in MPG mode.

Fig 2



MPG WIRELESS PENDANT

2 Input a G-code command.

- Use the machine's keypad or jog controls to enter the G-code command.
- For eample, to move the X axis 100 millimeters, you would enter the following command: G0 x 100.

3 Execute the G-code command.

- Press the MPG start button.
- The machine will then move to the specified position.

4 Use the jog controls to move the machine manually.

- The jog controls are typically labeled with the letters X, Y and Z.
- To move the machine in a particular axis, press and hold the corresponding jog control button.
- The machine will move in the specified direction in small increments.

Tips for MPG mode

1 Make sure that you are familiar with the G-code commands before you start using MPG mode.

2 Use the jog controls to move the machine in small increments until you are close to the deseired position.

3 Use the MPG start button to execute the G-code command.

4 Be careful not to crash the machine into the workpiece.

Checking offsets in MDI mode

1 Set the machine to MDI mode.

1 Enter the following G-codes to check the zero offset.

Code snippet

```
G53    G92    X0    Z0
G10    L2     P0    X0    Z0
```

1 The G53 command tells the machine to use the machine zero coordinates.

2 The G92 command sets the current position of the machine as the new machine zero.

3 The X0 and Z0 parameters set the X and Z axes to 0, respectively.

4 The G10 command sets the tool offset for the tool you are using to 0, 0.

5 Use a dial indicator to verify that the tool tip is located at the machine zero position.

6 Place the dial indicator on the workpiece at the machine zero position.

7 Move the tool tip towards the dial indicator until it touches the tip of the indicator.

8 The reading on the dial indicator should be 0.0000.

Checking offsets in MPG mode

1 Set the machine ot MPG mode.

2 Zero the X, and Z axes by moving the tool tip to the machine zero position and pressing the MPG zero button for each axis.

- To zero the X axis, move the tool tip to the dia of job, where the X axis is zero. Then, press the MPG zero button for the X axis.
- To zero the Z axis, move the tool tip to the face of the job, where the X axis is zero. Then, press the MPG zero button for the Z axis.

3 Enter the following G-code to check the zero offset.

Code snippet

```
G53    G92    X0    Z0
```

1 The G53 command tells the machine to use the machine zero coordinates.

2 The G92 command sets the current position of the machine as the new machine zero.

3 The X0 and Z0 parameters set the X and Z axes to 0, respectively.

4 The G10 command sets the tool offset for the tool you are using to 0, 0.

5 Use a dial indicator to verify that the tool tip is located at the machine zero position.

6 Place the dial indicator on the workpiece at the machine zero position.

7 Move the tool tip towards the dial indicator until it touches the tip of the indicator.

8 The reading on the dial indicator should be 0.0000.

Programme 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.**

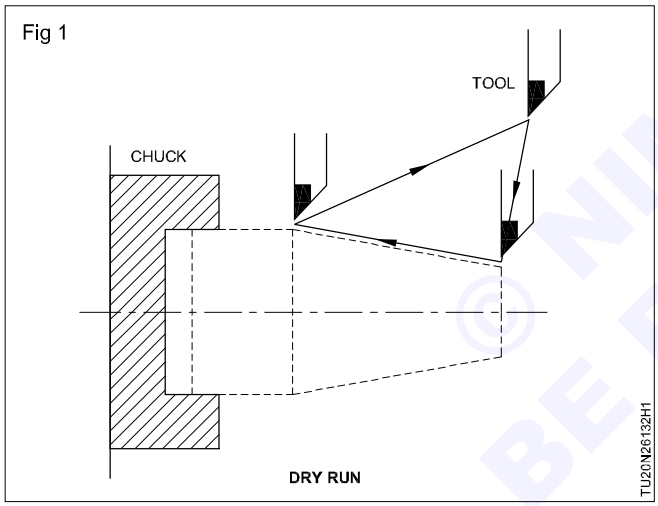
Dry run

The tools is moved at the feed rate specified by a parameter regardless of the feed rate specified in the program. This function is used for checking the movement of the tool under the state that the 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 automatie/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.

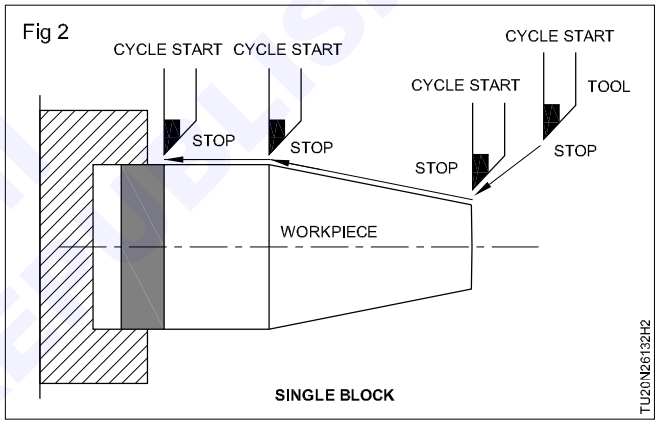
Steps for single block (Fig 2)

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



Checking finish size by over sizing through tool offset

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 single block mode 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 in Table 1

Table 1

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

Note

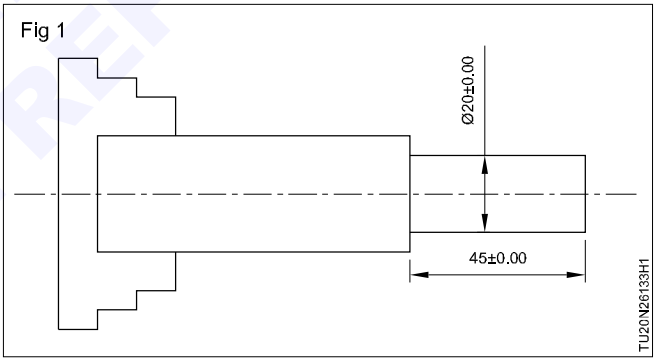
- Required size of the is job X 20 ± 0.00
  Z 45 ± 0.00
- Go to wear offset page and input the X,Z difference value
- Add the Tool wear offset difference value on the table column in Table 2.

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

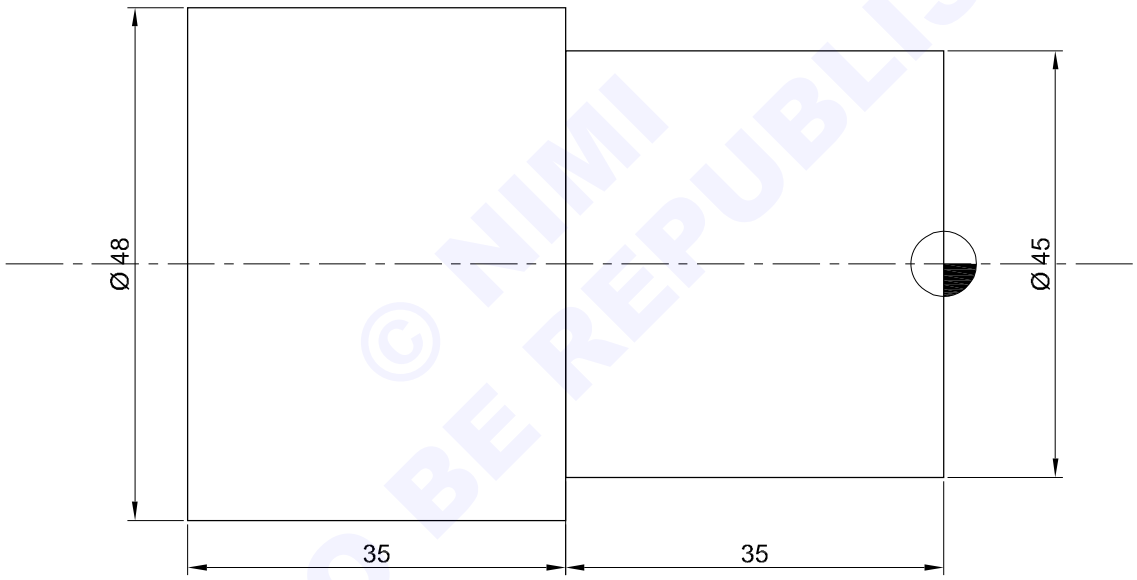


Exercise 2.6.134

Part programme preparation simulation and automatic mode execution

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		-		-	-	2.6.134
NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.
SCALE 1:1	FACING AND TURNING PROGRAM				DEVIATIONS	TIME :
					CODE NO.TU20N26134E1	

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; - Tool change with spindle on cc10

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;

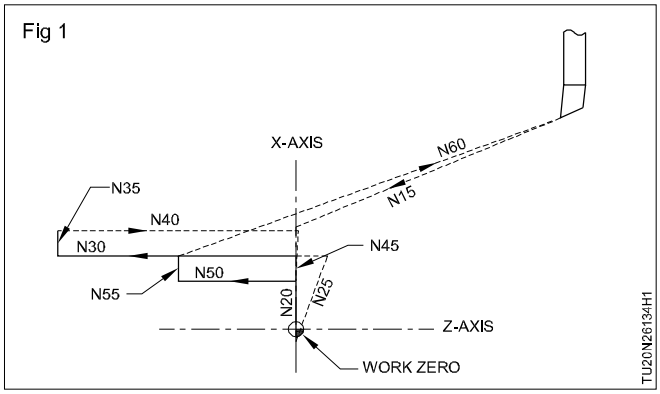
N65 G28 G91 X0.00 Z0.00 T0500 M05;

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 G0.1 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

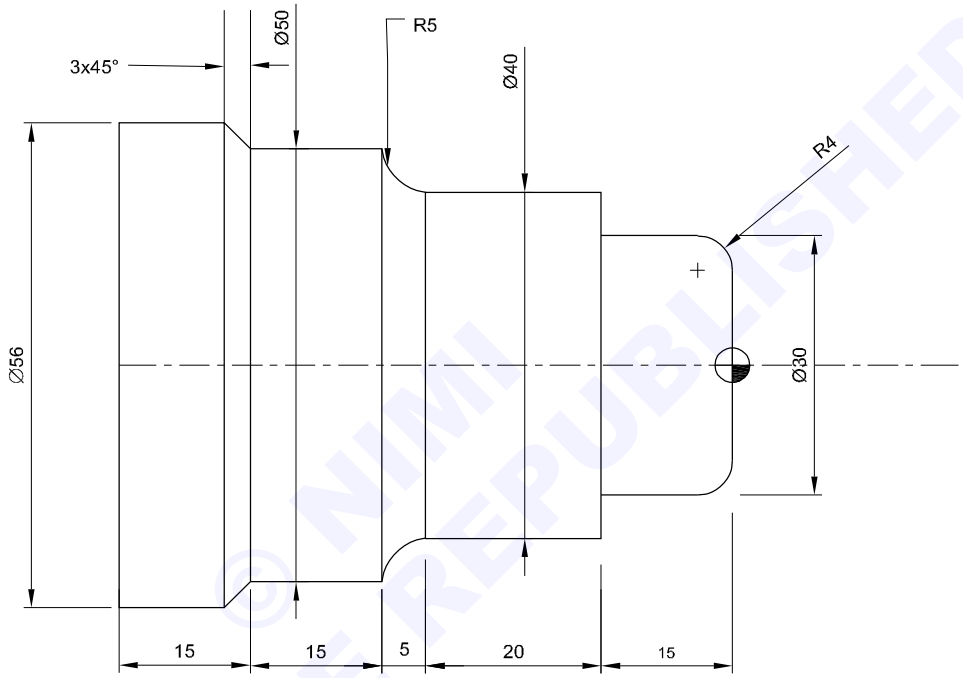
N80 M05

N85 M30

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.**



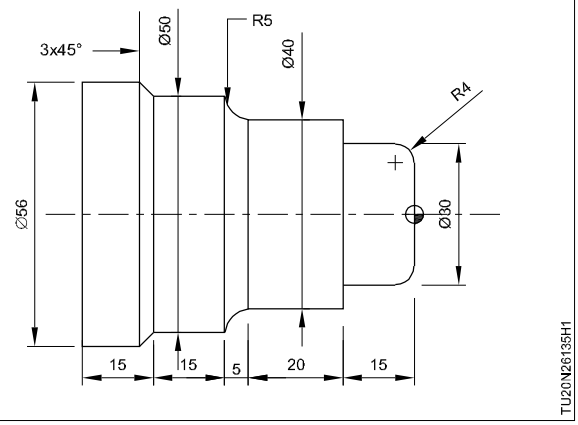
Job Sequence

- 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 75	-	Fe310	-	-	2.6.135
NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.
SCALE 1:1	TURNING OPERATION				DEVIATIONS ± 0.1	TIME :
					CODE NO. TU20N26135E1	

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

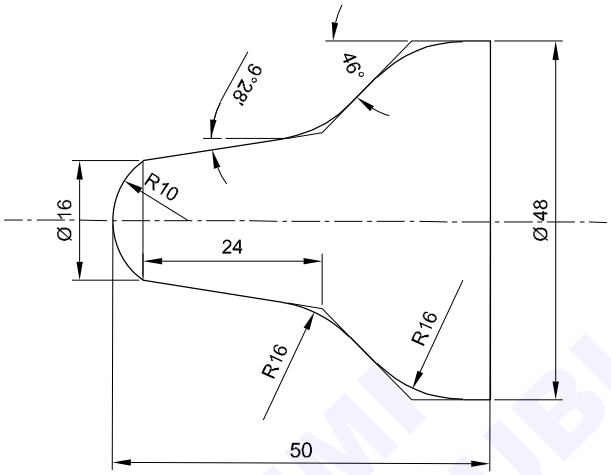
U : Finishing Allowance in 'X' axis

W : Finishing Allownace in 'Z' axis

F : Feed

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 work.**



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 contour 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

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

Instructor shall demonstrate on how to create contour programming

-	-		-	-	-	2.6.136
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. TU20N26136E1	

Contour Program

Enter the values as in Fig 1 to 6.

Fig 1

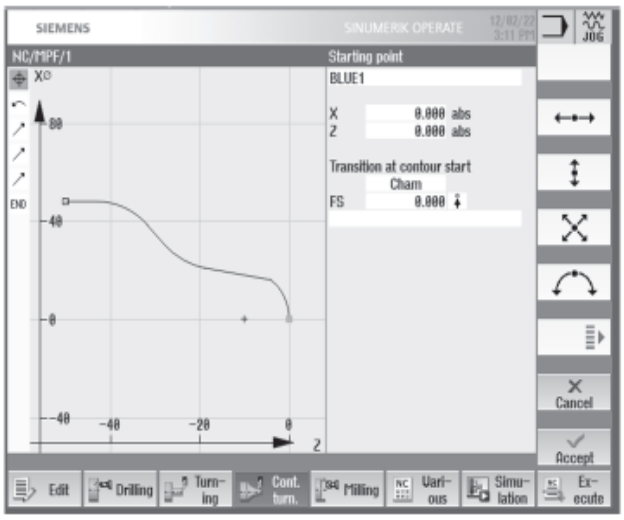


Fig 2

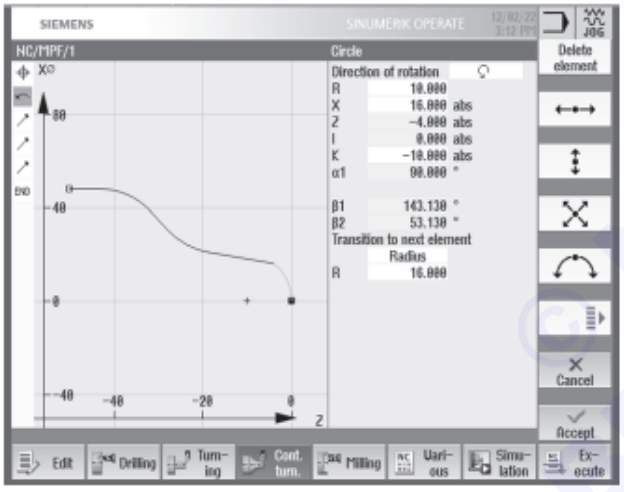


Fig 3

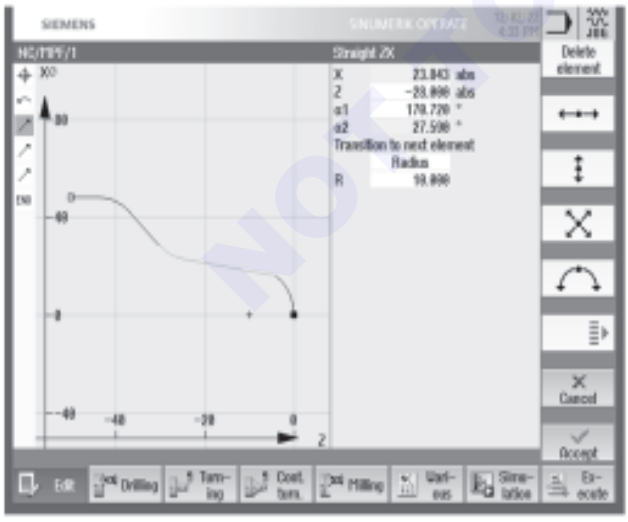


Fig 4

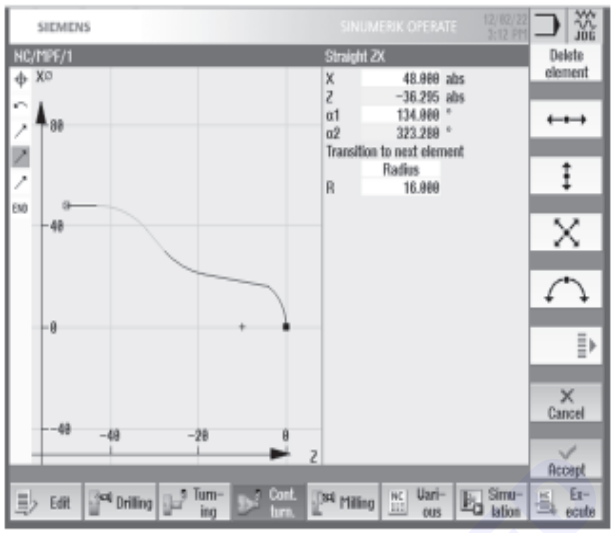


Fig 5

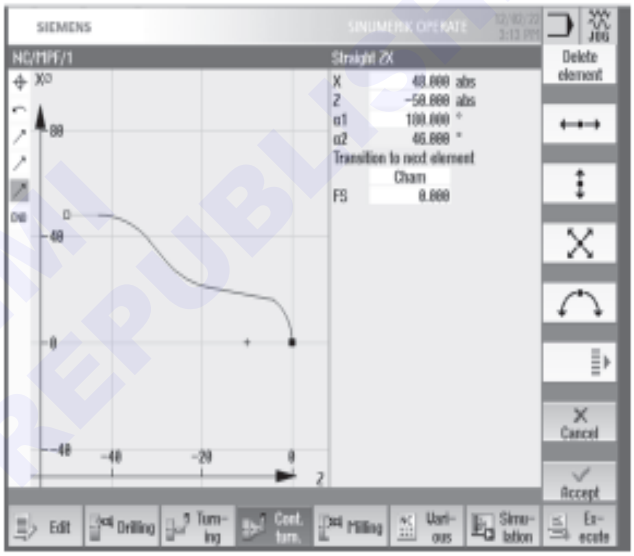
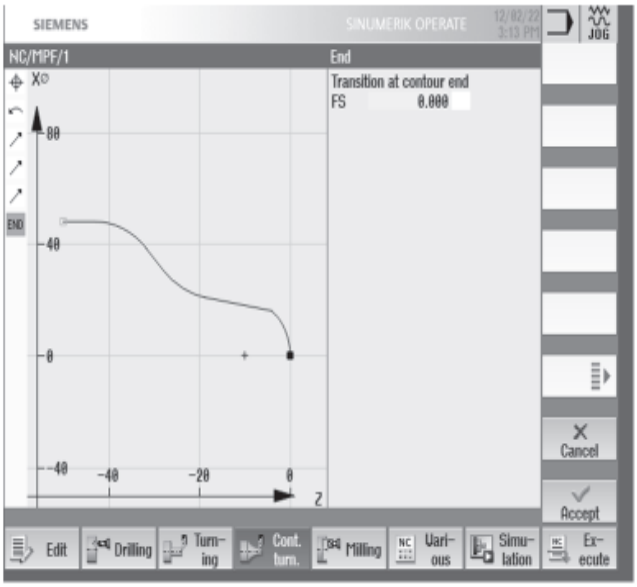
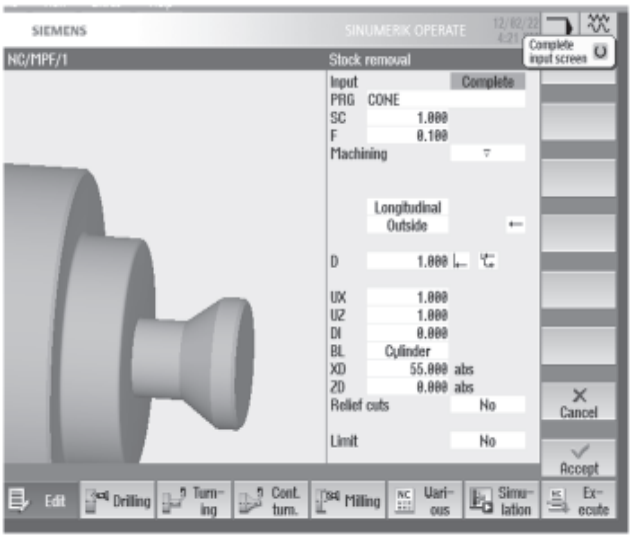


Fig 6



Stock removal details. (Fig 7)

Fig 7



CONTOUR PROGRAM

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__DIgK 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:

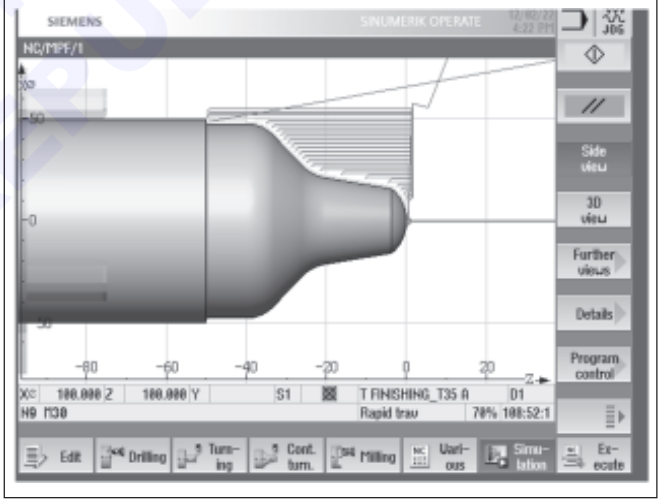
N7 G0 X100 Z100

N8 M05

N9 M30

Simulated view (Fig 8)

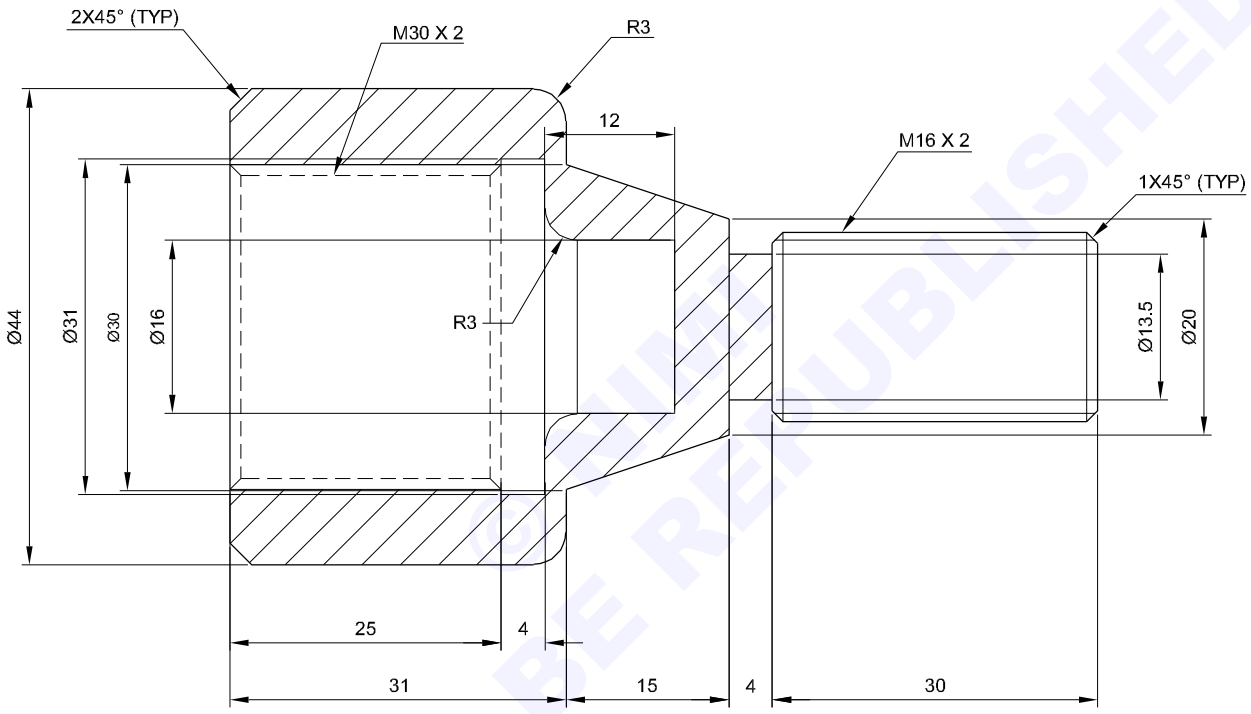
Fig 8

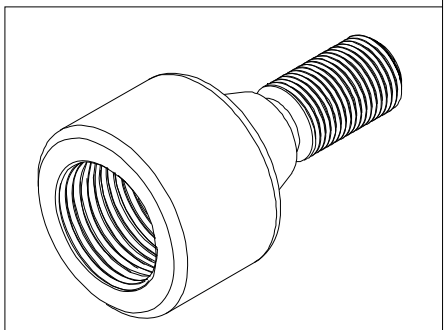


Machining parts on CNC lathe with parallel taper, step, radius, turning, grooving and threading

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.**





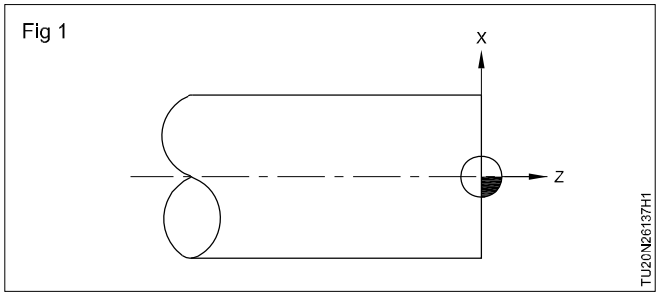
1	Ø50 X 110	-	ALUMINUM	-	1	2.6.137
NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.
SCALE 1:1	EXTERNAL AND INTERNAL TURNING OPERATION				DEVIATIONS ± 0.02	TIME :
					CODE NO. TU20N26137E1	

Job Sequence

1st Operation (External)

- Write the CNC program for external operations in fanuc and siemens control using stock removal grooving and threading cycles.
- 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

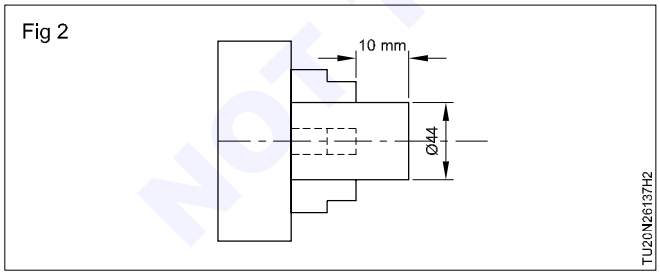


- Set the tool offset for rough turning finish turning grooving and threading tool.
- Check the tool offset
- Run the program in auto mode
- Check the dimensions of the work piece. It any error correct it by wear offset. And once again run the program and check the dimension
- Load the part program and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

2nd Operation (Internal)

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

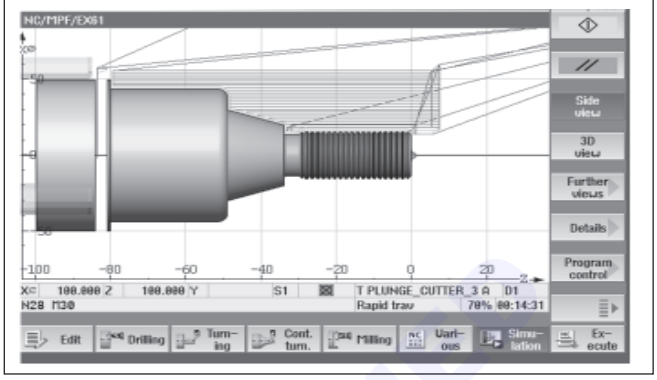
Fig 2



- 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 (Figs 3 to 7)

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

N31 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 4

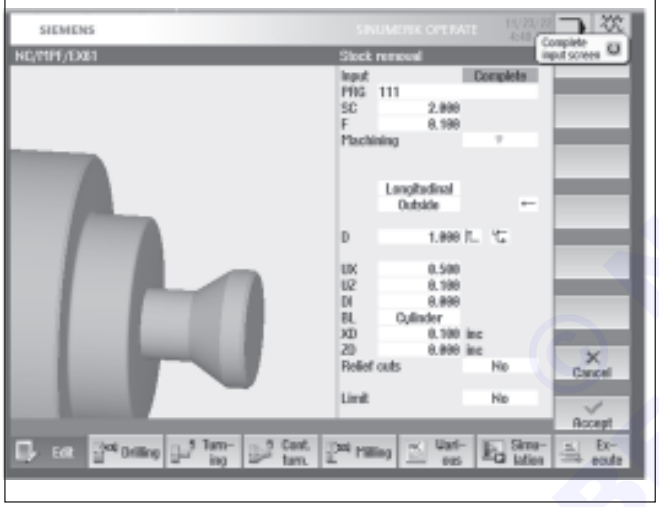


Fig 5

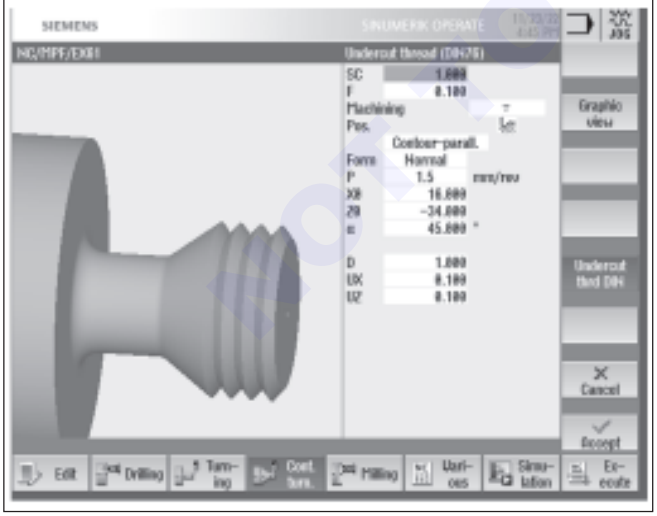


Fig 6

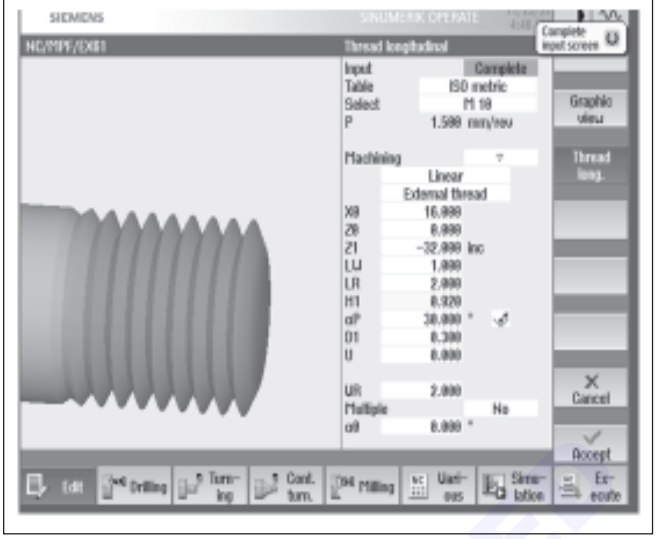
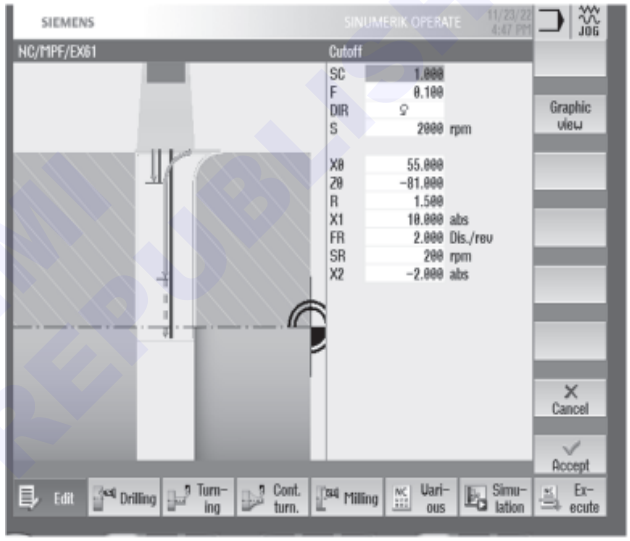
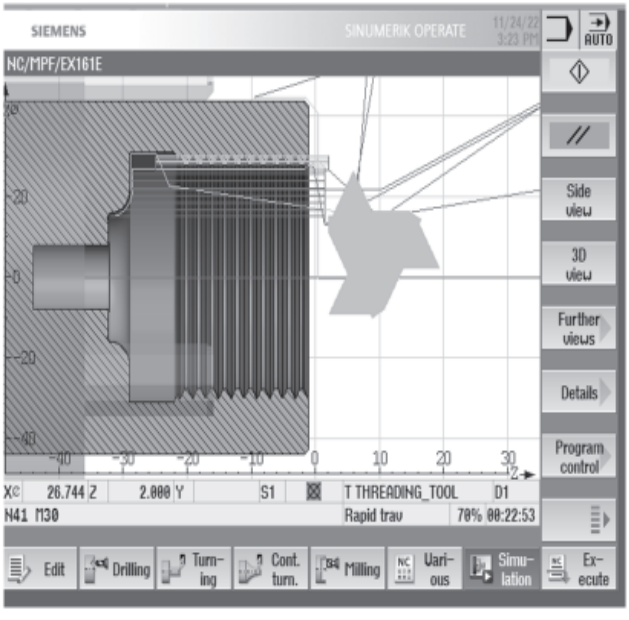


Fig 7



Internal Operation (Fig 8)

Fig 8

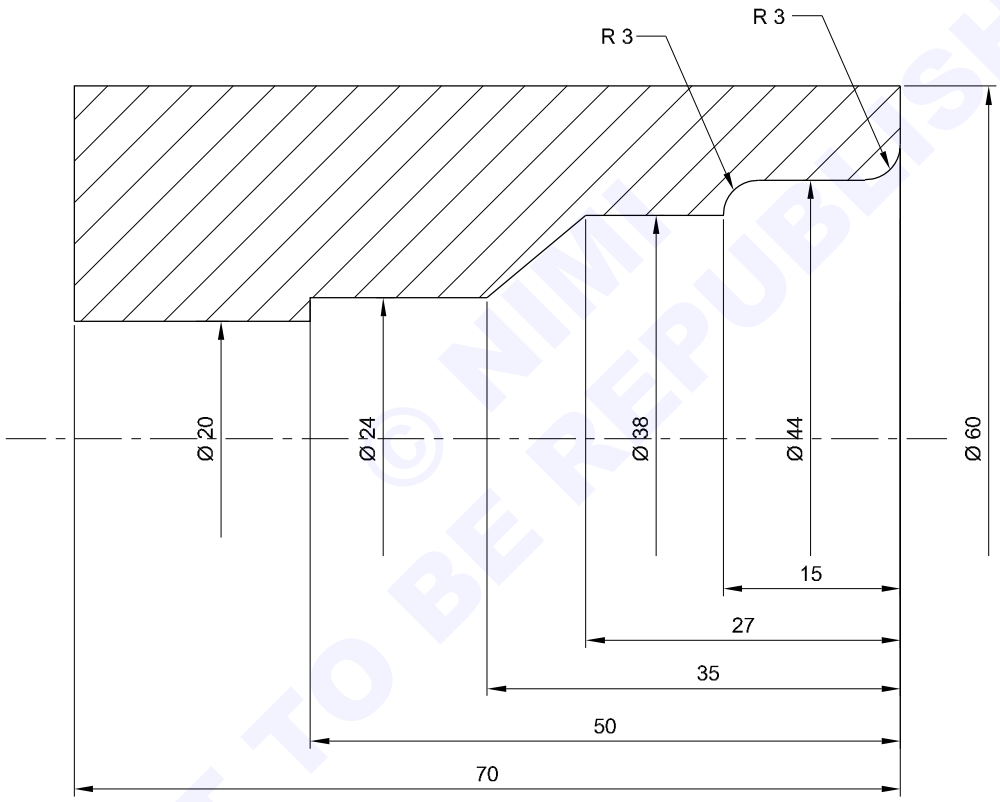


```
SUB PROGRAM 222
N1 G01 X30 Z0 F0.1
N2 G01 X27.744 Z-2
N3 G01 X27.744 Z-29
N4 X22 Z-29
N5 G02 X16 Z-32 CR=3
N6 G01 X16 Z-44
N7 M17
SUB PROGRAM 333
N1 G01 X30 Z0 F0.1
N2 G01 X27.744 Z-2
N3 G01 X27.744 Z-29
N4 X22 Z-29
N5 G02 X16 Z-32 CR =3
N6 G01 X16 Z-44
N7 M17
N1 G54 G90 G95 G71
N2 WORKPIECE(,,, "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_T35l"
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 l"
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
```

Carryout drilling & boring cycles in CNC turning

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 check the dimensions.**



1	PRE MACHINED Ø 60 X 70	--	Fe310	–	–	2.6.138
NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.
SCALE 1:1	DRILLING AND BORING				DEVIATIONS	TIME :
					CODE NO. TU20N26138E1	

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 G71
- 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 as per 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)

Nl G80 G40;

N2 G18 G90 G21 GS4 G99 G97;

N3 T0101 Sl000 M04; (TURNING TOOL)

N4 G00l X70.00 Z0.0;

N5 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 Ql0.0 Pl00 F0. I;

N15 G28 U0.0 W0.0 M05 T0300;

N16 T0606 S800 M04; (BORING TOOL)

N 17 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 R3.0;

N24 G01 X38.0 Z-33.0;

N25 G01 X24.0 Z-35.0;

N26 G01 X24.0 Z-50;

N27 G01 X20.0 Z-50.0;

N28 G00 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;

Geometry wear correction geometry and wear offset correction

Objectives: At the end of this exercise you shall be able to
- **set the geometry offset in fanuc controller**
- **set the tool wear offset.**

Job Sequence

Geometry offset

The G code 3290 is used to set the geometry offset in a fanuc controller. The geometry offset is a value that is used to compensate for the difference between the physical zero position of the machine and the programmed zero position of the workpiece. This difference can be caused by a number of factors, such as the position of the tool tip in the holder, the backlash in the machine's leadscrews, and the wear on the machine's components.

- Set the machine to the home position.
- Enter G code 3290 followed by X & Z offset.
- Press input button.
- Geometry offset will be set to the specified values.

Code snippet

G3290x0.005 Z0.1

Tool wear offset

- Enter MDI mode. To do this, press the MDI button on the control panel.
- Enter the G5013 code. This will bring up the wear offset setting screen.
- Enter the wear offset value. This is the amount of wear that you want to compensate for. For example, if the tool has worn 0.1 mm, you would enter 0.1.
- Press the enter key. This will save the wear offset value.

Additional information keep in mind

- The wear offset value is entered in millimeters.
- The wear offset value is applied to all subsequent movements of the tool.
- The wear offset value is not applied to canned cycles.
- The wear offset vaue is reset to zero when the machine is turned off.

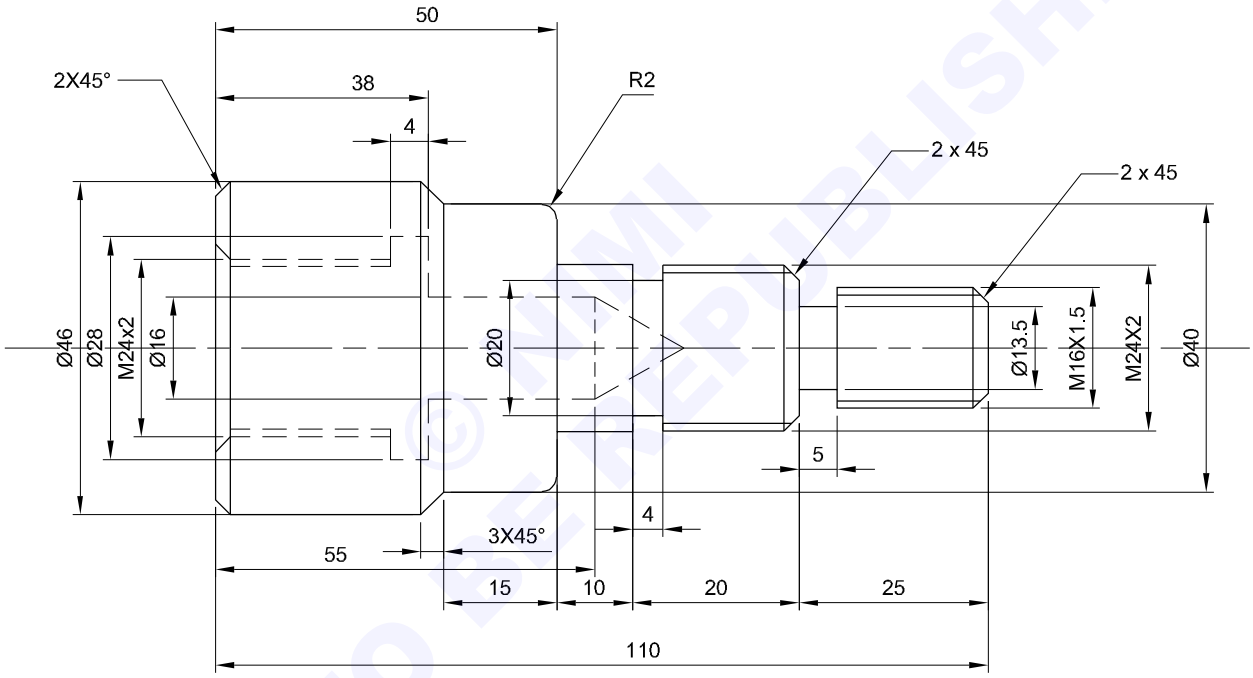
Example

- If the programmed size x 40.44, and machined size is 40.49, then 0.05 mm is increased due to tool wear to recitify this take the tool offset no: of wornout tool and enter the value u0.05.
- Tool offset no and wear offset no are same.

Produce components on CNC machine, involving turning OD & ID grooving, OD & ID threading

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

- **write the part programme**
- **set the machine with necessary tools**
- **run the machine to produce the job.**



ALL DIMENSIONS ARE IN mm

1	Ø48 x 115	-	MS ROD	-	-	2.6.140
NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.
SCALE 1:1	TURNING , THREADING AND GROOVING OPERATION				DEVIATIONS ± 0.02	TIME :
					CODE NO.TU20N26140E1	

Job Sequence

1st Operation

05349

Nl; [Turning]l

G28 U0 W0;

G97 S1200 M03;

T0101;

G00 X50.0 Z5.0 M08;

G01 Z3.0 F0.1;

G71 U1.0 R1.0;

G71 P1O Q20 U0.0 W0.0 F0.1;

N10 G01 X12.0 F0.1;

Z0.0;

X16.0 Z-2.0;

Z-25.0;

X20.0

X24.0 Z-27.0;

Z-55.0;

X36.0;

G03 X40.0 Z-57.0 R2 F0.1;

G01 X40.0 Z-70.0 F0.01;

G0I X46.0 Z-73.0 F0.1;

N20 G01 Z-75.0 F0.1;

G00 X60.0 Z10.0 M09;

G28 U0 W0;

M05;

M01;

N2; [OD Grooving]

G28 U0 W0;

G97 S700 M03;

T0303;

G00 X17.0 Z5.0 M08;

G01 Z-23.0 F0.1;

G75 R1.0;

G75 X13.5 Z-25.0 P500 Q2000 F0.08;

G00 X25.0;

G01 Z-44.0 F0. 1;

G75 R1.0;

G75 X20.0 Z-45.0 P500 Q1000 F0.08;

G00 X50.0;

Z10.0 M09;

G28 U0 W0;

M05;

M01;

N3; (Threading)

G28 U0 W0;

G97 S600 M04;

T0505;

G00 X17.0 Z5.0 M08;

G01 Z3.0 F0.1; [M16x1.5]

G92 X16.0 Z-22.0 Fl.5;

X15.8;

X15.6;

X15.4;

X15.2;

X15.0;

X14.8;

X14.6;

X14.4;

X14.2;

X14.1;

X14.065;

G00 X 25.0;

Z-23.0; [M24 x 2]

G76 P030060 Q300 R0.02;

G76 X 21.42 Z42.0 P1300 Q300 F2;

G00 X 60.0;

Z10.0 M09;

G28 U0 W0;

M05;

M01;

M30;

IInd Operation Reverse the job in soft jaws clamping on Ø40

O5350;

G28 U0 W0;

G97 S1200 M03;

T0101;

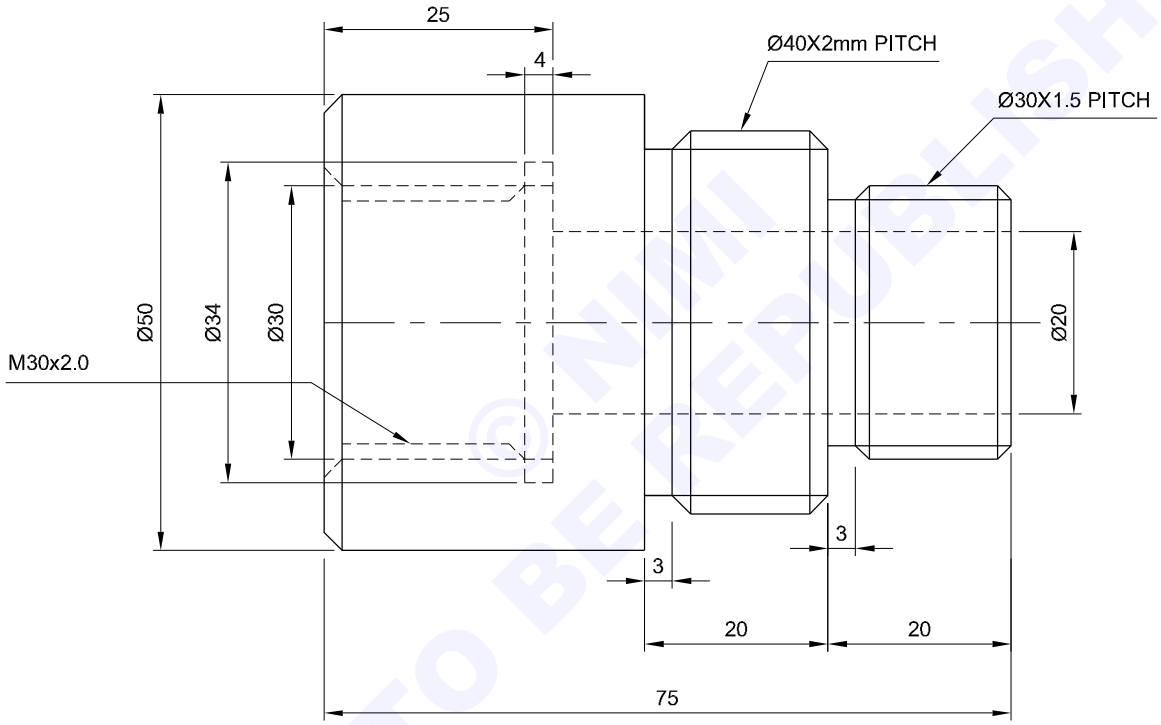
G00 X55.0 Z5.0 M08;

G01 Z3.0 F0.1;

```
G71 U1.0 RI1.0;
G71 P10 Q20 U0.0 W0.0 F0.1
N10 G01 X0.0 F0.1;
Z0.0;
X42.0;
X46.0 Z-2.0;
N20 G01 Z-40.0 F0.1;
G00 X 60.0 Z10.0 M09;
G28 U0 W0;
M05;
M01
N2; (Centre Drilling);
G28 U0 W0;
G97 S1800 M03;
T0202;
G00 X0  Z5.0 M06;
G01 Z-7.0 F0.1;
G00 Z10.0 M09;
G28 U0 W0;
M05;
M01;
N3; [∅16 Drill ]
G28 U0 W0;
G97 S1000 M04;
T0404;
G00 X0.0 Z5.0 M08;
G74 R2.0;
G74 Z-55.0 QI000 F0.1;
G00 Z10.0 M09;
G28 U0 W0;
M05;
M01;
N4; [Boring Tool]
G28 U0 W0;
G97      S1200   M03;
T0606;
G00 X15.0 Z5.0  M08;
G71 U1.0 R1.0;
G71 P30 Q40 U0.0 W0.0 F0.1;
N30 G01 X24.0 Z0.0 F0.1;
X21.42  Z-2.0;
Z-38.0;
X16.0;
N40 G01 Z2.0 F0.1;
G00 Z10.0 M09;
G28 U0 W0;
M05;
M01;
N5; (ID Grooving 4mm width)
G28 U0 W0;
G97 S600 M03;
T0707;
G00 X17.0 Z5.0  M08;
G01 Z-38.0 F0.1;
G75  R1.0;
G75 X 28.0 Z-38.0 P400 F0.08;
G00 X17.0;
Z10.0 M09;
G28 U0 W0;
M05;
M01;
N6 [ID Threading]
G28 U0 W0;
G97  S600  M03;
T0808;
G00 X 20.0 Z5.0 M08;
G76 P030060 Q20.0       R20;
G76 X24.0 Z-35.0 P3000 Q300 F2.0;
G00 X15.0;
Z10.0    M09;
G28 U0 W0;
M05;
M01;
M30
```

Produce components by involving turning operation and part programme

Objectives: At the end of this exercise you shall be able to
- **input the part programme and simulate the programme**
- **set the machine necessary tools**
- **run the machine to produce the job.**



ALL CHAMFER 2X45°

1	Ø48 x 115	-	MS ROD	-	-	2.6.141
NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.

SCALE 1:1	TURNING OPERATION	DEVIATIONS ± 0.1	TIME :
		CODE NO.TU20N26141E1	

Job Sequence

1st Operation

```
05349
NI; [Turning]I
G28 U0 W0;
G97 S1200 M03;
T0101;
G00 X50.0 Z5.0 M08;
G01 Z3.0 F0.I;
G71 U1.0 R1.0;
G71 P10 Q20 U0.0 W0.0 F0.1;
N10 G01 X28.0 F0.1;
Z0.0;
X30.0 Z-2.0;
Z-20.0;
X38.0
X40.0 Z-22.0;
Z-40;
X50;
G01 X50 Z-50 F0.1;
N20 G01 Z-52 F0.1;
G00 X60.0 Z10.0 M09;
G28 U0 W0;
M05;
M01;
N2; [OD Grooving - 2mm width]
G28 U0 W0;
G97 S700 M03;
T0202;
G00 X32.0 Z5.0 M08;
G01 Z-19.0 F0.1;
G75 R1.0;
G75 X28.0 Z-20.0 P500 Q2000 F0.08;
G00 X42.0;
G01 Z-39.0 F0.1;
G75 R1.0;
G75 X38.0 Z-40.0 P500 Q2000 F0.08;
G00 X60.0;
Z10.0 M09;
G28 U0 W0;
M05;
M01;
N3; (Threading)
G28 U0 W0;
G97 S600 M04;
T0303;
G00 X32.0 Z5.0 M08;
G01 Z3.0 F0.1; [M30x1.5]
G92 X30.0 Z-18.0 FI.5;
X29.8;
X29.6;
X29.4;
X29.2;
X28.0;
X28.8;
X28.6;
X28.4;
X28.2;
X28.1;
X28.065;
G00 X42.0;
Z-19.0; [M40 x 2mm]
G76 P030060 Q300 R0.02;
G76 X 37.42 Z-38 P1300 Q300 F2;
G00 X60.0;
Z10.0 M09;
G28 U0 W0;
M05;
M01;
M30;
```

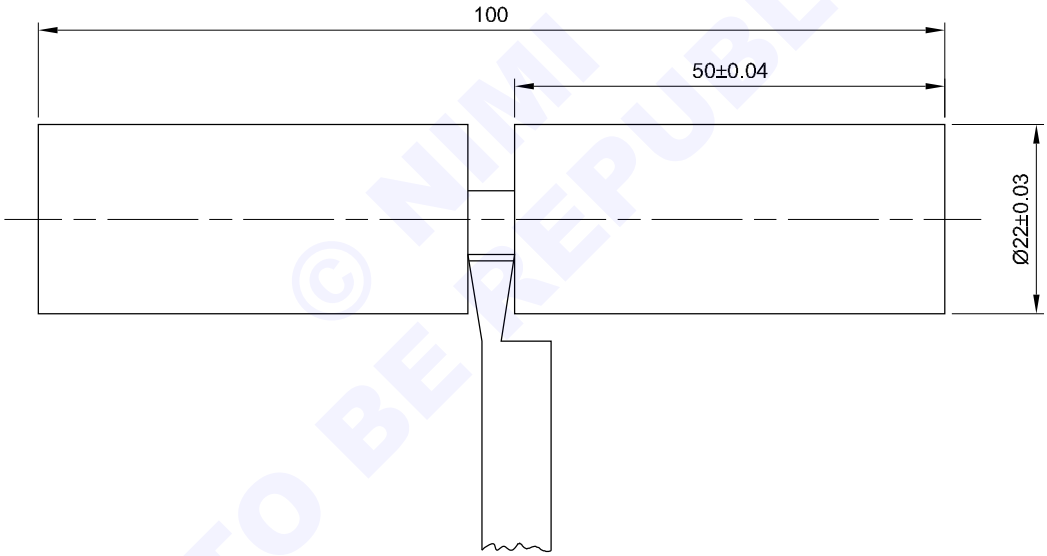
IInd Operation Reverse the job in soft jaws clamping on Ø50

```
O5350;
N1; [Turning]
G28 U0 W0;
G97 S1200 M03;
T0101;
G00 X60.0 Z5.0 M08;
G01 Z3.0 F0.1;
G71 U1.0 RI1.0;
```

```
G71 P10 Q20 U0.0 W0.0 F0.1
N10  G01 X0.0  F0.1;
Z0.0;
X48.0;
X50.0     Z-2.0;
N20 G01 Z-35.0  F0.1;
G00  X60.0 Z10.0 M09;
G28 U0  W0;
M05;
M01
N2; (Centre Drilling);
G28 U0  W0;
G97 S1800          M03;
T0404;
G00 X0.0 Z5.0 M06;
G01 Z-7.0 F0.1;
G00 Z10.0 M09;
G28 U0  W0;
M05;
M01;
N3; [Drilling ∅20mm]
G28  U0 W0;
G97 S1000 M04;
T0505;
G00 X0.0 Z5.0 M08;
G74 R2.0;
G74 Z-75.0 Q1000 F0.1;
G00 Z10.0 M09;
G28 U0  W0;
M05;
M01;
N4; [Boring]
G28 U0  W0;
G97 S1200 M03;
T0606;
G00 X 15.0 Z5.0 M08;
G71 U1.0 R1.0;
G71 P30 Q40 U0.0 W0.0 F0.1;
N30 G01 X 30.0 Z0.0 F0.1;
X27.8     Z-2.0;
Z-25;
X20;
N40 G01 Z2.0 F0.1;
G00 Z10.0 M09;
G28 U0  W0;
M05;
M01;
N5; (ID Grooving 4mm width)
G28 U0  W0;
G97 S600 M03;
T0707;
G00 X 18.0 Z5.0 M08;
G01 Z-25 F0.1;
G75  R1.0;
G75 X 34.0 Z-25 P400 F0.08;
G00 X18.0;
Z10 M09;
G28 U0  W0;
M05;
M01;
N6 [ID Threading]
G28 U0  W0;
G97  S600  M03;
T0808;
G00 X 20.0  Z5.0 M08;
G76 P030060 Q20.0        R20;
G76 X 30.0 Z-22 P3000 Q300 F2.0;
G00 X15.0;
Z10.0 M09;
G28 U0  W0;
M05;
M01;
M30
```

Part off: part pogramme in CNC machines

Objectives: At the end of this exercise you shall be able to
- **write the part programme for part of work**
- **input the part programme into the machine**
- **set the machine with necessary tools**
- **run the machine to produce the job.**



1	Ø24x100	-	MS ROD	-	-	2.6.142
NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.

SCALE 1:1	PARTING OFF OPERATION	DEVIATIONS : ± 0.02	TIME :
		CODE NO.TU20N26142E1	

Job Sequence

- Study the part drawing
- Prepare the CNC program
- Hold the job on chuck by projection 60mm length in turning and part off.
- Enter the program
- Take the offset
- Run the program in auto mode
- Take the part off piece and check the dimension
- Remove the job and switch OFF the machine

Program

O0010

N1 (Turning)

G28 U0 W0;

G97 S1200 M03;

T0101;

G00 X30.0 Z5.0 M08;

Z0.0;

G01 Z24.0 Z2.0;

G01 Z-55.0 F0.1;

G00 X26.0 Z2.0

X23.0;

G01 Z-55.0 F0.1;

G00 X26.0 Z2.0;

X22.015;

G01 Z-55.0 F0.1;

G00 X30.0 Z5.0 M09;

G28 U0 W0;

M05;

M01;

N2; [3mm parting tool]

G28 U0 W0;

G97 S600 M03;

T0303;

G00 X23.0 Z5.0 M08;

G01 Z-53.0 F0.1;

G75 R1.0;

G75 X-1.0 Z-53.0 P400 Q0 F0.1;

G00 X30.0;

Z10.0 M09;

G28 U0 W0;

M05;

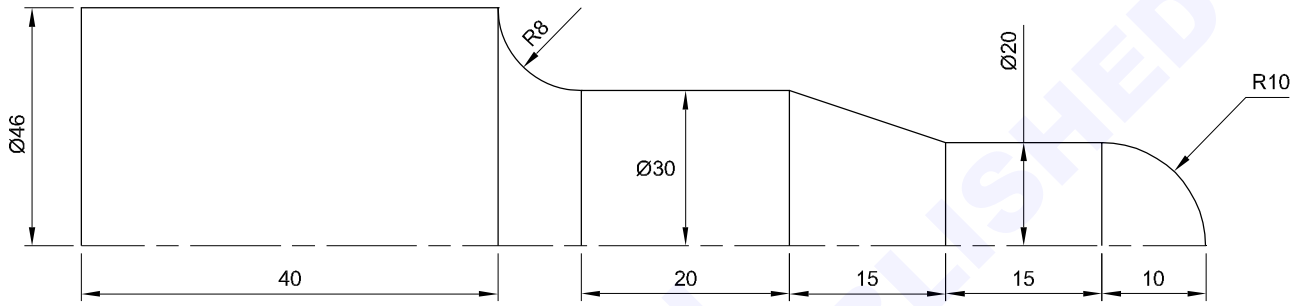
M01;

M30;

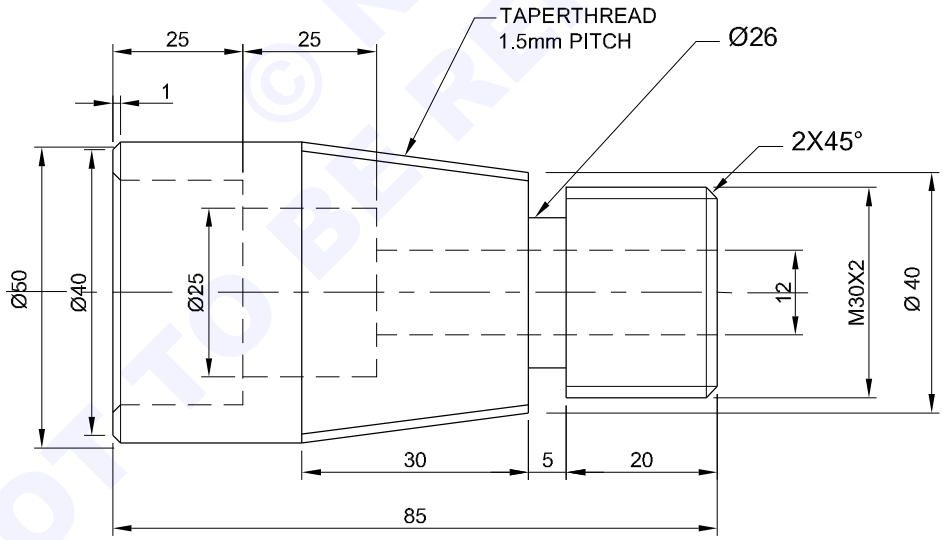
Produce job involving profile turning, threading, taper and boring

Objectives: At the end of this exercise you shall be able to
- **prepare a part programme for given sketch**
- **input the part programme into the machine**
- **produce the job in SBL/AUTO mode.**

TASK : 1



TASK : 2



2	Ø55 x 90	-	MS ROD	-	-	2.6.143
1	Ø100 x 110	-	MS ROD	-	-	2.6.143
NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.

SCALE 1:1	**PROFILE TURNING**	DEVIATIONS ± 0.02	TIME :
		CODE NO. TU20N26143E1	

Job Sequence

Program

%

04133

N10 G21 G40 G90;

N15 G28 U0 W0;

N20 G50 S1500 T0101;

N25 G96 S400 M03;

N30 G00 G42 X52 Z5;

N35 G71 U1 R2;

N40 G71 P45 Q75 U1 W1 F0.1;

N45 G01 X0;

N50 Z0.0;

N55 G03 X20 Z-10 R10;

N60 G01 Z-25;

N65 G01 X30 Z-40;

N70 Z-60;

N75 G02 X46 Z-68 R8;

N80 G01 Z-108;

N85 G70 P45 Q80 F0.05;

N90 T0100 M09

N95 G28 U0 W0;

N100 M05;

N105 M30;

- Study the part drawing and list the tools required
- Preapre the CNC program.
- Get it checked by the instructor.
- Check the raw material size and conform with the part drawing,
- Hold the job on chuck by projection 55 mm length in turning.
- Enter the part programme in CNC machine (or) transfer the programme by simulator to CNC machine.
- Set the tool to the required turret station.
- Measure work offset in X and Z direction and enter the work offset page say G54/G55.
- Measure the tool offset in all tools in X and Z direction and enter the tool geometry offset page.
- Enter the tool type and tool nose radius
- Check the work offset and tool offset.

If there is any mistake correct the programme ask your instructor for guidance

- Run the programme in single block by setting the offset away from the work zero
- Observe the spindle direction speed tool position carefully.
- Run the programme in auto mode in original tool and work set.
- Check the dimension. If any correction changes the wear offset and run the programme.
- Check the dimension and surface finish.
- Remove he job from the machine. Check all the dimension once again.
- Switch off the machine.

TASK 2:

% (1st Operation)

000125

N1;

G28 U0 W0;

G96 S250 M03;

G50 S1500;

T0101;

G00 X55.0 Z5.0 M08;

G71 U1.0 R1.0;

G71 P10 Q20 U0.0 W0.0 F0.0;

N10 G01 X0.0 F0.1;

Z0.0;

X48.0;

X50.0 Z-1.0;

N20 G01 Z-50.0 F0.1;

M09;

G28 U0 W0;

M05;

M01;

N2; (Centre Drill)

G28 U0 W0;

G97 S1800 M04;

T0202;

G00 X0.0 Z5.0 M08;

G01 Z-7.0 F0.1;

G00 Z10.0 M09;

G28 U0 W0;

M05;

M01;

N3; (12 mm Drill)
G28 U0 W0;
G97 S1500 M04;
T0404;
G00 X0.0 Z5.0 M08;
G74 R2.0;
G74 Z-88.0 QI000F0.I;
G00 Z10.0 M09;
G28 U0 W0;
M05;
M01;
N4;
G28 U0 W0;
G97 S1000 M03;
T0606;
G00 X11.0 Z5.0 M08;
G71 U0.5 R1.0;
G71 P.30 Q40.0 U0.0 W0.0 F0.1;
N30 G01 X 25.0 F0.1;
Z-50.0;
X40.0;
Z-1.0;
N40 G01 X 42.0 Z0.0 F0.1;
G00 X 10.0 Z5.0 M09;
G28 U0 W0;
M05;
M01;
M30;

Programme for Grooving threading & taper threading

00126;
N1;
G28 U0 W0;
G97 S1200 M03;
T0101;
G00 X55.0 Z5.0 M08;
G71 U10 R1.0;
G71 P10 Q20 U0.0 W0.0 F0.1;
N10 G01 X 0.0 F0.1;
Z0.0;
X26.0;
X30.0 Z-2.0;
Z-25.0;
X40.0;
X50.0 Z-55.0;
N20 G01 Z-65.0 F0.1;
G00 X 60.0 Z5.0 M09;
G28 U0 W0;
M05;
M01;
N2; (Grooving 3mm)
G28 U0 W0;
G97 S600 M03;
T0303;
G00 X31.0 Z5.0 M08;
G01 Z-23.0 M08;
G75 R1.0;
G75 X26.0 Z-25.0 P600 Q2000 F0.08;
G00 X60.0;
Z10.0 M09;
G28 U0 W0;
M05;
M01;
N3; (Threading)
G28 U0 W0;
G97 S600 M04;
T0505;
G00 X 31.0 Z5.0 M08;
G01 Z4.0 F0.I;
G76 P030060 Q20 R0.02;
G76 X27.42 P13000 Q300 F2.0;
G00 X60.0;
G01 Z-22.0 F0.1;
G76 P030060 Q20 R0.02;
G76 X38.58 Z-57.0 P097S Q200 R-5.5 F1.5;
G00 X60.0 Z5.0 M09;
G28 U0 W0;
M05;
M01;
M30;

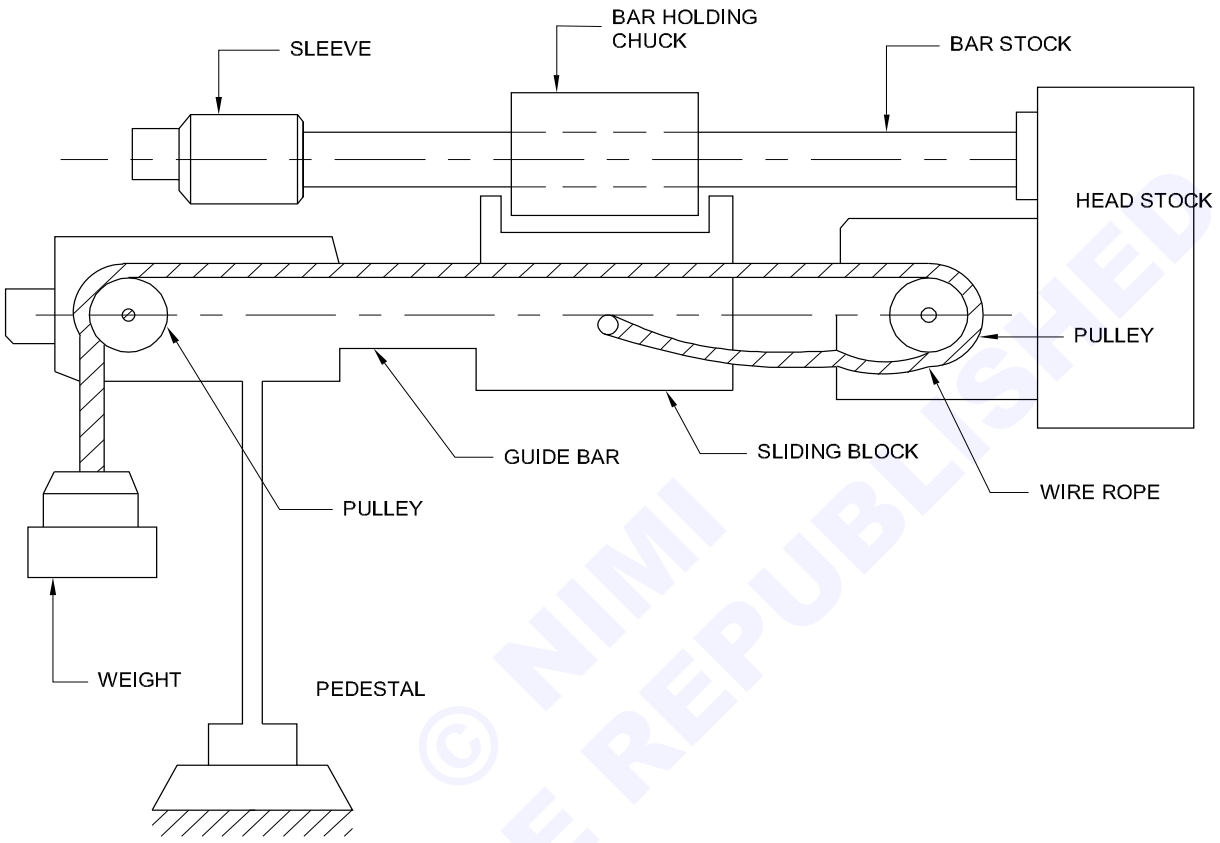
Calculate R parameter for tapered threading on FANUC with G76 threading cycle.

R = (Start Diameter - End Diameter) / 2

Demo on machine bar feeding system (video)

Objective: At the end of this exercise you shall be able to
- **identify a bar feeding system.**

Fig 1



Job Sequence

The instructor guide to trainee on website in the revelent bar feeding work in CNC M/C (video)

Advantages of bar feeding mehanism

- Reduce the machine tool operator's time and effort by auto feeding raw material.
- Support bar stock to reduce vibrations that can tranfer to the cutting zone and create dimension and surface finish problems.
- Enable the machine tool to operate at optimum spindle speeds.
- Minimise change - over times.
- Reduce material wastage.
- Provide consistent, reliable operation that enables higher volume of production.

Optional

Instructor to arrange a visit to an industry, where the bar feeding facility is a available, to collect details on bar feeding system

↓

(Or alternately)

↓

Arrange video clipping of bar feeding mechanism, and CNC machines, using bar feeding to get the trainees acquainted with the technology

↓

Guide the trainees to prepare a flow diagram collect sketch of a bar feeding a system from the networking

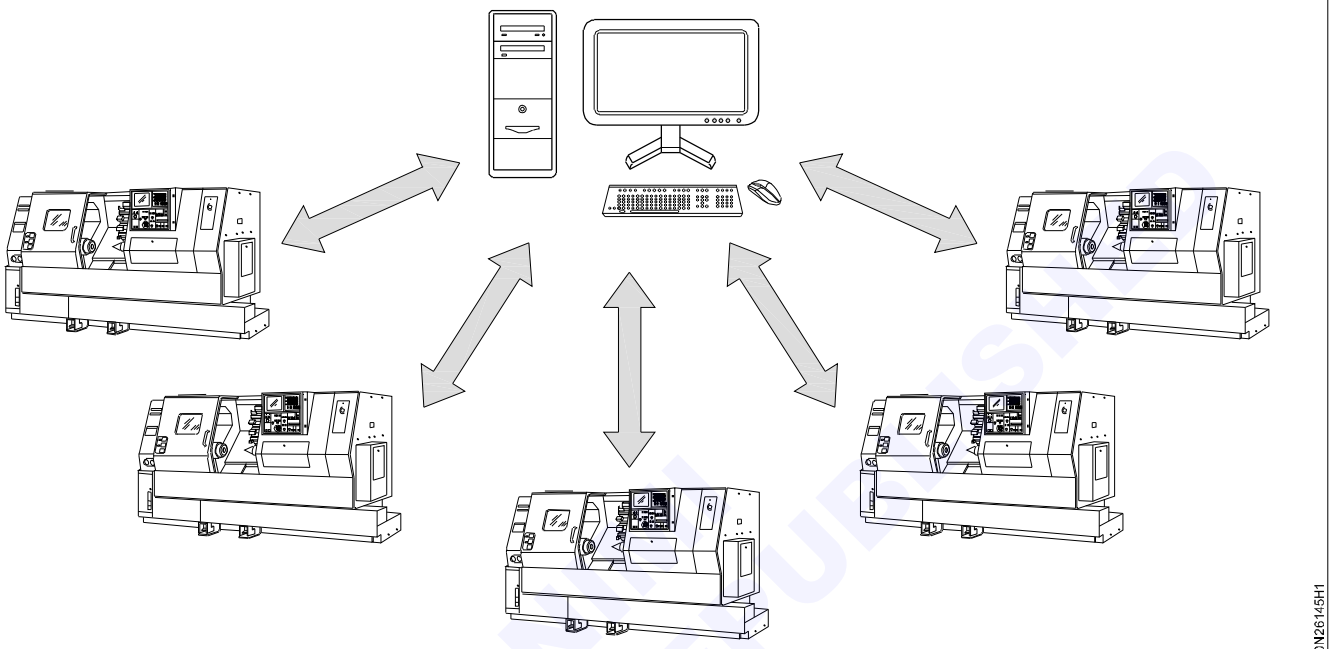
↓

Instructor to explain through a chart/photograph

DNC system setup (optional)

Objective: At the end of this exercise you shall be able to
- **setup direct numerical control system.**

Fig 1



Job Sequence

[Learn the instructor by viewing videos of DNC system]

Switch computer

↓

Select google website

↓

Select DNC setup

↓

View the video on DNC system

↓

Record the observations

↓

Discuss with you instructor on transfer of command eic.

Detailed steps for setting up a CNC machine in DNC mode

1 Connect the CNC machine to the DNC server
   - If you are using a serial connection, connect the CNC machine's serial port to the DNC server's serial port.
   - If you are using an ethernet connection, connect the CNC machine's ethernet port to the DNC server's ethernet port.
   - If you are using a USB cable, connect the CNC machine's USB port to the DNC server's USB port.

2 Configure the CNC machine for DNC mode
   - The specific steps for this will vary depending on the make and model of the machine.
   - In general, you will need to set the following parameters in the machine's control system.
     a The DNC server IP address
     b The DNC server port number
     c The DNC mode
   - Once you have set these parameters, you will need to save the settings.

3 Load the DNC software on the server

- There are a number of DNC software packages available, so you will need to choose one that is compatible with your CNC machine and the DNC server.
- Once you have chosen a DNC software package, you will need to install it on the server.

4 Create a DNC database

- The DNC database is where the CNC programs will be stored.
- The DNC software will create the database for you, but you will need to specify the location of the database file.

5 Transfer the CNC programs to the DNC database.

- You can transfer the CNC programs to the DNC databse using a varity of methods, such as FTP, SFTP, or SCP.
- Once you have transferred the CNC programs to the database, you will need to update the DNC software so that it knows where the programs are located.

6 Start the DNC software

- Once you have started the DNC software, it will monitor the CNC machine and transfer programs to the machine as needed.

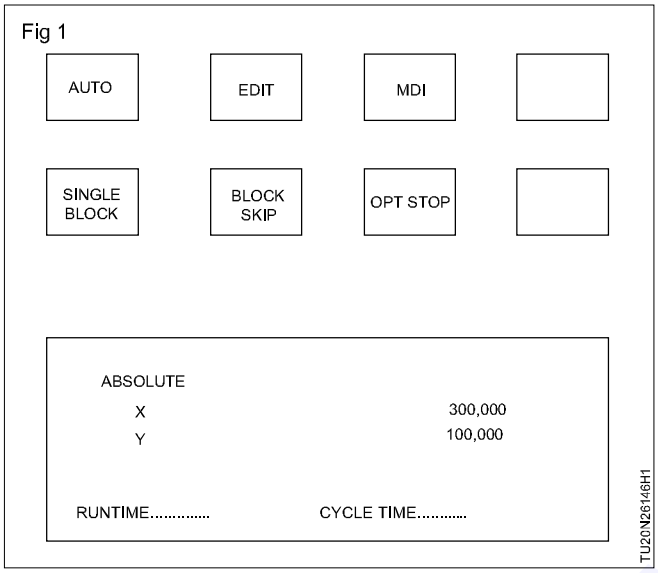
Here are some additional tips for settting up a CNC machine in DNC mode:

- Make sure that the CNC machine and the DNC server are properly configured.
- Use a secure connection between the CNC machine and the DNC server.
- Back up the DNC database regularly.
- Test the DNC system before using it in production.

Run the machine on DNC mode (optional)

Objective: At the end of this exercise you shall be able to
- **run the machine in DNC mode.**

Fig 1



Job Sequence

Step 1: Set the CNC machine to DNC mode

1 On the CNC machine's control panel, press the menu button.

2 Scroll down to the DNC menu and select it.

3 Ente the code that is specified in the machine's operating manual to set the machine to DNC mode.

Step 2: Connect the CNC machine to the computer or DNC device.

1 If you are using a network connection, make sure that the CNC machine and the computer are both connected to the same network.

2 If you are using a dedicated cable, connect the cable to the CNC machine and the computer.

Step 3: Transfer the CNC programs to the CNC machine.

1 Use a DNC software application to transfer the CNC programs to the CNC machine.

2 Alternatively, you can manually transfer the files by copying them to a USB drive and then inserting the USB drive into the CNC machine.

Step 4: Start the CNC machine.

1 Once the CNC programs have been transferred to the CNC machine, you can start the CNC machine.

2 The CNC machine will now begin executing the CNC programs that have been transferred to it.

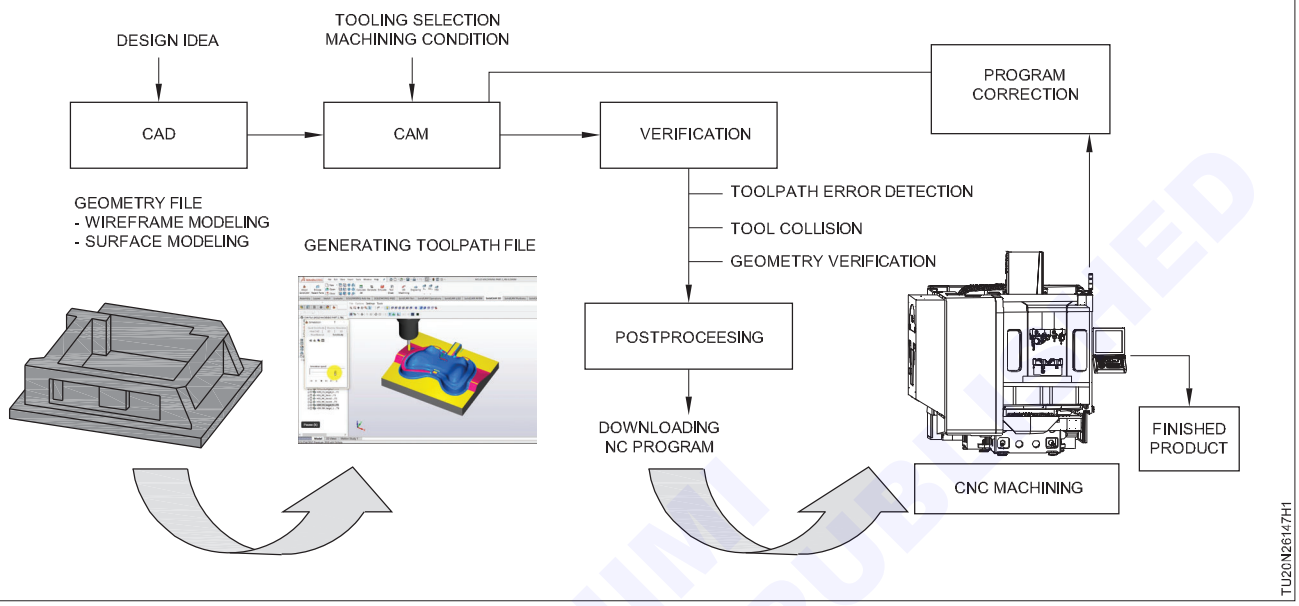
Here are some additional considerations when running a CNC machine in DNC mode:

- The DNC software application that you use should be compatible with the make and model of the CNC machine.
- The network connection that you use should be reliable and have enough bandwidth to support the transfer of CNC programs.
- The dedicated cable that you use should be of high quality and should be properly terminated.
- The CNC machine should be properly configured for DNC mode.

CAM programme execution (optional)

Objectives: At the end of this exercise you shall be able to
- **generate NC code for a CAD model**
- **load NC code in CNC machine and execute.**

Fig 1



Job Sequence

- Invoke the Auto CAD software
- Create CAD model of part to be machined.
- Import CAD model into CAM software.
- Select cutting tools and operations to be used.
- Calculate toolpaths.
- Generate NC code.
- Load NC code into CNC machine.
- Start CNC machine
- Monitor CNC machine operation
- Check machined part
- End

Skill Sequence

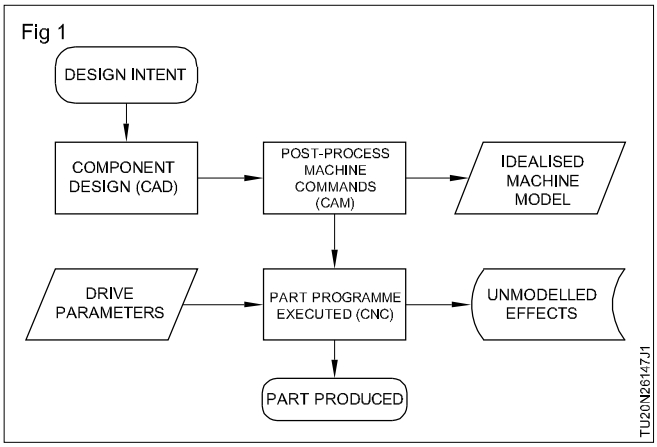
Flow chart

Objective: This shall be help to
- **follow the steps in CAD, CAM and CNC machining process.**

Here is a more detailed explanation of the flow chart:

- **Create CAD model of part to be machined:** This is the first step in the process. The CAD model is a digital representation of the part that will be machined. It is created using a CAD software package such as SolidWorks, AutoCAD, or Inventor.
- **Import CAD model into CAM software:** The CAD model is then imported into CAM software. The CAM software will use the CAD model to generate the toolpaths for the CNC machine.
- **Select cutting tools and operations to be used:** The CAM software will have a library of cutting tools that can be selected for the job. The user will also need to specify the cutting operations that will be used, such as milling, drilling, and tapping.
- **Calculate toolpaths:** The CAM software will use the CAD model and the selected cutting tools and operations to calculate the toolpaths. The toolpaths will specify the path that the cutting tools will follow to machine the part.

Fig 1



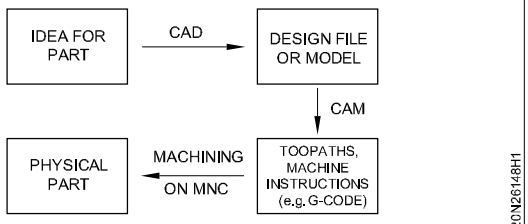
- **Generate NC code:** The CAM software will generate the NC code that will be used by the CNC machine. The NC code is a series of instructions that tell the CNC machine how to move the cutting tools.
- **Load NC code into CNC machine:** The NC code can be loaded into the CNC machine using a USB drive or an Ethernet connection.
- **Start CNC machine:** Once the NC code is loaded into the CNC machine, it can be started. The CNC machine will then follow the instructions in the NC code to machine the part.
- **Monitor CNC machine operation:** The CNC machine operation should be monitored to ensure that the part is being machined correctly. If any problems are encountered, the CNC machine operation can be stopped and the NC code can be modified.
- **Check machined part:** Once the CNC machine operation is complete, the machined part should be checked to ensure that it meets the required specifications.

This is just a general overview of the CAD-CAM-CNC machine operation sequence. The specific steps involved may vary depending on the CAD software, CAM software, and CNC machine being used.

Data input output on CNC machine

Objective: At the end of this exercise you shall be able to
- **data input and output practice of CNC machine.**

Fig 1



Job Sequence

Data input the steps on which data are input to run a CNC machine practically

1 Power on the CNC machine and wait for it to boot up.

2 Select the appropriate data input method.

- Punched tape· This is an older method that uses a long strip of paper with holes punched in it to store data.
- Floppy disk: This is a more modern method that uses a small, portable disk to store data.
- USB drive This is another modern method that uses a small, portable drive to store data.
- Network: This method allows you to transfer data from a computer to the CNC machine over a network.

3 Load the data into the CNC machine

- If you are using punched tape, you will need to insert the punched tape into the CNC machine's tape reader
- If you are using a floppy disk, you will need to insert the floppy disk into the CNC machine's floppy disk drive
- If you are using a USB drive, you will need to connect the USB drive to the CNC machine's USB port.

4 Input geometric data, tool data and motion data

Geometric data

- Workpiece dimensions
- Toolpath geometry
- Tool offset data
- Work offset data
- Coordinate system definition

Tool data

- Tool type
- Tool number
- Tool diameter
- Tool length
- Tool radius
- Tool clearance

Motion data

- Spindle speed
- Feed rate
- Tool engagement
- Cutting parameters
- Coolant parameters

Verify the data

- Once the data has been loaded, you should verify it to make sure that it is correct.
- This can be done by using the CNC machine's control panel or by using a computer program.
- If you find any errors in the data, you will need to correct them“before you can start the CNC machine.

Start the CNC machine.

- Once the data has been verified, you can start the CNC machine
- The machine will then begin to follow the instructions in the data and perform the desired operation.

Here are the data that are typically inputted to run a CNC machine.

- **Geometric data:** This data defines the shape of the part that you want to machine. It includes the coordinates of the points that make up the part, as well as the dimensions of the part.
- **Tool data-** This data defines the tools that you will be using to machine the part It includes the type of tool, the size of the tool, and the cutting“parameters for the tool.
- **Motion data:** This data defines the movements that the CNC machine will make to machine the part. It includes the feed rate, the spindle speed, and the sequence of movements that the machine will make.

In addition to these basic data, there may be other data that is specific to the “CNC machine or the part that you are machining. For example, you may need to input data about the machine's tool offsets or the part's workpiece coordinate “system.

Data output

The steps on how to output data from a CNC machine:

1 **Identify the type of data you want to output.** There are many different types of data that can be output from a CNC machine, including:
   - **G-code:** This is the code that tells the machine how to move and what to do.
   - **Toolpath data:** This data shows the path that the tool will take when machining the part.
   - **Machine status data:** This data shows the current status of the machine, such as its temperature, speed, and feed rate.
   - **Sensor data:** This data can be collected from sensors that are attached to the machine, such as vibration sensors or temperature sensors.

2 **Choose a method for outputting the data.** There are several different methods for outputting data from a CNC machine, including:
   - **USB:** This is a common method for outputting data from a CNC machine.
   - **Ethernet:** This method can be used to output data to a network or to a remote computer.
   - **Serial port:** This method can be used to output data to a serial device, such as a printer or a terminal.

3 **Connect the CNC machine to the output device.** Once you have chosen a method for outputting the data, you will need to connect the CNC machine to the output device. This can be done using a USB cable, an Ethernet cable, or a serial cable.

4 **Start the data output process.** Once the CNC machine is connected to the output device, you can start the data output process. This process will vary depending on the type of data you are outputting and the method you are using.

5 **Review the output data.** Once the data output process is complete, you should review the output data to make sure that it is correct.

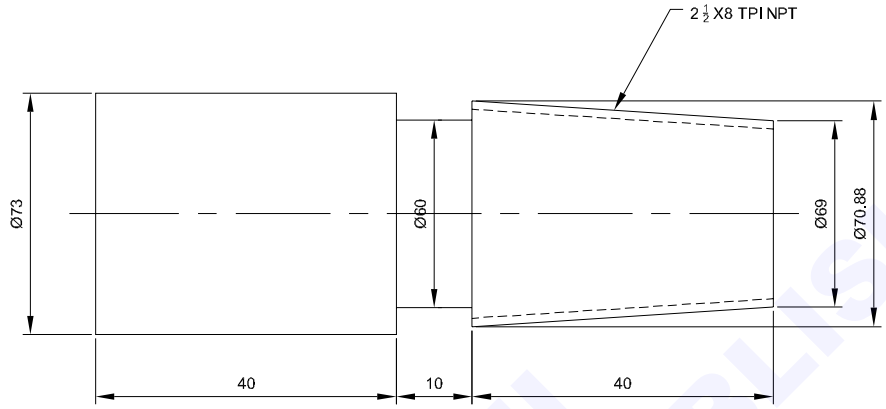
Here are some additional tips for outputting data from a CNC machine:

- Make sure that the CNC machine is properly calibrated before you start the data output process.
- Use a high-quality cable to connect the CNC machine to the output device.
- Follow the instructions that came with your CNC machine or output device

Thread on taper surface (V-form)

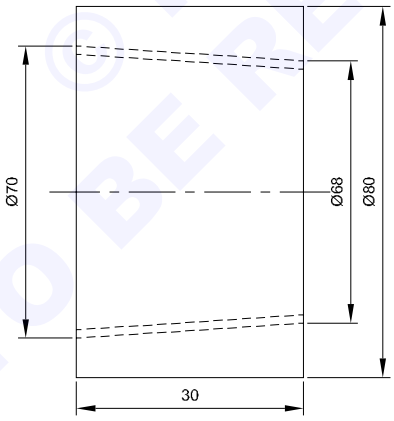
Objectives: At the end of this exercise you shall be able to
- **taper turning using taper turning attachment**
- **external taper thread by taper turning attachment**
- **internal taper thread by taper turning attachement.**

TASK: 1



TAPER THEADING (EXTERNAL)

TASK: 2



TAPER THEADING (INTERNAL)

ANGLE OF TAPER	1°47'
ANGLE OF THREAD	60°

NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.
2	Ø62x55	-	MS ROD	-	-	2.7.149
1	Ø34x155	-	MS ROD	-	-	2.7.149

SCALE 1:1	THREAD ON TAPER SURFACE (V FORM)	DEVIATIONS ± 0.02	TIME :
		CODE NO.TU20N27149E1	

Job Sequence

TASK 1 : **External thread**

- Check the raw material for its size.
- Hold the job in the 4 jaw chuck and true it.
- Finish facing on the end.
- Reverse the job hold in the chuck and true it.
- Face and maintain to the required total length of 90 mm.
- Loosen and remove the job and dismount from four jaw chuck.
- Hold the job in 4 jaw independent chuck.
- Turn the dia 73 mm to a length of 40 mm.
- Reverse the job and fix again in 4 jaw chuck to turn ∅70.8x50 mm length.
- Set the under cut tool.
- Turn undercut dia 60 x 10 mm length as per drawing.
- Make a taper on taper turning attachment method set the angle 1°47'.
- Set the change gear and norton gear box lever position for thread 2 ½ x 8 T.P.I.
- Cut the taper thread.
- Remove the burrs.

— — — — — — — —

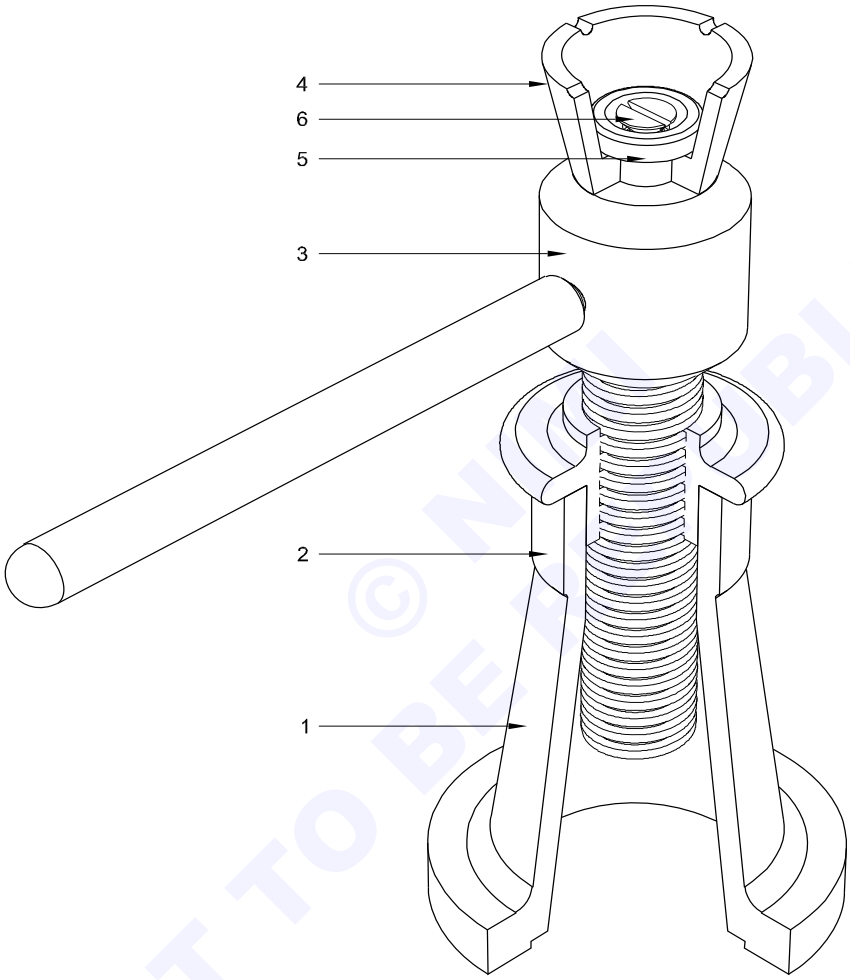
TASK 2 : **Internal thread**

- Check the raw material for its size.
- Hold the job in a four jaw chuck and true it.
- Set the tool for facing.
- Face one end and turn ∅80 mm for maximum length.
- Reverse the job, hold in the chuck and true it.
- Face and turn to maintain total length 30 mm and ∅80 mm.
- Drill dia 10 mm through hole.
- Enlarge the through hole to ∅68 mm.
- Set the natural tangent value 1º 47'
- Set the taper turning attachment for the 1º47' amount of taper.
- Taper bore by engaging the power feed for a total length.
- Complete the taper boring.

— — — — — — — —

Manufacturing & assembly of screw jack by performing different lathe operations

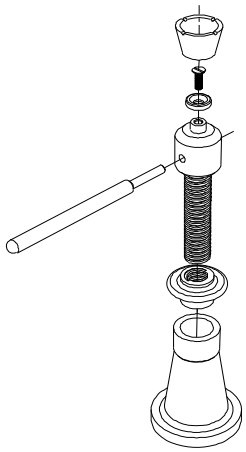
Objectives: At the end of this exercise you shall be able to
- **read the drawing**
- **arrange the tools and work**
- **produce each parts and assemble.**



NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.
1	Ø25 X 280	-	M.S	-	7	2.7.150
1	Ø29 X 40	-	M.S	-	6	2.7.150
1	Ø40 X 13	-	M.S	-	5	2.7.150
1	Ø70 X 47	-	C.S	-	4	2.7.150
1	Ø70 X 245	-	M.S	-	3	2.7.150
1	Ø90 X 50	-	G.M	-	2	2.7.150
1	Ø145 X 190	-	C.I	-	1	2.7.150

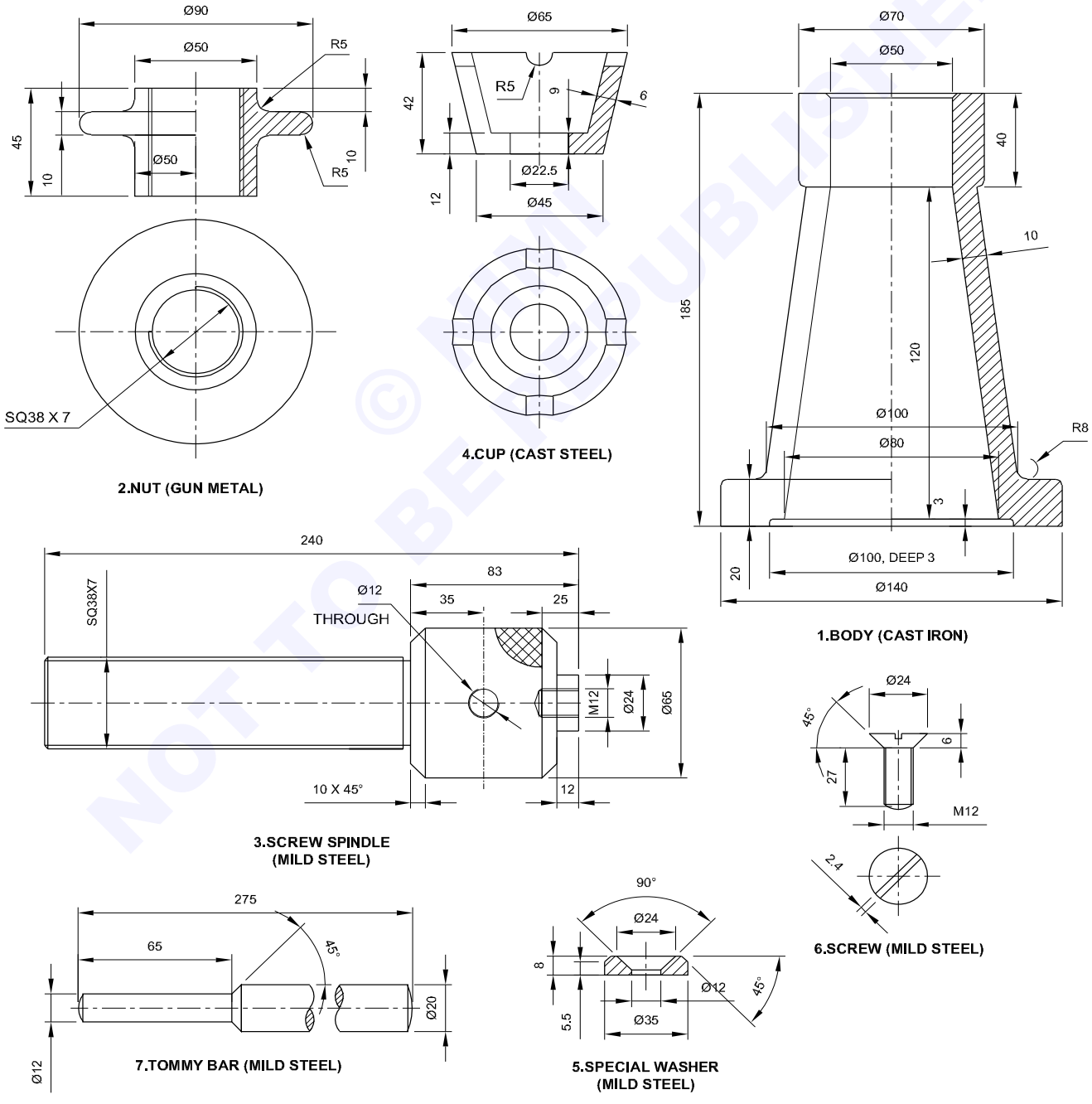
SCALE 1:1	SCREW JACK	TOLERANCE ±0.02	TIME :
		CODE NO. TU20N27150E1	

Fig 1



SCREW JACK

Fig 2



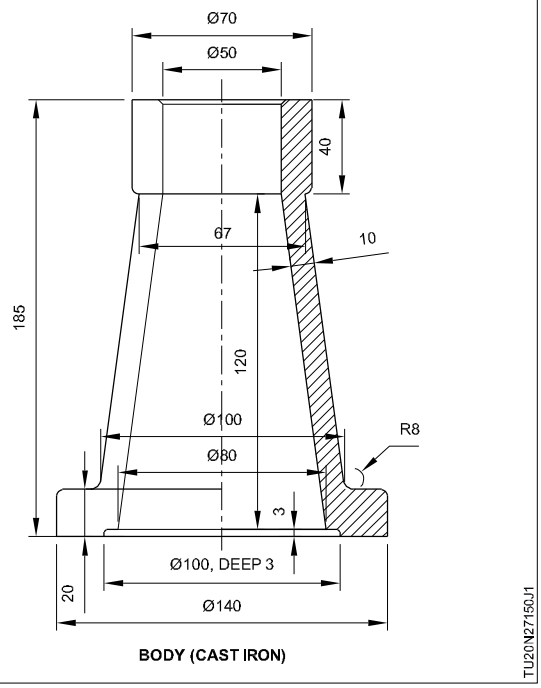
DETAILS OF SCREW JACK

TU20N27150H2

Job Sequence

1 Body (Fig 1)

Fig 1



BODY (CAST IRON)

- Hold the job in 4 jaw chuck and true it.
- Face the job, turn ∅140 in maximum length
- Face ∅100 to a depth 3mm s per drawing.
- Bore ∅50 through out the length after using pilot drill, ∅10, ∅18, ∅24 & ∅34mm drills respectively.
- Set the compound rest at angle 7' in cw direction.

Calculation

Major dia, D = 80mm

Minor dia, d = 50mm

Length ℓ = 120mm

$$\tan\theta = \frac{80-50}{2\times120}$$

$$\theta = \tan^{-1}\left[\frac{80-50}{2\times120}\right] = 7.1°$$

θ = half angle of taper

- Reverse the job, hold it with in 15mm.
- Face and correct the length to 185mm.
- Step turn 70mm to 40mm length.
- Taper turn the body portion with major dia, D = 100, minor dia, d = 67, and length l = 125 by keeping radius R8 fillet.

Calculation

$$\tan\theta = \frac{D-d}{2\times\ell}$$

$$\frac{100-67}{2\times125}$$

$$\theta = \tan^{-1}\left[\frac{100-67}{2\times125}\right]$$

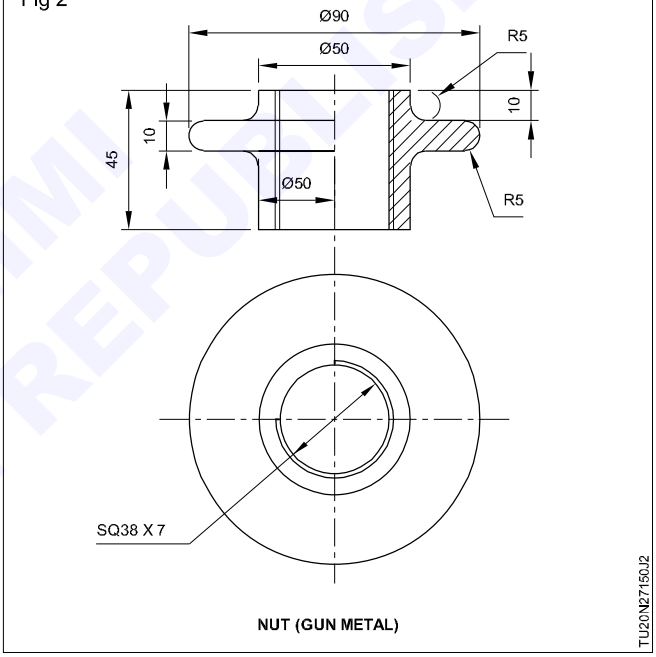
= 7.5

θ = half of taper

- Remove the job and deburr it.

2 Nut (Fig 2)

Fig 2

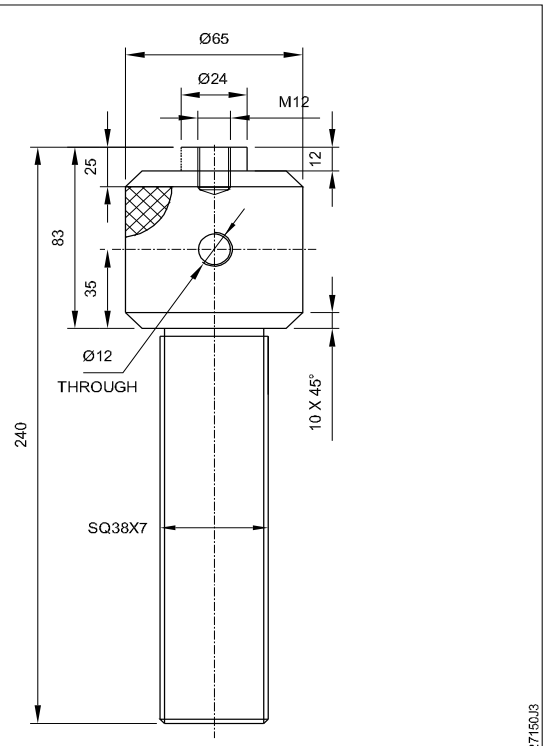


NUT (GUN METAL)

- Hold the job on 4 jaw chuck with maximum over hanging.
- Face and step turn ∅50 into 25mm length.
- Drill and bore a hold of ∅34.5 mm
- Cut internal square thread.
- Step turn OD ∅90x10
- Reverse the job
- Step turn ∅50 x 10mm keeping radius R5.
- Form radius R5 on dia ∅90 step.
- Remove the job and deburr it.

3 Screw spindle (Fig 3)

Fig 3



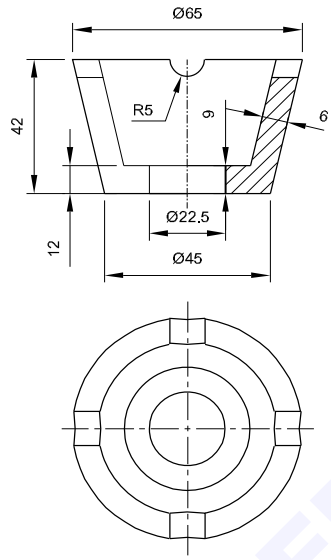
SCREW SPINDLE (MILD STEEL)

- Hold the job on 4 jaw chuck and face it.
- Step turn ∅22 and ∅65 upto 12mm and 71 mm length respectively.
- Drill ∅10.5 hole upto 25mm depth.
- Machine tap M12 x 1.5mm thread in above hole.
- Step turn ∅38 upto maximum possible length.
- Knurl the ∅65 portion with diamond knurling tool.
- Chamfer both end of ∅65mm portion in 10 x 45.
- Reverse the job.
- Face to correct length of 240mm
- Complete the step of ∅38 and chamfer the end.
- Cut the square thread SQ 38x7 to 157mm length.
- Drill a hole of ∅12mm in knurled portion by using drilling machine.
- Remove the job and deburr it.

4 Cup (Fig 4)

- Hold the blank of ∅68x70mm in a 4 jaw chuck and true it.
- Face & turn 65 upto 55mm length.
- Mark 10mm from face end.
- Remove the job and set on a drilling machine.
- Drill four ∅10 hole $\perp^r$ to each other on the marked portion.
- Reload the job on lathe face up to 10mm length (so that the holes become half)
- Drill through hole ∅22.5
- Taper turn OD with D = ∅65, d = ∅45 and length ℓ =42
- Taper turn ID upto 30mm keeping the same angle

Fig 4

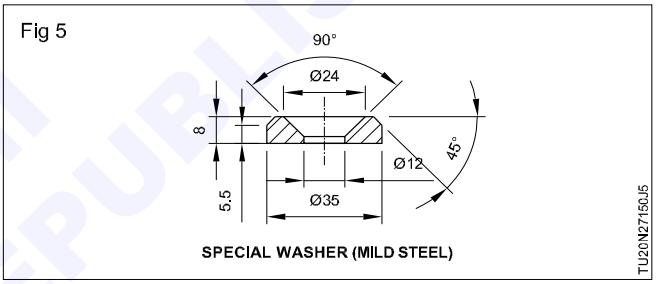


CUP (CAST STEEL)

- Part the job keeping 42mm length.

5 Washer (Fig 5)

Fig 5

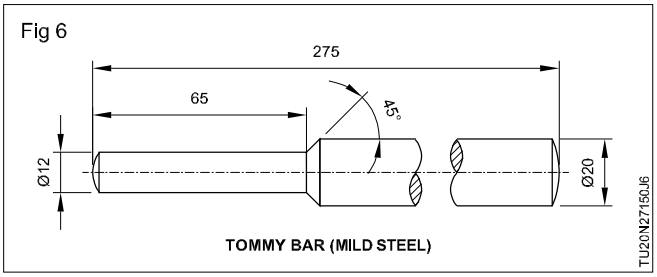


SPECIAL WASHER (MILD STEEL)

- Hold the blank of ∅40mm in 4 jaw chuck and true it.
- Face and drill a hold ∅12 upto 10mm.
- Correct the OD with ∅35 upto 10mm.
- Chamfer the end 2.5 x 45°
- Counter sunk the hole with a 90° counter sunking tool.
- Part the job keeping a length of 8mm.
- Deburr & inspect it.

6 Tommy bar (Fig 6)

Fig 6



TOMMY BAR (MILD STEEL)

- Hold the job on 4 jaw chuck & true it.
- Face & turn ∅12 upto 65mm length.
- Turn ∅20 upto 27.8mm length and chamfer it.
- Reverse the job correct the length of 275mm with a slight curve form at the end.
- Knurl upto 50mm length from the end.
- Remove the job, deburr & inspect.

Prepare differnet type of documentations and methods of recording information

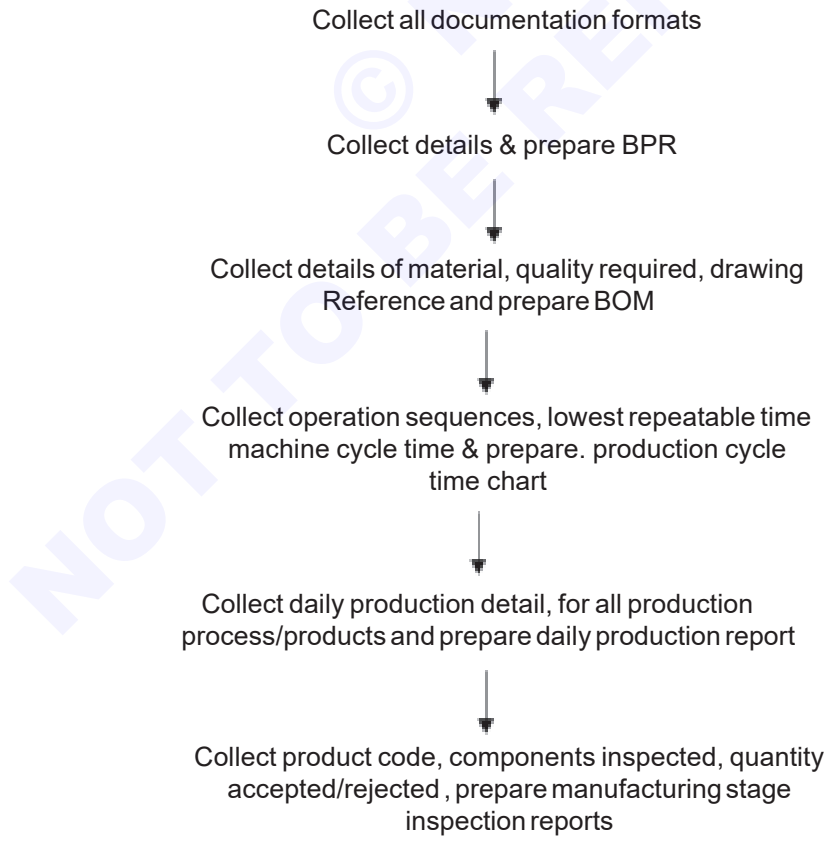
Objectives : At the end of this exercise you shall be able to
- **prepare and fill up batch processing record format**
- **prepare and fill up bill of materials (BOM)**
- **prepare and fill up production cycle time format**
- **prepare and fill up daily production report format**
- **prepare and fill up manufacturing stage inspection report format.**

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 instructor/ Training Officer.

Note:
- **Instructor / Training Officer should arrange for a industrial visit near by your institute, collect inputs and fill up format as required.**
- **Trainees will be guided by the concerned instructor/ TO.**
- **Collect necessary information forms and instruct the trainees to reproduce the forms and guide them in Documentation**

Flow chart



BATCH PROCESSING RECORD - FORMAT - 1

Batch Processing Record	
Description of job	Batch no. :
Part no. : Name of part :	Batch quantity : Batch record no. : Purchase order no. :
Description of process :	
Manufacturing Organisation :	
Period of manufacture (Year - Qtr): Start date of manufacture: End date of manufacture:	
Number of pages according to batch: Inserted pages: Manufacturing facilities:	
Total number of pages	
1. Operator / Technician — Date — Name and signature	
2. Production in-charge: — Date — Name and signature	
3. Section manager — Date — Name and signature	
4. Plant in-charge: — Date — Name and signature	
5. Production manager — Date — Name and signature	
Remarks (if any)	

BILL OF MATERIAL (BOM) - FORMAT-2

(as per IS: 11666 - 1985)

S. No.	Item No.	Description	Quantity	Reference dwg no.	Material as per standard	Remarks

Date:

Place: Incharge

PRODUCTION CYCLE TIME - FORMAT-3

Organisation Name: Department / Section:	Process:	Line Incharge:	Date/Time:

Operator:											
Operations Sequence	Observed Time								Lowest Repeatable	Machining Cycle Time	Notes

DAILY PRODUCTION REPORT - FORMAT-4

	Daily Production Report	
Date:	**Department:** **Section:**	**Organisation Name:**

	Process - I		Process-II		Process-III		Process-IV		Quality Control		Packing	
	Planned	Completed	Planned	Completed	Planned	Completed	Planned	Completed	Planned	Completed	Planned	Completed
Job Order No. Quantity Material & Size												
Job Order No. Quantity Material & Size												
Job Order No. Quantity Material & Size												
Job Order No. Quantity Material & Size												
Job Order No. Quantity Material & Size												

Signature of section Incharge

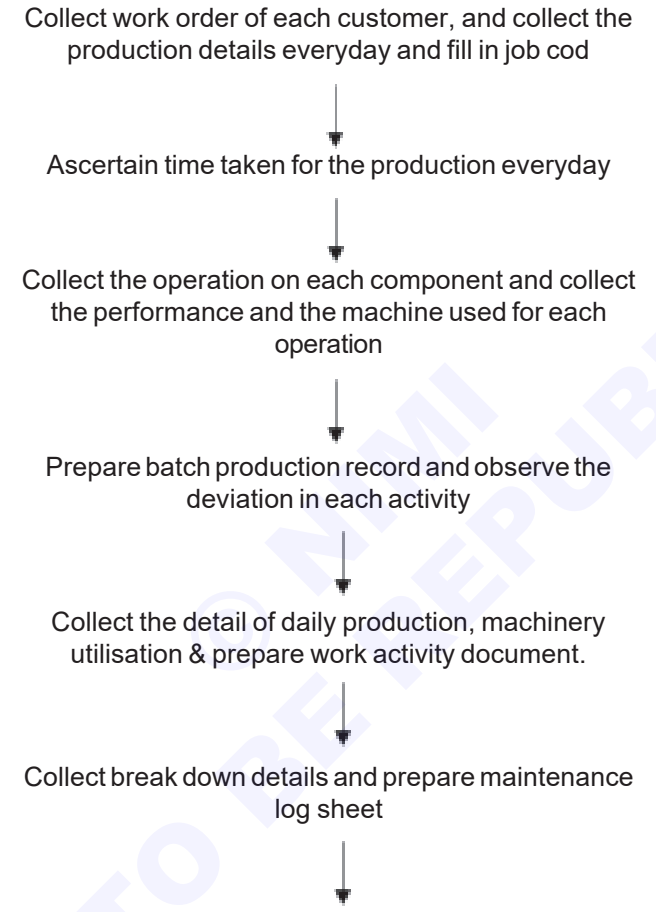
MANUFACTURING STAGE INSPECTION REPORT - FORMAT-5

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

Documentations - 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.**

Flow diagram



Job Sequence

- Study the different types of documentation provided (format).
- Visit industry and collect the input / information from industry and fill it up in all the format.
- Prepare the required in format with the knowledge gained during the industrial visit.
- Record relevant information in the format.
- Get it checked by your instructor/ Training Officer.

Note:

- **Instructor / Training Officer should arrange for a industrial visit near by your institute, collect inputs and fill up format as required.**
- **Trainees will be guided by the concerned instructor/ TO.**
- **Collect necessary information forms and instruct the trainees to reproduce the forms and guide them to fill it up.**

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

Batch Production Record in accordance with batch processing record

Manufacturing Organisation Name: _____

Description of job: _____

Name of part: _____

Batch No.: _____

The following deviations have appeared (continued)

No. process step	Name of processing step	Document page no.	Short description of deviation
1	Raw material preparation: Operation 1: _____ Operation 2: _____ Operation 3: _____		1. _____ 2. _____ 3. _____ 4. _____
2	Sizing of material: Operation 1: _____ Operation 2: _____		1. _____ 2. _____ 3. _____

ESTIMATION SHEET - FORMAT-4

Part Name: _____	Part No.: _____	Part Drawing
Assembly: _____	Material: _____	
Assembly No.: _____	Stock size: _____	

Operation No.	Operation description	Machine	Estimated time	Rate / piece per hr.	Tools

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

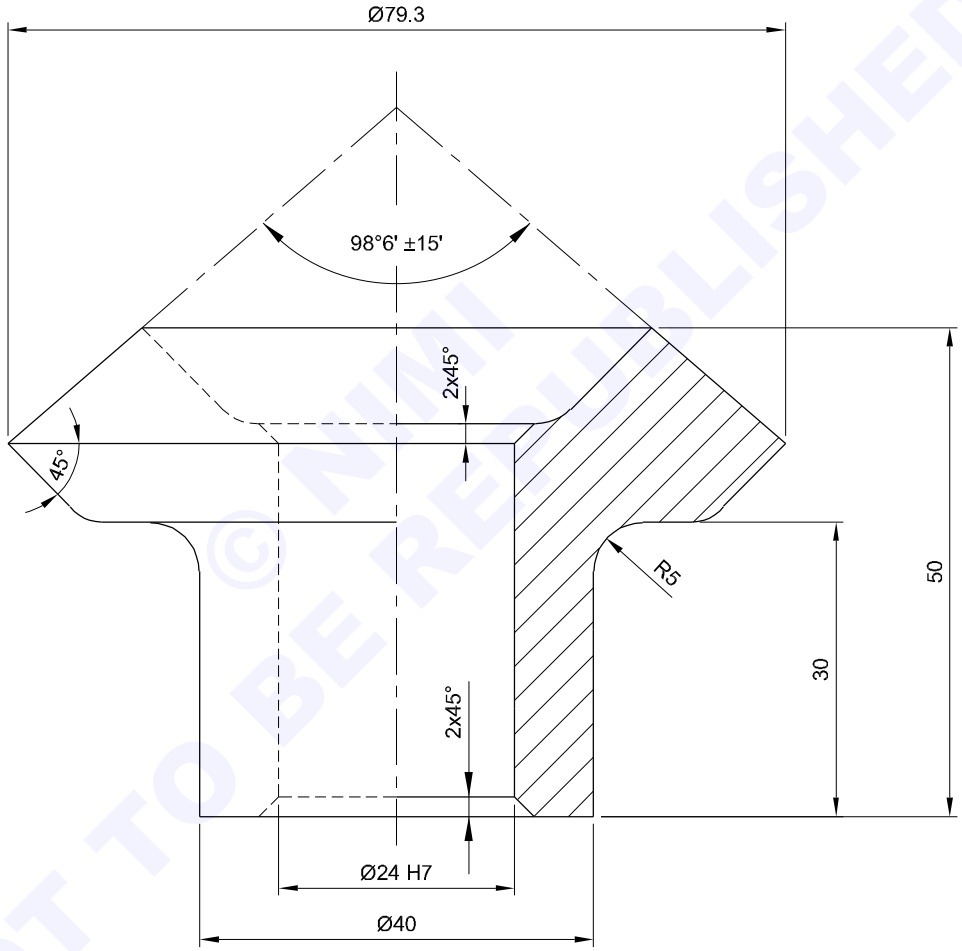
Organisation Name :				
Department :				
Section :				
History sheet of machinery & Equipment				
Description of equipment				
Manufacturer's address				
Supplier's address				
Order No. and date				
Date on which received				
Date on which installed and placed				
Date of commissioning				
Size: Length 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

Turn bevel gear blank

Objectives: At the end of this exercise you shall be able to
- **turn a bevel gear blank**
- **maintaining the shape and dimensions.**



ALL DIMENSIONS ARE IN mm

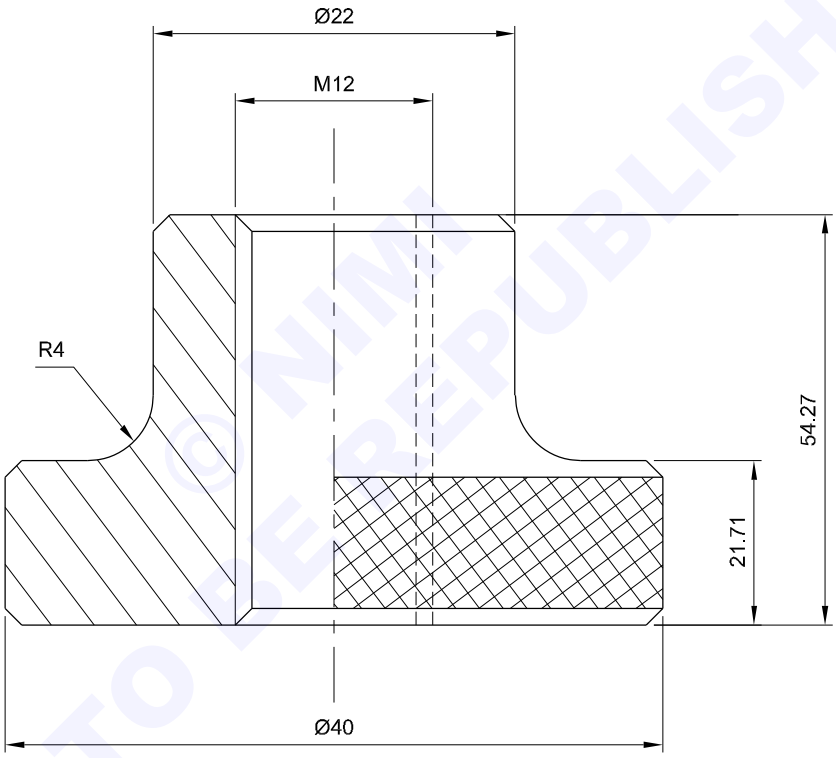
1	Ø 85 - 55	-	Fe310	-	-	2.7.152
NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.
SCALE 1:1	TURN BEVEL GEAR BLANK				DEVIATIONS ± 0.02	TIME :
					CODE NO.TU20N27152E1	

Job Sequence

- Check the raw material size.
- Hold the job in 4 jaw chuck and true it.
- Face the one end.
- Turn ∅42 mm 25 mm length.
- Form radius with 5mm filleted tool at the edges 30 mm length by turning the step 40 mm.
- Centre drill the job.
- Drill a pilot hole with a ∅12 mm drill bit.
- Enlarge the hole with a ∅20 mm drill.
- Bore the drilled hole to ∅23.75 mm.
- Chemfer the edge of the hole 2 x 45º.
- Ream the bored hole with a machine reamer ∅24 H7.
- Reverse the job and reset it by holding on ∅40 mm.
- Face the other end to maintain 50 mm lengh.
- Set compound slide 50° and boring of taper with form of 5 mm radius end of reamed hole.
- Chamfer the reamed hole 2 x 45°.
- Mount the job on the mandrel along with bush and nut.
- Hold the mandrel between the centres.
- Turn ∅79.3 mm.
- Mark the distance with a V tool to 12 mm from the end of face.
- Set compound slide 49º to vertical axis and form the bevel edge.
- Lock the carriage while bevelling the edge.
- Set compound slide 45º to vertical axis and form of the taper.
- Remove the burrs.

Reading of part drawing & make process plan for turning operation and make arbor with clamping heagonal nut

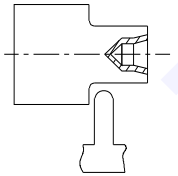
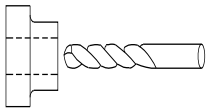
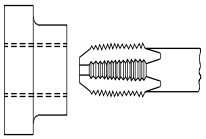
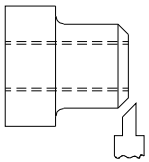
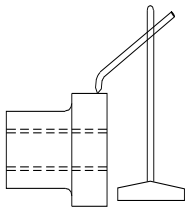
Objectives: At the end of this exercise you shall be able to
- **read a part drawing**
- **make a process plan.**

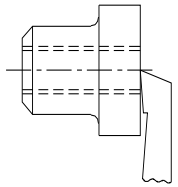
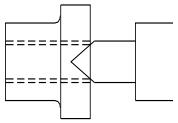
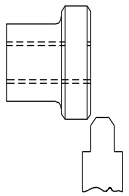


ALL DIMENSIONS ARE IN mm

1	Ø 45 x 60	-	-	-	-	2.8.153
NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.

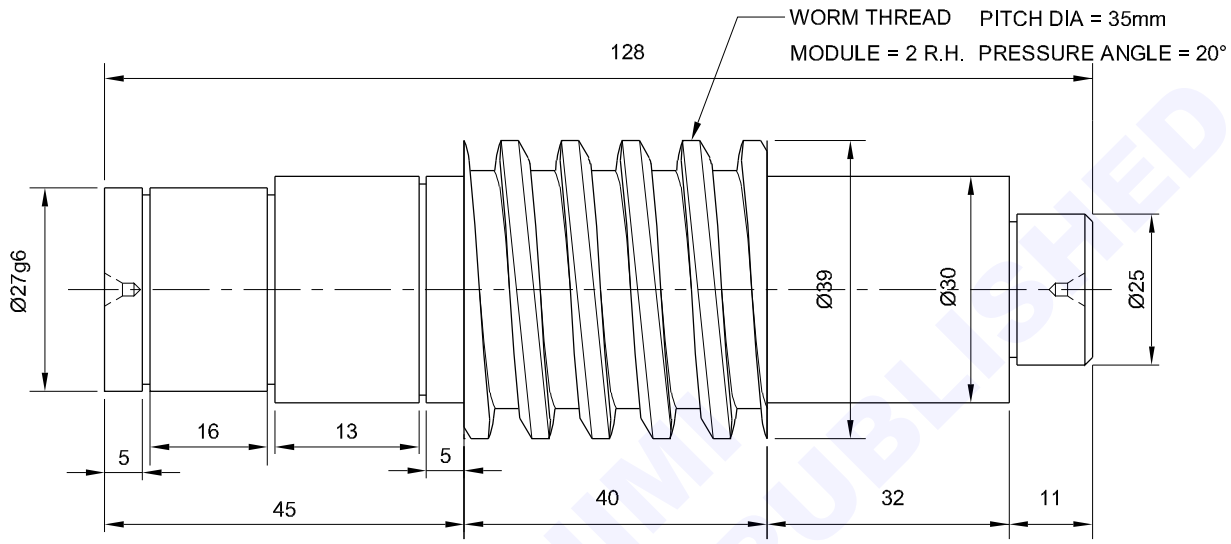
SCALE 1:1	**READING OF PART DRAWING AND MAKE PROCESS PLAN AS ABOVE**	TOLERANCE ±0.01	TIME:
		CODE NO.TU20N28153E1	

Part No.	Description of part	Working steps (Job sequence)	List of tools, equpts. Measuring & Marking tools	Machining parameters				Formula used	Estimated time	Time taken
				Cutting	R.P.M.	Feed	Coolant			
		Form a radius of 4 mm with a radius form tool and maintain step length 15 mm	4 mm radius tool 150 mm vernier caliper, radius gauge	30	300	hand feed	Soluble oil	-	10 min	
		Pilot drill ∅5 mm and drill ∅10.25 mm through and chamfer	∅5 mm drill ∅10.25 mm drill, HSS chamfering tool	35	700	hand feed	Soluble oil	-	10 min	
		Cut internal thread M12 by using a M12 machine tap	M12 tap, tap holder, thread plug gauge	20	300		mobile oil	-	20 min	
		Form chamfer 1 x 45° on the edge of step ∅22 mm	HSS chamfering tool	30	500	hand feed	soluble oil		10 min	
		Reverse and hold the work on ∅22 mm and true the dia and face	chuck key, surface gauge						10 min	

Part No.	Description of part	Working steps (Job sequence)	List of tools, equpts. Measuring & Marking tools	Machining parameters				Formula used	Estimated time	Time taken
				Cutting	R.P.M.	Feed	Coolant			
		Remove material by facing and maintain total length 25 mm	Facing tool, 150 mm vernier caliper	30	500	0.1 mm/ rev	soluble oil		10 min	
		Turn 39.7 mm remaining length & support with the revolving centre.	HSS RH turning tool, 150 mm vernier caliper, revolving centre.	35	500	0.2 mm/ rev	mobile oil		20 min	
		Diamond Knurl on ϕ39.7	HSS Universal Knurling tool		140	Hand feed	Mobile oil		10 min	
		Chamfer the knurled surface 1 x 45º	HSS double side chamfering tool	35	400	hand feed	soluble oil		15 min	

Practice of special operations on lathes worm gear cutting (shaft)

Objectives: At the end of this exercise you shall be able to
- **grind under cut tool**
- **resharpen worm thread cutting tool**
- **cut worm thread an required.**



Job Sequence

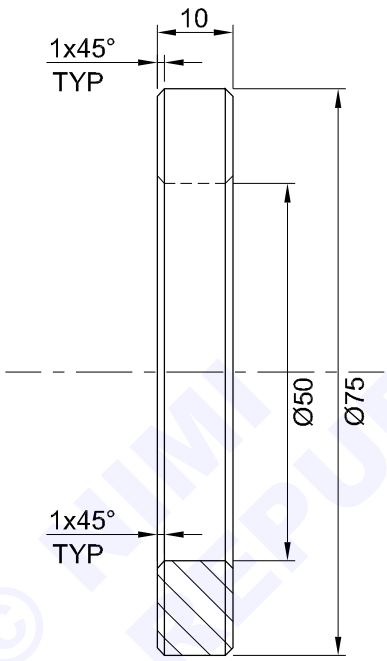
- Check the raw material size.
- Hold the job in the 4 jaw chuck and true it.
- Face the one end.
- Centre drill the work.
- Turn ∅39.5 for the maximum length.
- Turn the step to dia 30mm x 43mm length.
- Turn the step to dia 25mm x 11 mm length.
- Undercut and maintain ∅21 and grooving width of 2 mm with grooving tool.
- Reverse and hold the job in the chuck and true it.
- Face the other end to maintain 128 mm length.
- Centre drill the work.
- Turn ∅30 x 45 mm length.
- Turn step ∅27 x 25 mm length.
- Chamfer the end 2 x 45º.
- Undercut from face end after 5 mm maintain ∅23 x2 width.
- Undercut from face end after 23 mm to maintain groove ∅23 x 2 mm width.
- Undercut from face end after 38 mm.
- Undercut from face end after 35 mm to maintain groove ∅26 mm x 2 mm width.
- Loosen and remove the job from 4 jaw chuck.
- Set the live centre and dead centre.
- Hold the job between centres.
- Finish turn ∅39 mm x 40 mm length.
- Set the included angle 29° thread tool.
- Cut the worm thread to a depth 4.5 mm.
- Remove burrs.

1	Ø 42 x 130	-	Fe310	-	-	2.8.154
NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.

SCALE 1:1	PRACTICE ON SPECIAL OPERATION ON LATHE / WORM GEAR CUTTING	DEVIATIONS ± 0.02	TIME :
		CODE NO.TU20N28154E1	

Boring on lathe with soft jaws to make bush with collar

Objectives: At the end of this exercise you shall be able to
- **set job in soft jaw**
- **drill through hole**
- **make chamfer on both sizes**
- **bore the workpiece to the required size.**



Job Sequence

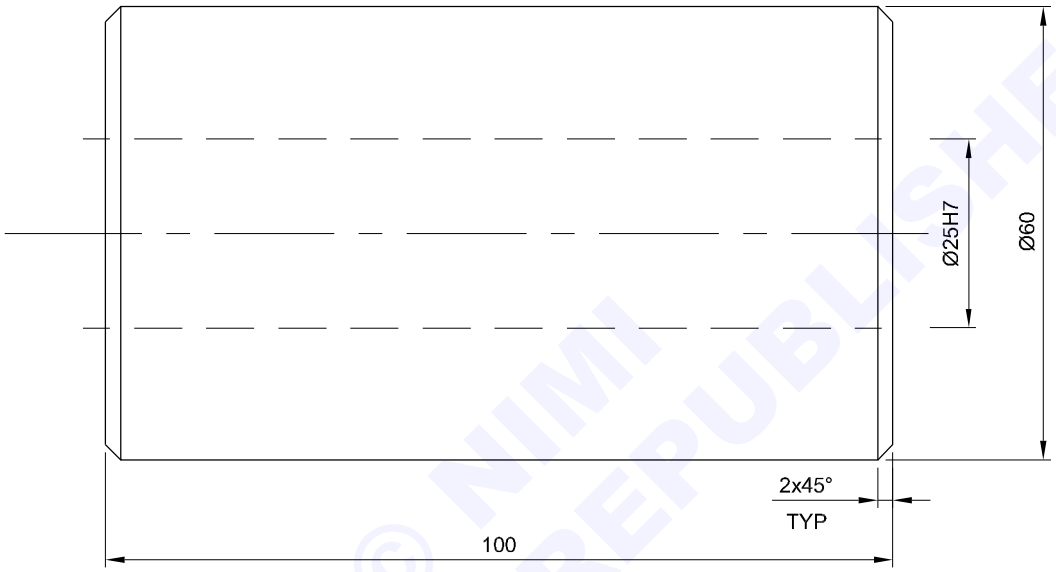
- Check the raw material for its size.
- Hold job in 4 jaw chuck about 15 mm outside and true it.
- Finish face end and turn ∅75 mm x 10 mm length.
- Centre drill the work.
- Drill ∅10 mm, ∅16 mm, ∅25 mm through hole.
- Bore through hole to ∅48 mm (premachining).
- Chamfer one end 1 x 45°.
- Reverse the job and true it.
- Face the other end maintain length 10 mm
- Chamfer outer diameter by 1x45°.
- Loosen and remove the job from chuck.
- Hold the job in soft jaws.
- Set the boring tool correct centre height
- Bore the job to ∅50 mm.
- Check the bore size useing bore dial gauge.

1	Ø 80 - 15	-	Fe310	-	-	2.8.155
NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.

SCALE 1:1	BORING ON LATHE WITH SOFT JAWS TO MAKE BUSH WITH COLLAR	DEVIATIONS ± 0.02	TIME :
		CODE NO.TU20N28155E1	

Make a Arbor Support bush (Proof machining)

Objectives: At the end of this exercise you shall be able to
- **finish the bore within close limits**
- **select the internal grooving tool**
- **chamfer the edge and make an arbor bush.**



Job Sequence

- Hold the job in four jaw chuck and true it.
- Face the one end.
- Turn dia 60 mm for the maximum possible length.
- Chamfer the end 2x45°
- Reverse the job and reset it.
- Face the other end and maintain to 100mm length.
- Turn to ∅60.00mm and complete the plain turning operation.
- Chamfer the end 2x45°
- Centre drill the job
- Drill the job with a pilot drill ∅12mm
- Drill the job with a ∅20mm drill bit.
- Bore the drilled hole to ∅24.75 mm and chamfer 1x45°.
- Ream the job with a ∅25 H7 machine reamer.
- Check the dimensions after deburing.

1	Ø 65 - 105	-	Fe310	-	-	2.8.156
NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.

SCALE 1:1	MAKE ARBOR SUPPORT BUSH (PROOF MACHINING)	DEVIATIONS ± 0.02	TIME :
		CODE NO.TU20N28156E1	