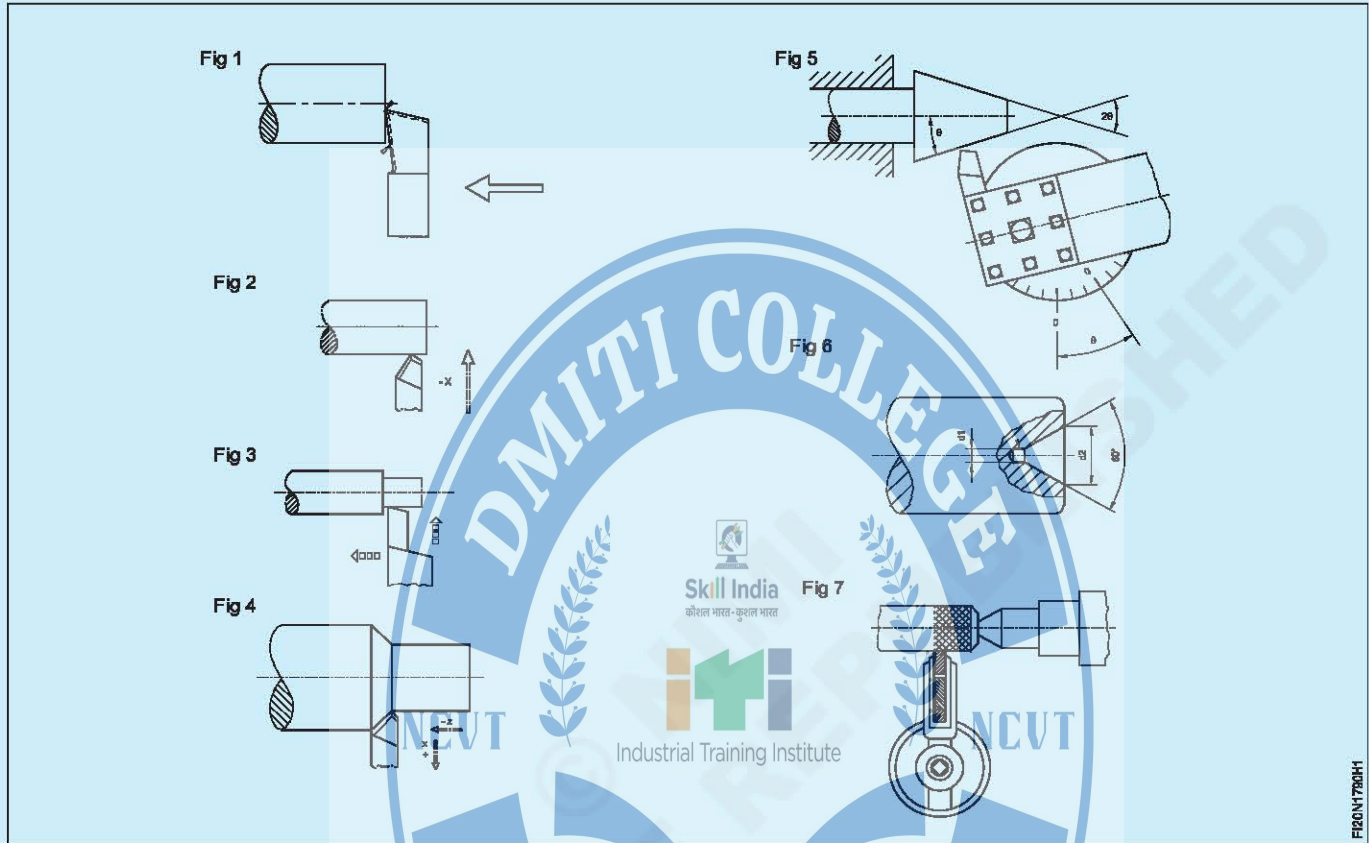


Lathe operations

Objective: At the end of this exercise you shall be able to
 • record the different lathe operations in table 1.



Note: Instructor shall demonstrate to the trainees regarding the different lathe operation performed in lathe.

Record the lathe operations in Table 1

TABLE 1

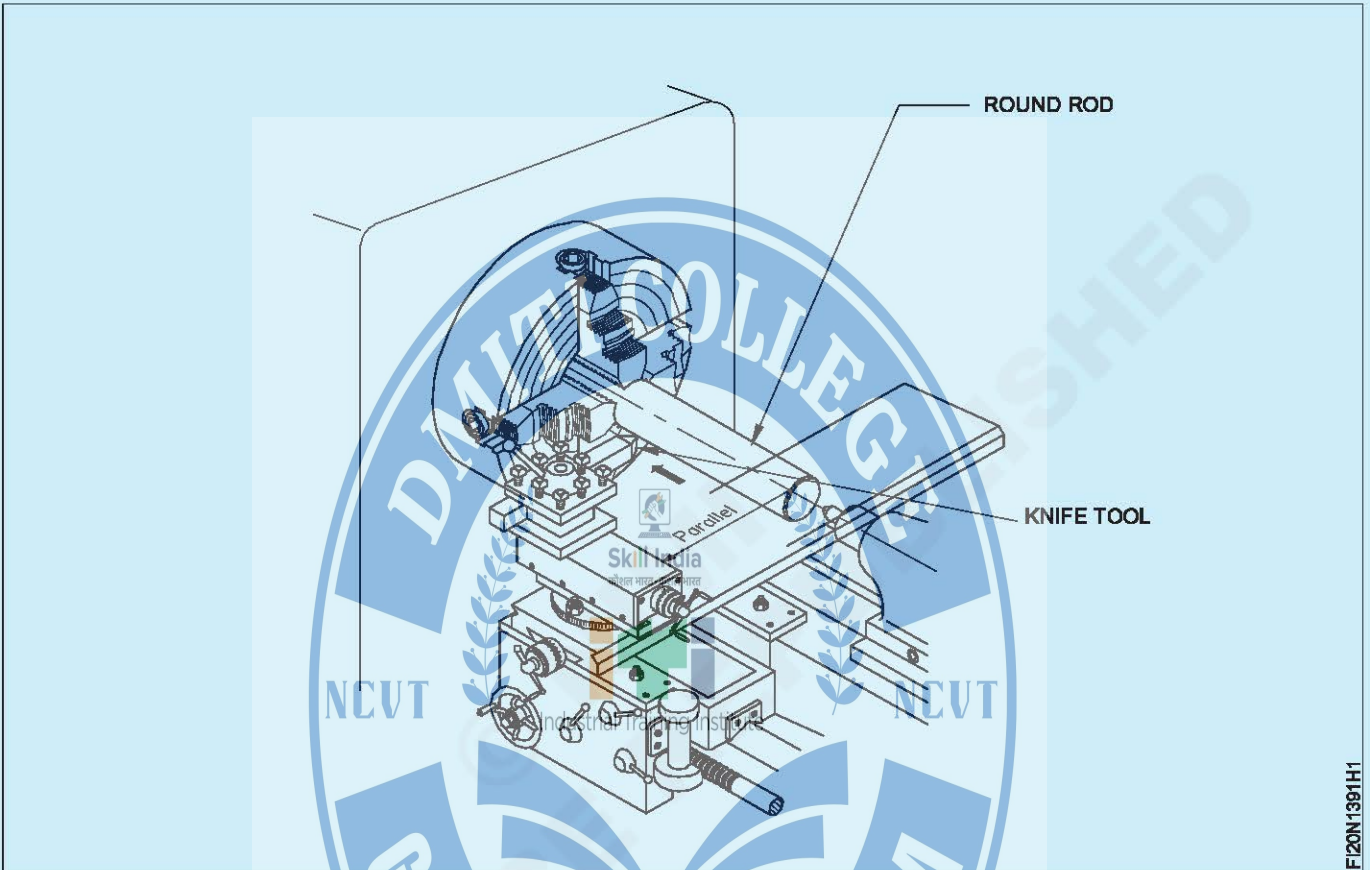
Fig.No.	Name of the operation
1	
2	
3	
4	
5	
6	
7	

Get it verified by your instructor.

True job on four jaw chuck using knife tool

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

- set the round rod/job in four jaw chuck
- true the round rod/job using knife tool.



Job Sequence

- Open one jaw by chuck key.
- Turn the chuck and open the opposite jaw
- Open all four jaws approximately more than job diameter
- Keep the job inside the jaws
- Make jaws close and hold the job
- Check the job trueness by knife tool.
- Tighten all the jaws.
- Check the trueness of the job by rotating the chuck in neutral position using knife tool.
- Knife tool should touch the job evenly.
- Check once again for the true running of the job.

Skill Sequence

Truing work in a four jaw chuck with the help of a surface gauge

Objective: This shall help you to

- true a round rod in a four jaw independent chuck with the help of a surface gauge.

If truing is not done before turning, the following will be the results.

Uneven load on the cutting tool.

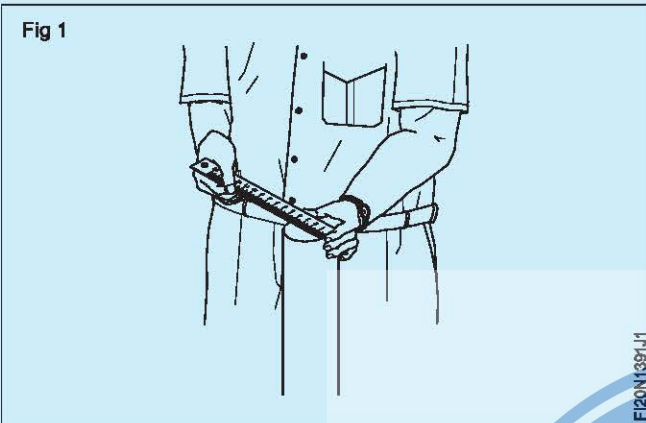
For the same depth more metal will be removed from the out of centre portion.

Surface turned may not be cylindrical.

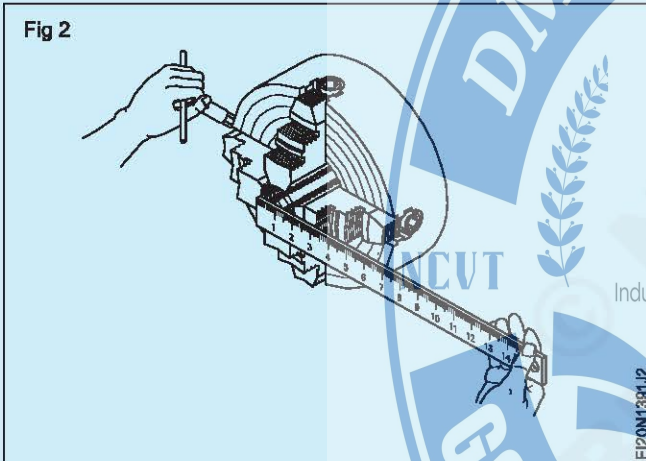
During truing

Keep the main spindle in a neutral position.

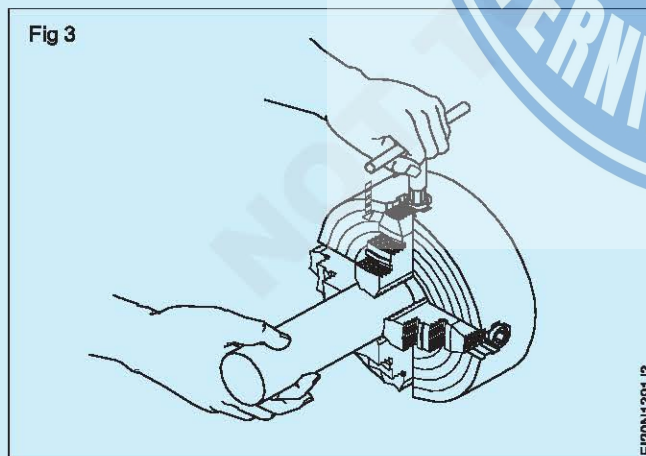
Measure the job diameter with an outside caliper or with a steel rule. (Fig 1)



Position the four jaws of the independent chuck, equidistant from the centre. The distance between the inner face of the opposite jaws is equal to the diameter of the work. (Fig 2)



Open the adjacent jaws sufficiently enough to insert the work. (Fig 3)



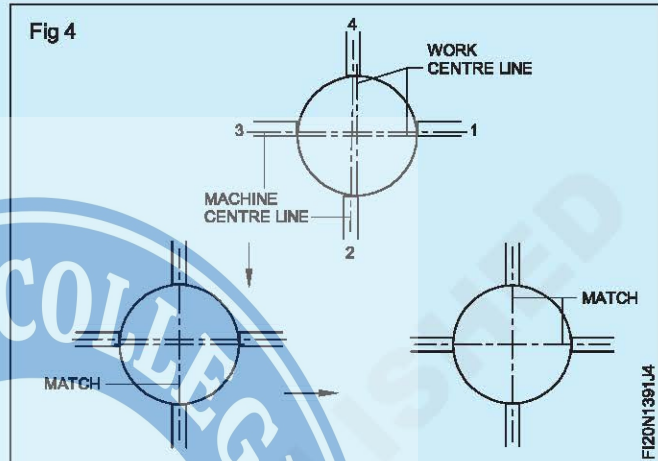
Place the work inside the chuck, keeping sufficient portion outside the chuck for turning, and tighten the two adjacent jaws, enough to grip the work.

Fix the knife tool on the bed-ways close to the chuck.

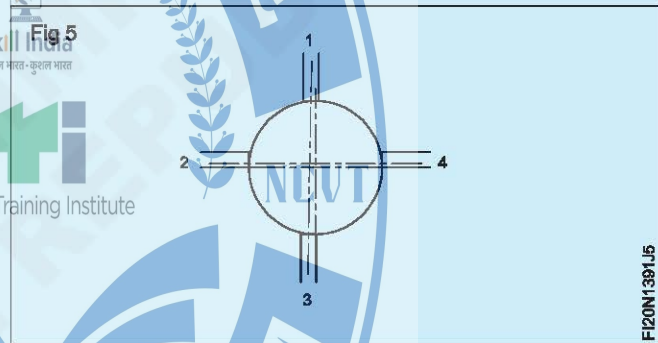
Adjust the tool to make its tip move close to the top or side portion of the work with a minimum gap.

Rotate the chuck by hand and observe the gap between the tool and work surface for the position of the two opposite jaws.

Open the jaw slightly where the gap is more, and tighten the opposite jaw. (Fig 4).



Repeat until the gap is the same. (Fig 5)



Repeat the above sequences for the other set of opposite jaws.

Bring the tool point tip closer to the work surface.

Rotate the chuck by hand and observe the gap.

Engage the spindle levers at about 250 rpm and run the machine.

Touch the tool point on the job.

If the line on the job is uniform tighten the jaw.

Repeat till a uniform line is formed.

Finally, tighten the opposite jaws with the same amount of pressure.

Check once again for the true running of the work.

Face both the ends for holding between centres

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

- set the job on a four jaw chuck
- set the tool on the tool post
- face the job
- measure the length with vernier caliper.



Job Sequence

- Check the raw material for its size.
- Hold the job in a four-jaw independent chuck with 25 mm overhang and true it.
- Set the R.H. facing tool in the tool post.
- Set the R.P.M.
- Face one end of the work.
- Mark the job 250mm long and punch witness marks on the circumference.
- Reverse the job, clamp it in the chuck and true it again.
- Set the spindle speed closer to 318 revolution per minute.
- Face the length up to the half punch mark level and maintain 250 mm long.
- deburr and check the job.

1	Ø40 X 260	→ 1.7.93	Fe310	-	-	1.7.92
NO.OFF	STOCK SIZE	SEMI PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.
SCALE : NTS	FACE BOTH THE ENDS FOR HOLDING BETWEEN CENTERS				DEVIATIONS : ± 0.04mm	TIME :
					CODE NO. F20N1792E1	

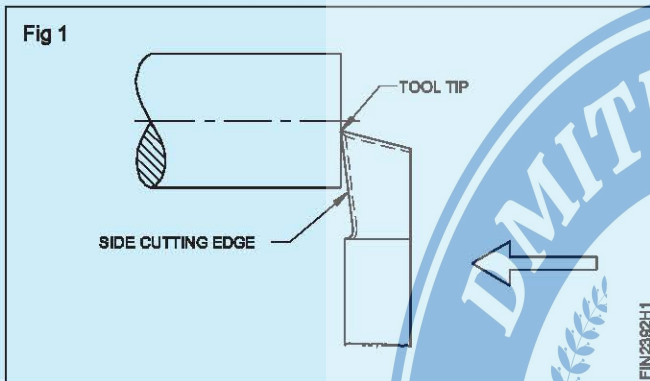
Skill Sequence

Finish-facing the work with a right hand facing tool

Objective: This shall help you to

- finish-facing the work using a right hand facing tool.

When more metal is to be removed on the face of work, we prefer to do rough facing by an L.H. facing tool or a L.H. roughing tool, feeding the tool from the periphery of the work towards the centre. Finish-facing is done to get a better surface finish on the face of the work by removing the rough facing. The normal R.H. facing tool, having its cutting edge straight, may be kept slightly inclined to the face of the work during facing. A tool, having its cutting edge itself ground at an angle, may be used. (Fig 1)



The procedure of the finish-facing the work with such a tool is given in sequence below.

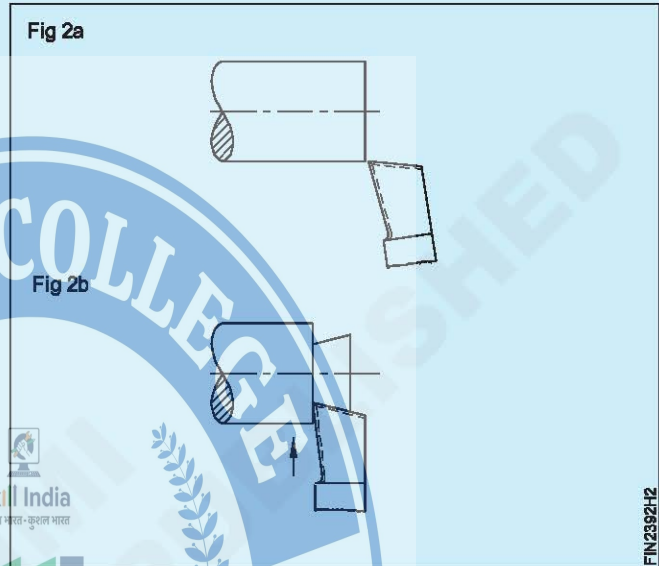
Hold the tool in the tool post to the correct centre height with its axis at right angles to the axis of the work and with a minimum overhang.

Set the machine to about 500 rpm. (Calculate the spindle speed by choosing the recommended cutting speed for finish-facing and the mean diameter of the work).

Start the machine and touch the tool point to the work-face by moving cross slide and carriage movement. Move the tool away from the work (Fig 2a) and set the top slide graduated collar to zero, eliminating backlash. Lock the carriage.

Feed the tool about 0.5 mm by the top slide.

Feed the tool towards the centre of the work by the cross-slide till the tool point crosses the centre. (Fig. 2b) Move back the tool to the starting position (Fig. 2a).



Advance the tool by a further 0.5 mm inside the work by the top slide.

Engage the power feed (set at 0.05 mm/rev.) and allow the tool to travel towards the centre of the work, removing the metal.

Repeat the sequence until the required amount of material is removed.

Observe the finish obtained.

Using roughing tool parallel turn ± 0.1 mm

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

- hold the work in lathe chuck
- grind RH turning tool
- set the turning tool in tool post
- set the machine spindle speed for turning
- parallel turn the work by hand feed method with various depths of cuts.



Job sequence

- Hold the work in four jaw chuck.
- Grind and fix RH turning tool and turn $\text{Ø}36$ to maximum length of job with the R.P.M. close to 318.
- Check the diameter by using vernier caliper
- Chamfer the end $3 \times 45^\circ$ and deburr.
- Reverse the job and hold in four jaw chuck.
- Turn the remaining length to $\text{Ø}36$ mm by parallel turning.
- Chamfer the end and deburr.

1	EX.NO.2.3.92	1.7.92	Fe310	—	—	1.7.93
NO.OFF	STOCK SIZE	SEMI PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.
SCALE :NTS	USING ROUGHING TOOL PARALLEL TURN ± 0.1mm.				DEVIATIONS : ± 0.1 mm	TIME :
					CODE NO. F120N1793E1	

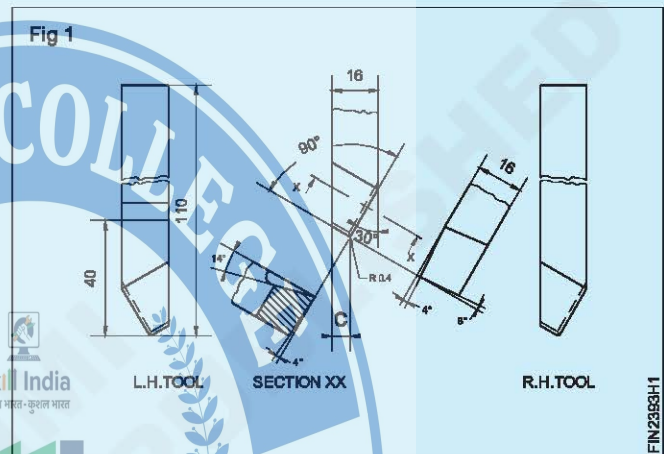
Skill Sequence

Rough turning tool grinding

Objective: This shall help you to

• **grind rough turning tool with various angle.**

- Rotate the wheel by hand and observe for free rotation.
- Check the grinding wheels for true running.
- Wear goggles.
- Dress the wheels by a wheel dresser.
- Adjust the tool-rest to maintain a minimum gap from the wheel face to a minimum of 2 to 3 mm.
- Hold and apply the side flank of the tool to the front face of the grinding wheel at 30° to horizontal.
- Move the tool left to right and vice versa to grind the side cutting edge angle to cover $2/3$ rd width of the tool.
- Grind a side clearance angle of 8° , the bottom of the edge touching the wheel first.
- Rough grind the end cutting edge angle of 30° and the front clearance angle of 5° simultaneously.
- Hold the top flank of the tool against the wheel face inclined at 14° , the rear the side contacting the wheel first, and grind the side rake angle of 14° .
- Ensure that the ground portion is parallel to the side cutting edge.
- Finish grind all the faces on the finishing wheel.
- Grind a nose radius of approximately R. 0.4 mm.
- Check the angles with a tool angle gauge and template.
- Lap the cutting edge with a oilstone.
- The top rake (back rake) angle should be kept at 0° .



NCVT

Industrial Training Institute

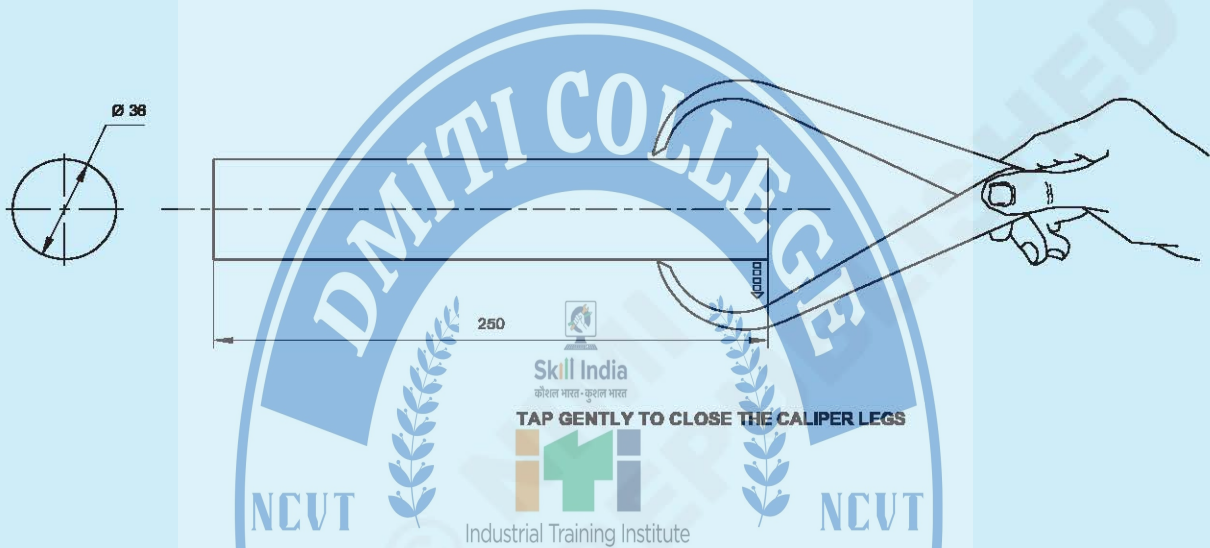
NCVT

GOVERNMENT OF INDIA

Measure the diameter using outside caliper and steel rule

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

- check the diameter using outside caliper
- measure diameter with steel rule.



Job Sequence

- Hold the out side caliper
- Open the caliper legs approximately more than job diameter
- Adjust the outside caliper leg to touch the diameter of the job.
- Adjust the caliper legs to just touch outer diameter of the job.
- Repeat the same procedure in different position of the job
- Measure the diameter by using steel rule.

1	—	1.7.93	Fe310	—	—	1.7.94
NO.OFF	STOCK SIZE	SEMI PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.
SCALE : NTS	MEASURE THE DIAMETER USING OUTSIDE CALIPER AND STEEL RULE				TOLERANCE: ±0.5mm	TIME:
					CODE NO: FI20N1794E1	

Skill Sequence

Measuring with outside calipers

Objectives: This shall help you to

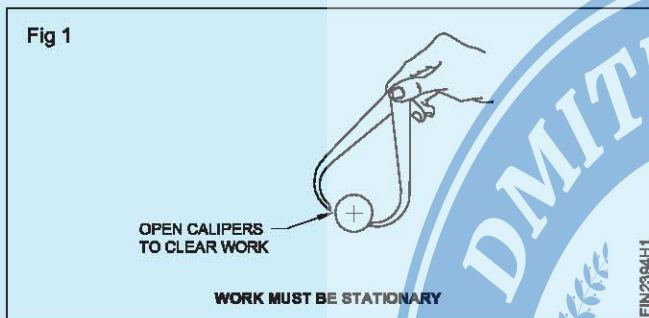
- select the right capacity caliper for measurement
- set the sizes both in firm joint and spring calipers
- read the sizes by transferring them to a steel rule or other precision measuring devices as the case may be.

Outside calipers

Select a caliper based on the diameter to be measured.

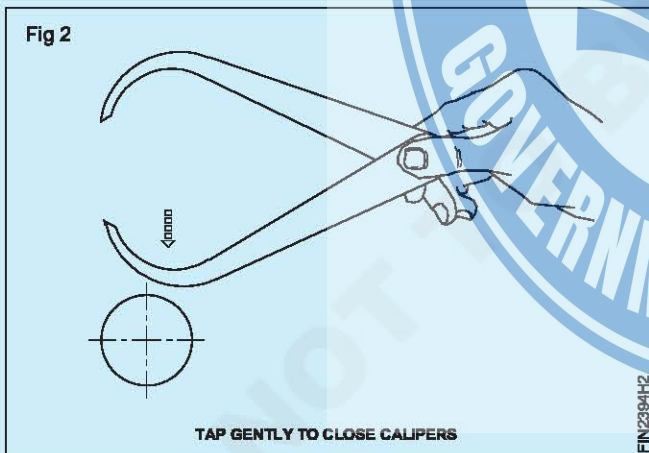
A 150 mm capacity outside caliper is able to measure sizes from 0-150 mm.

Open out the jaws of the calipers until they pass clearly over the diameter to be measured. The work must be stationary when measuring the sizes. (Fig 1)



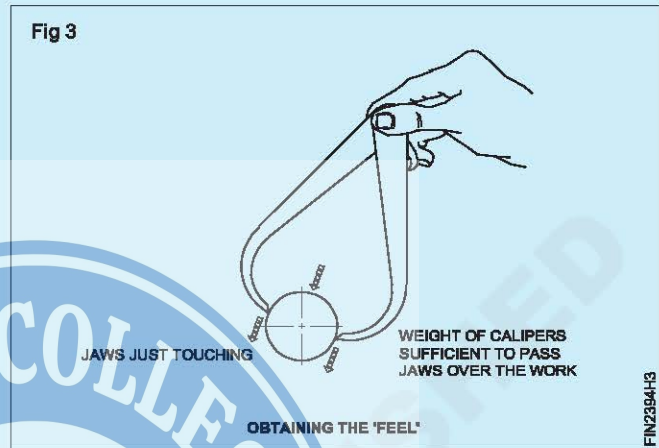
Place one point of the leg over the workpiece and get the sense of feel of the other point of the leg.

If there is clearance on the other point of the leg, gently tap the back of one leg of the firm joint calipers so that it just slips from the external diameter of the workpiece to give the right sense of 'feel'. (Fig 2)



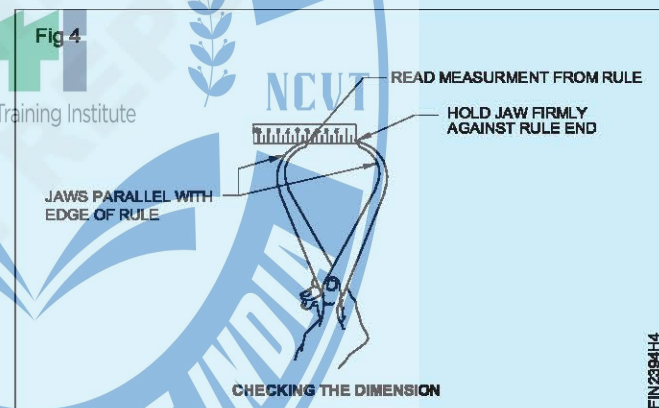
Because the accuracy of reading the sizes depends mainly upon the sense of feel of the user, high care should be exercised to get the correct feel. (Fig 3)

In the case of spring outside calipers, adjust the screw nut so that the adjustment of the caliper just slips from the external diameter of the workpiece to give the right sense of feel.



When you have adjusted the outside caliper for the correct 'feel' transfer the measurement to a steel rule or any other precision measuring instrument as the case may be.

Keep the graduated steel rule on a flat surface and hold the point of one leg firmly against the rule end. (Fig 4)



The point of one leg must be placed over the graduation so that the point of the other leg is parallel with the edge of the steel rule.

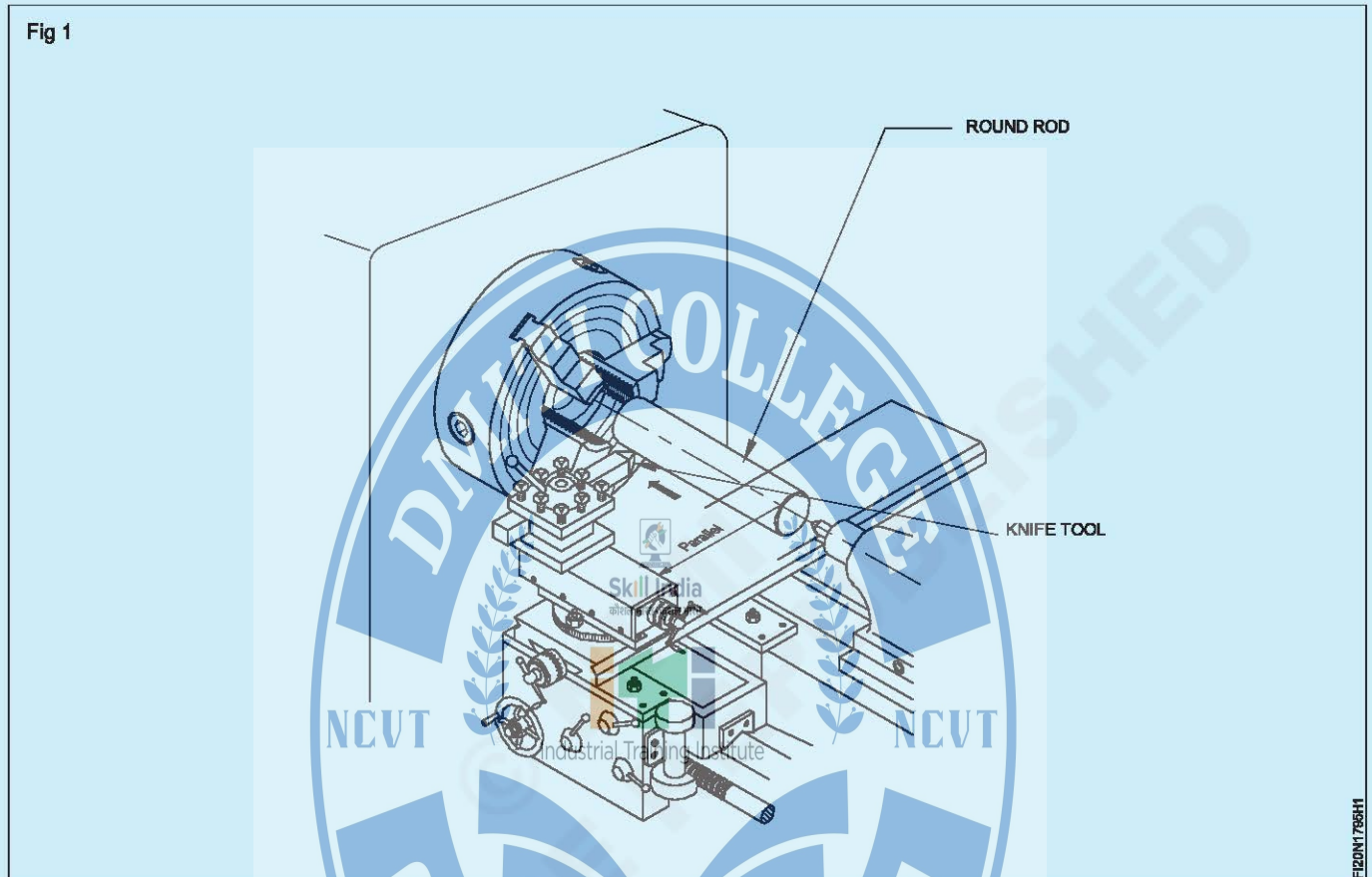
Record the reading to an accuracy of ± 0.5 mm.

In the case of precision measurements, transfer the measurements over an inside micrometer or vernier caliper. This measurement will give an accuracy of ± 0.01 or ± 0.02 mm. Here, the sense of feel of the user is very important in deciding the reading.

Holding job in three jaw chuck

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

- hold the job in three jaw chuck.



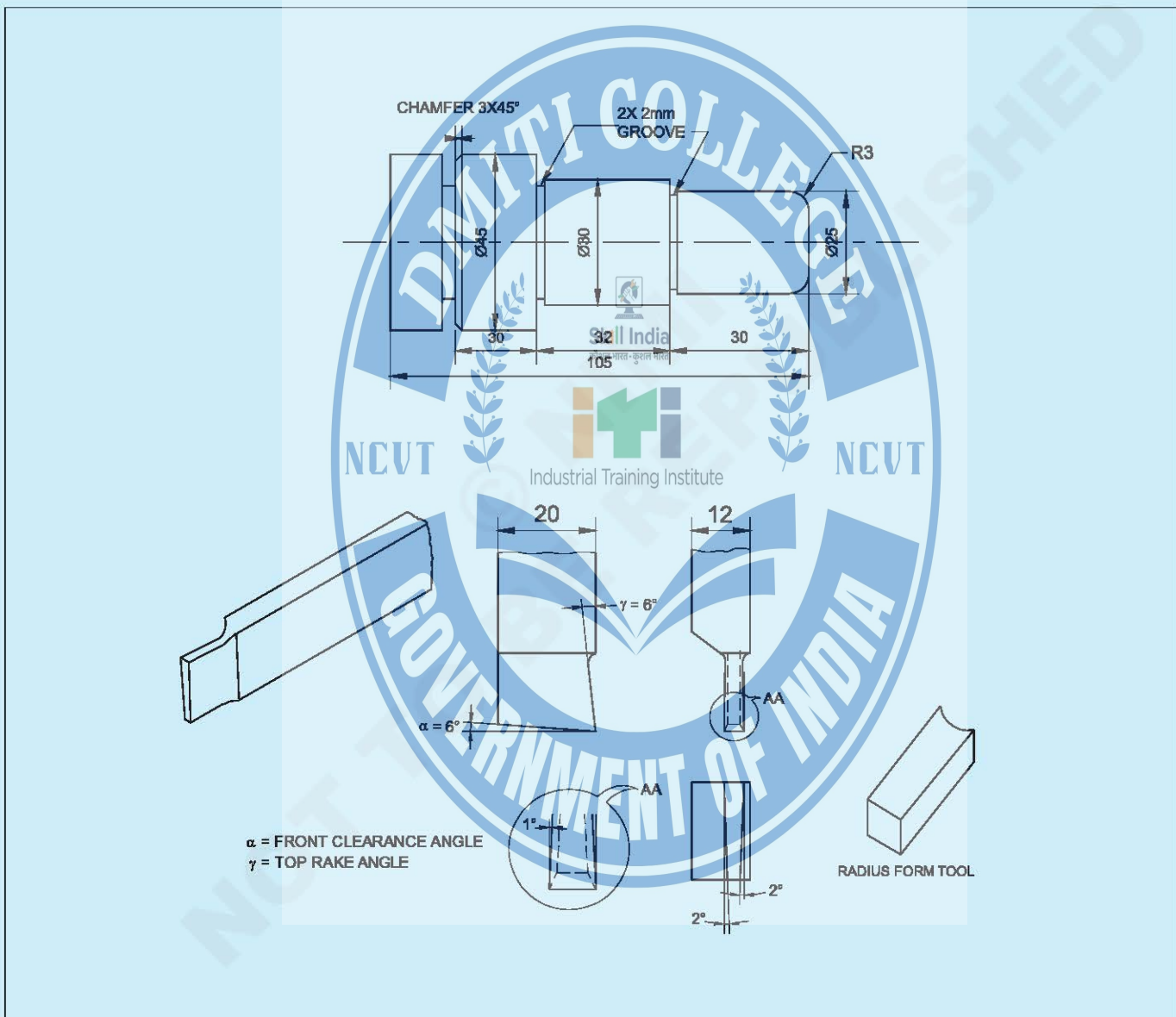
Job Sequence

- Check the raw material size.
- Open the jaws by chuck key
- Open the jaws approximately more than job diameter
- Keep the job in the chuck with overhang of 75 mm
- Tighten the jaws as required
- Check the trueness of the job

Perform the facing, plain turn, step turn, parting, deburr, chamfer corner, round the ends, and use form tools

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

- hold the work in three jaw chuck
- face the end and plain turning
- step turn job ± 0.1 using power feed and a knife tool
- form turning using form tool
- parting off using parting tool
- grind parting tool and make under cut to the required size.



1	Ø50-105	-	Fe 310-O	-	-	1.7.96
NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.
SCALE : NTS	PERFORM THE FACING ,PLAIN TURN, STEP TURN,PARTING,DEBURR, CHAMFER CORNER, ROUND THE ENDS AND USE FORM TOOLS				TOLERANCE $\pm 0.04\text{mm}$	TIME :
					CODE NO. F120N1798E1	

Job Sequence

- Check the raw material for its size.
- Hold the job in a 3 jaw chuck keeping about 75mm outside the chuck.
- Set the tool to the correct centre height.
- Select and set the correct spindle R.P.M.
- Face one side first and turn the outer diameter to \varnothing 45mm for the maximum possible length.
- Turn \varnothing 30 mm x 32 mm length as shown in job drawing.
- Turn \varnothing 25 mm x 30 mm length.
- Set the under cut tool, radius tool, to the correct centre height and hold it rigidly.
- Form a square groove 2 mm depth x 2 mm width at 30 mm and 62mm from the end face.
- Form a radius 3 mm at \varnothing 25mm x width 30 mm at the end face.
- Reverse and hold the job at \varnothing 30mm x width 32mm inside the three jaw chuck keeping about 40mm length outside the chuck and true the job.
- Turn \varnothing 45mm x 40mm length.
- Set the 2mm width parting tool to the correct centre height
- Part the job using plunge cut method at \varnothing 45 mm x width 8mm from the end face.
- Face the other end to a total length of 92mm.
- Set the chamfering tool to the correct centre height.
- Chamfer the \varnothing 45mm corner to $3 \times 45^\circ$.
- Remove the sharp edge.
- Check the dimensions.

Skill Sequence

Turning steps of different diameters

Objectives: This shall help you to

- turn steps of different diameters for definite length on a shaft.

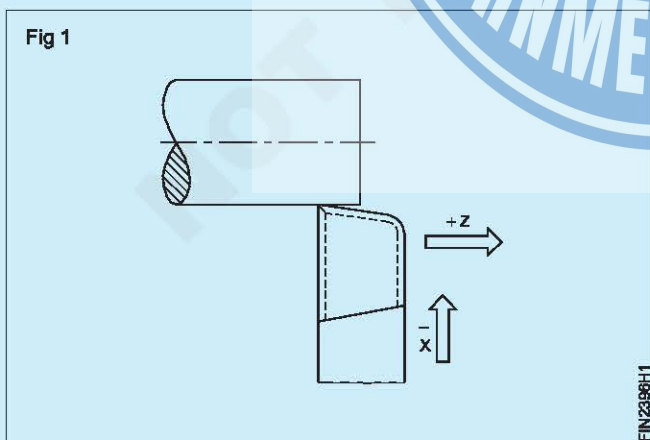
When the width of step to be turned is more than the width of the tool, it is turned by using a R.H. knife-edge tool.

Hold the previously turned shaft in three jaw chuck and true it at both ends (near the chuck and overhanging end).

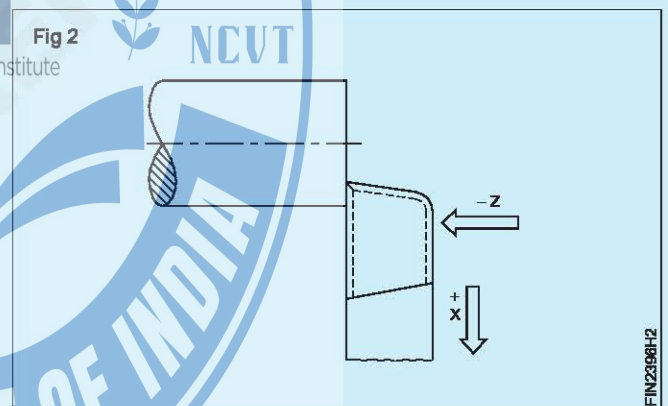
Hold the RH Knife-edge tool in the tool post with its cutting edge at centre height and at right angle.

Set the machine to 300 r.p.m.

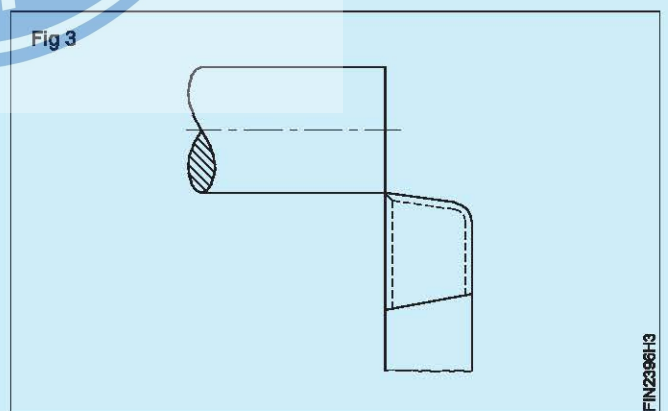
Start the machine and touch the tool tip on the surface of work to set the cross-slide graduated collar to zero with the backlash eliminated. (Fig 1)



Withdraw the tool from work and make the cutting edge contact the face of the work to set the top slide graduated collar to zero with a backlash eliminated. (Fig.2)



Position the tool tip near the edge of the work. (Fig.3)



Give a depth of cut to turn steps progressively. (Fig. 4)

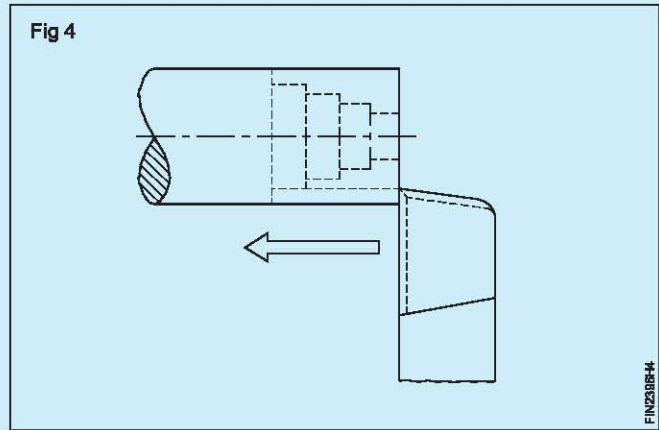
Advance the tool axially to the required length by rotating the top slide hand wheel.

(The rotation of the top slide hand wheel should be continuous and uniform till the required length is reached).

Restrict the depth of cut to a maximum of 3 mm for each cut.

Repeat the depth of cuts till the required diameter is reached.

Keep the carriage in the locked position.



Corner forming tool

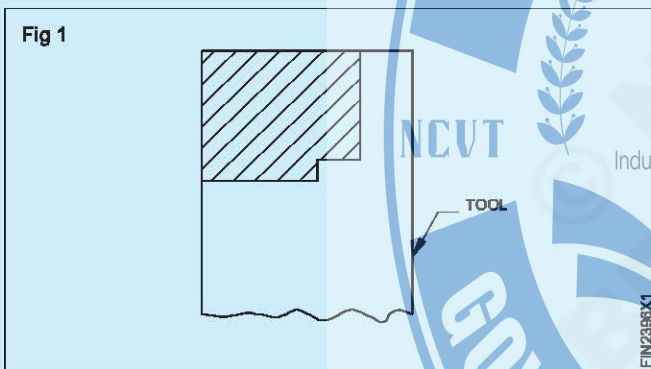
Objective: This shall help you to
 • **grind the corner forming tool.**

Procedure for grinding corner forming tool (for external operation).

Set the pedestal grinder for tool grinding.

Wear goggles.

Dress rough wheel, grind the tool approximately to a depth of 10.00 mm, maintaining a wall thickness of 3 to 4 mm. (Fig 1)



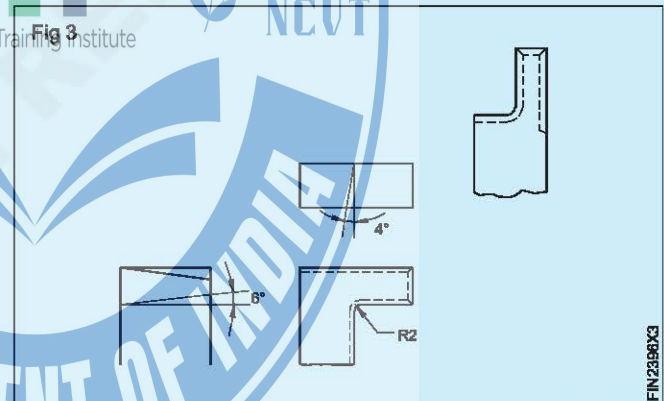
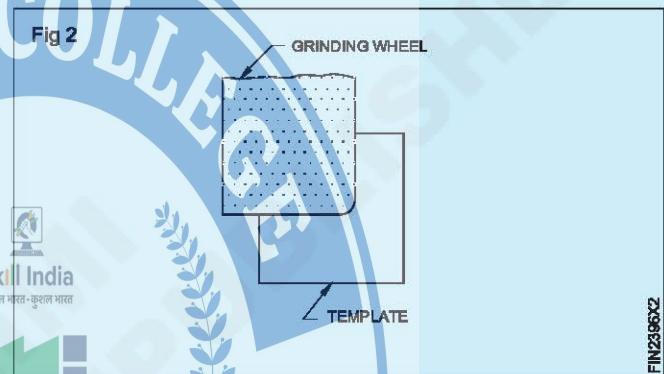
Prepare a template out of M.S. sheet to check positional accuracy. (Fig 2)

Now dress smoother wheel to form 2R at the corner.

Carborundum dresser is used to form corner radius.

Check the dressed wheel with template.

Grind 4°-6° front clearance angle at the front and 3°-4° side clearance. (Fig.3)



Remove the step and form 2R and check with template.

Deburr with oil stone.

No rake angle is given.

Parting off operation

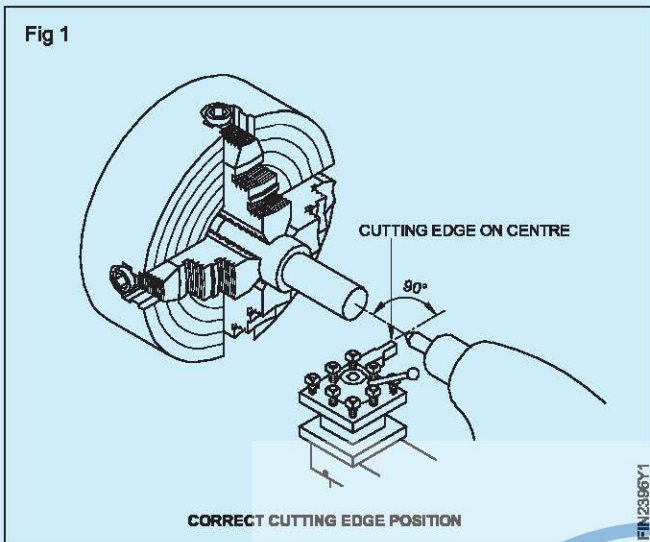
Objectives: This shall help you to
 • **set the parting off tool in the machine to the correct centre height**
 • **follow the correct procedure while parting off**
 • **observe certain precautions while parting off.**

Parting off operation

Parting off or cutting off is the operation of severing a finished part from the rough or finished stock.

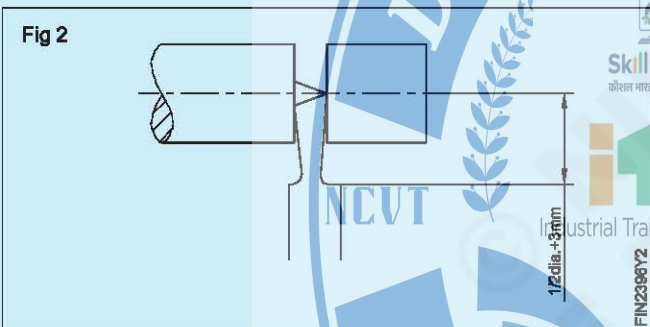
Setting of parting tool

Set the parting tool exactly on the centre with as little back rake as possible. (Fig 1)



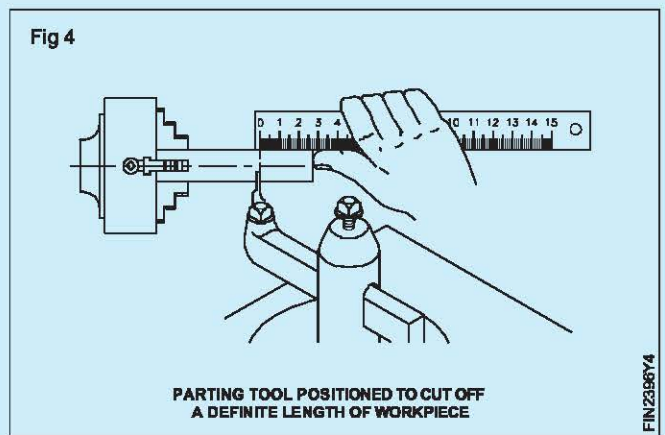
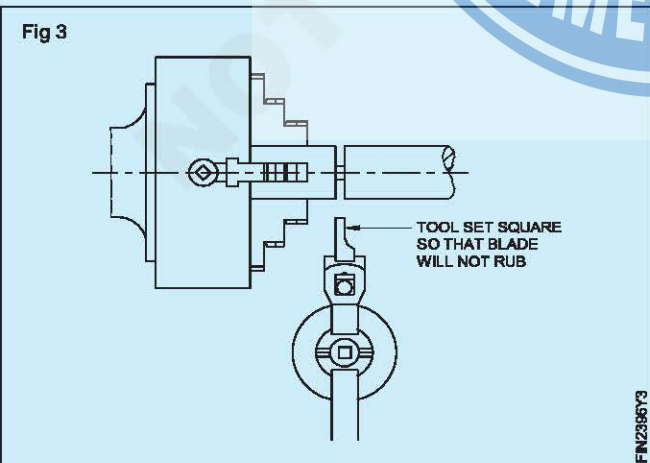
Adjust the parting off tool so that it extends one half the diameter of the work plus about 3mm for clearance from the tool-holder (Fig 2)

If the cutting tool is too high, it will not cut through the work piece. If it is too low, the work may be bent and the cutting tool damaged.

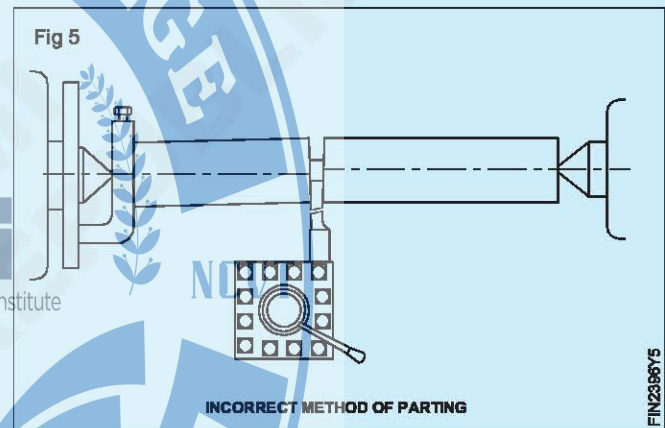


Procedure

- Select the correct type of tool for a specified job.
- Hold the work with the minimum overhang in a chuck.
- Set the tool square with the work so that it does not rub against the sides of the groove, as it is fed into the work (Fig 3)



- Set the spindle speed to half the speed for turning.
- Move the carriage so that the right hand side of the blade is at the point where the work is to be cut off. (Fig 4)
- Start the lathe and feed the tool steadily into the work using the cross-slide handle.
- Continue to feed the tool into the work until the part is severed.



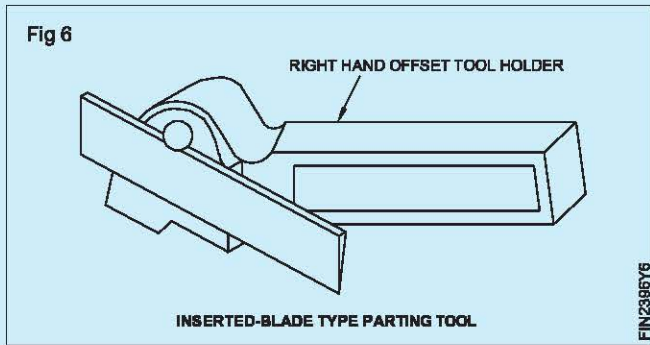
Precautions

- The work should protrude from the chuck jaws, sufficiently enough to permit the cut to be made as close as possible to the chuck jaws.
- The work must always be held securely in a chuck or a collet.

If the workpiece is held between centres, it may bend or break and fly out of the lathe during parting off. (Fig 5)

- Use a right hand offset tool-holder (Fig 6)
- A work having more than one diameter should be gripped on the large diameter while parting.

Intermittent feed tends to dull the tool's cutting edge.



Heavy feed causes jamming and tool breakage.

Use sufficient coolant on steel. Brass and cast iron should be cut off dry.

Make sure the saddle is locked during the entire operation.

Reduce the rate of feed, when the work is almost cut off.

While parting off long work, it should be supported with the tailstock centre.

If the machine is in good condition, the automatic cross feed may be used.

When the tool has penetrated to about the depth of its width, withdraw it and move it sideways with the compound slide and feed again.

The above operation should be repeated frequently to minimise the tendency of the tool to dig in and cause trouble.

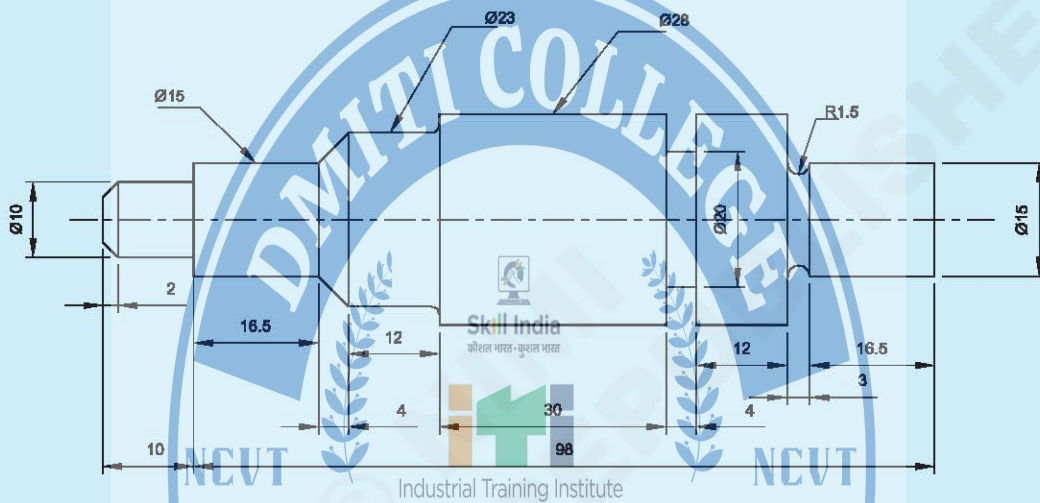
When the parting off operation is almost completed, hold the workpiece by hand to prevent it from falling, so that damage can be avoided.



Shoulder turn : Square , filleted, beveled under cut shoulder, turning-filleted under cut, square beveled


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

- set and true the job in a 3-jaw chuck
- face, plain and step turn the work by hand tool to an accuracy of ± 0.1 mm
- form square filleted under cut
- set and true the job to the run out accuracy of 0.1 mm
- turn square filleted shoulder
- turn bevel shoulder.



Job Sequence

- Check the raw material size.
- Hold the job in three jaw chuck with minimum overhang.
- Set the R.H. facing tool rigidly to the correct centre height with a minimum overhang.
- Set the machine to the predetermined R.P.M.
- Lock the carriage and face one end.
- Set the R.H. turning tool in the tool post rigidly.
- Turn the job to $\varnothing 28$ mm to the maximum possible length.
- Step turn $\varnothing 15$ mm to a length of 19.5 mm.
- Check the dimensions with a vernier caliper.
- Do filleted under cut R1.5x1.5 depth
- Make under cut of 4mm width and 4mm depth.
- Reverse the job and hold it on the finished surface.
- Face the job to 108 mm length.
- Check the length with a vernier caliper.
- Turn the job to $\varnothing 23$ mm to the 16 mm length.
- Step turn $\varnothing 15$ mm to a length of 26.5 mm.
- Bevel the $\varnothing 23$ mm step to an angle of $4 \times 45^\circ$.
- Turn $\varnothing 10$ mm x 10 mm length
- Chamfer the $\varnothing 10$ mm step to an angle of $2 \times 30^\circ$
- Remove burrs from the job.
- Check the angle with a vernier bevel protractor.
- Check the dimensions with a vernier caliper.

1	$\varnothing 30-120$	—	F8310	—	—	1.7.97
NO.OFF	STOCK SIZE	SEMI PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.
SCALE 1:1	SHOULDER TURN: SQUARE, FILLETED, BEVELED UNDER CUT SHOULDER, TURNING- FILLETED UNDER CUT, SQUARE BEVELED				DEVIATIONS LINEAR ± 0.04 mm ANGULAR $\pm 30'$	
					TIME	
CODE NO. F120N1797E1						

Skill Sequence

Form an undercut shoulder at the junction of two diameters

Objectives: This shall help you to

- set the undercutting tool in the tool post
- set the tool at the required position
- perform undercut operations
- check the undercut width and depth with a vernier caliper.

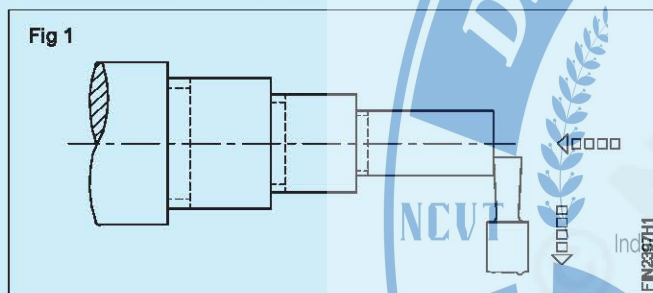
The end of a section to be threaded is mostly undercut to provide a channel into which the threading tool may run. It allows the mating part to sit squarely against it. When the diameter is to be finished to size by grinding, a channel is generally cut against the shoulder to provide a clearance for the grinding wheel, thus ensuring a square corner.

To form an undercut shoulder at the junction, the following procedure is to be followed.

Select a suitable tool bit or grind one to the shape and size required.

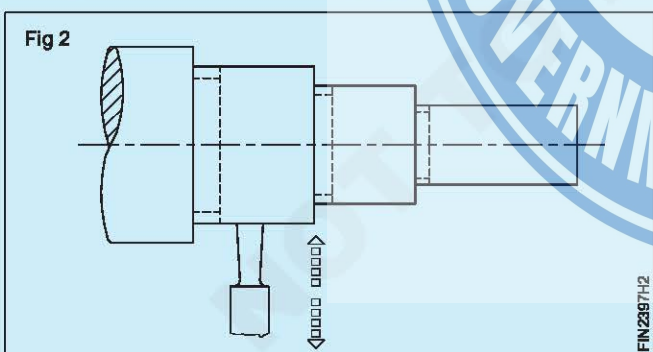
Set correct spindle speed, and start the machine.

Rotate the carriage handle until the tool almost touches the face of the work. (Fig. 1)



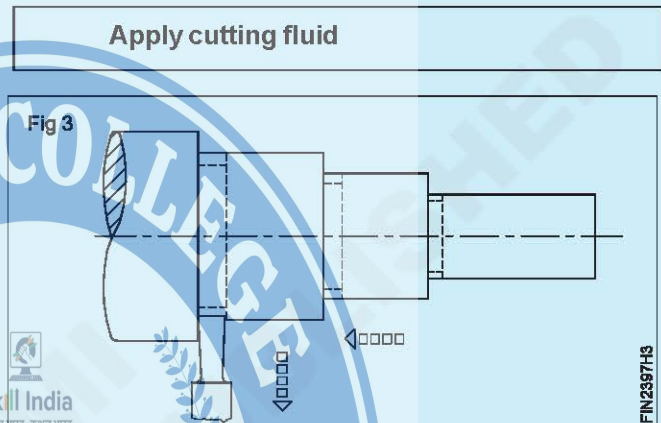
Lock the saddle in this position.

Rotate the cross-slide handle and touch the work surface lightly with the front cutting edge of the tool. Set the cross-slide graduated collar to zero. (Fig. 2)

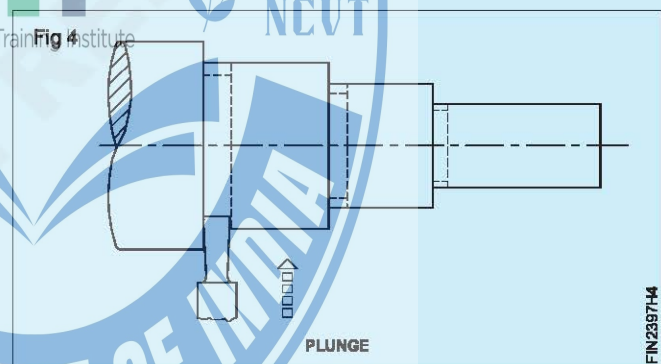


Rotate the top slide handle until the tool marks the shoulder lightly. (Fig 3)

Note the reading on the graduated collar of the top slide feed screw, and set the reading to zero.



Feed the tool slowly and evenly into the work to the required depth using cross-slide handle (Fig 4)



Stop the lathe and check the undercut for its dimensions.

Remove sharp corners, if any.

Machining various shoulders

Objective: This shall help you to

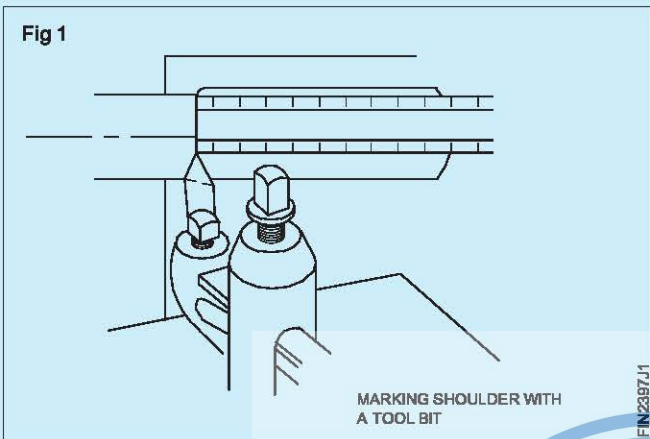
- machining various shoulders.

Machining a square shoulder

Face the end of the work to provide a reference surface point from which to take measurements.

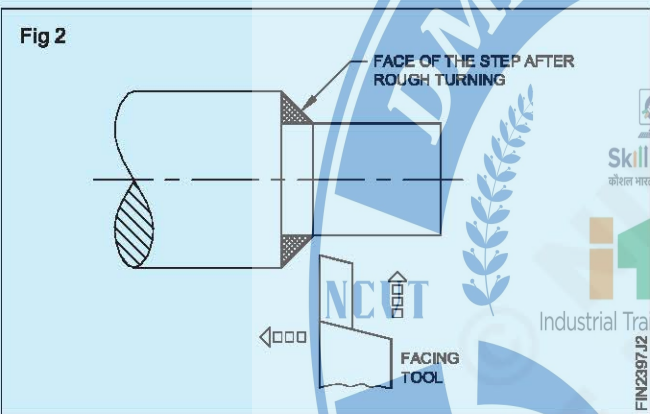
Lay out the position of the shoulder by one of the following methods.

Cut a light groove with the point of a sharp tool bit around the circumference of the work to mark the required length. (Fig 1)



Rough and finish turn the diameter to within about 1 mm of the required length.

Mount a facing tool bit in the tool-holder and set it to centre. (Fig 2)



Make sure that the tool bit is set up with the point close to the work, and with a slight space along the side cutting edge.

Apply chalk or lay out dye to the small diameter, as close to the shoulder as possible.

Before starting the lathe, the tool bit should be brought fairly close to the diameter, by using a piece of paper or thin stock between the tool bit point and the work diameter.

Start the lathe and bring the facing tool in until it just removes the chalk or the layout dye.

Note the reading on the graduated collar of the cross-slide screw.

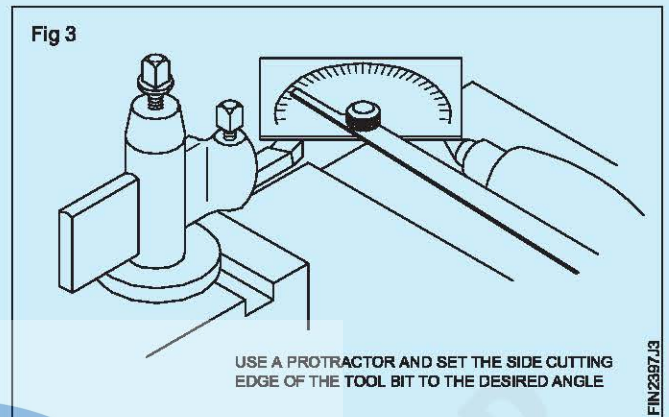
Bring the tool bit towards the shoulder with the carriage hand wheel until a cut is started.

Face the shoulder by turning the cross-slide handle anti-clockwise, thus cutting from the centre to the outside.

For successive cuts, return the cross-slide screw to the same graduated collar setting. Repeat the above procedure until the shoulder is machined to the correct length.

Machining a beveled shoulder (Fig 3)

Lay out the position of the shoulder along the length of the workpiece, and set the tool as shown in Fig 3.



Rough and finish turn the small diameter to size.

Mount a side cutting tool in the tool-holder and set it to centre.

Apply chalk or layout dye to the small diameter as close as possible to the shoulder location.

Bring the point of the tool bit in until it just removes the chalk or layout dye.

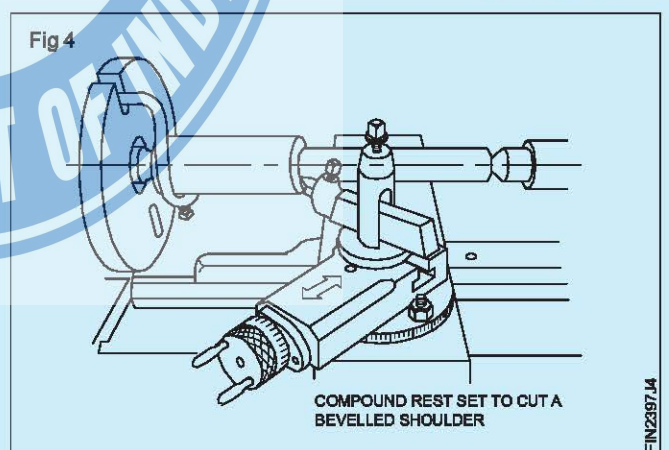
Turn the carriage hand wheel by hand to feed the cutting tool slowly into the shoulder.

Apply a cutting fluid to assist the cutting action and to produce a good surface finish.

Machine the beveled shoulder until it is to the required size.

If the size of the shoulder is large, and chatter occurs when cutting with the side of the tool bit, it may be necessary to cut the beveled shoulder using the compound rest.

Set the compound rest to the desired angle. (Fig 4)



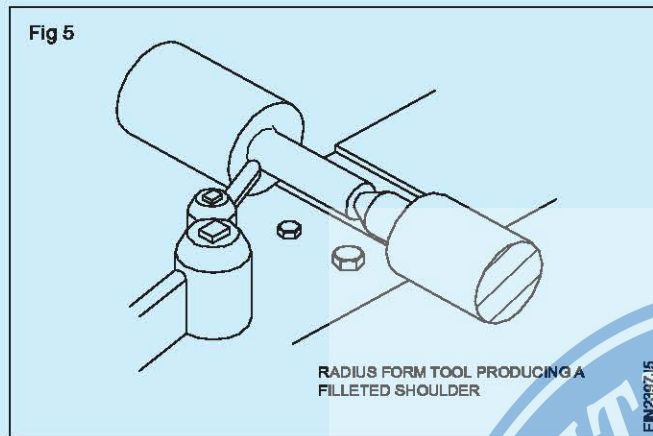
Adjust the tool bit so that only the point will cut.

Apply a cutting fluid to assist cutting action. Progressively machine the bevel. Always cut outwards and start each cut near the outermost edge of the face of the shoulder. Be careful not to damage the small diameter when preparing to make each new cut. At the start of the final cut, bring

the point of the tool bit in, until it just removes the chalk or layout dye at the innermost edge of the original shoulder face.

Machining a filleted shoulder (Fig 5)

Lay out or mark the location of the shoulder on the workpiece.



When laying out for a filleted shoulder make allowance for the radius to be cut. If a filleted shoulder has a 4 mm radius and is 60 mm from the end of the workpiece, the layout should be 56 mm from the end. This would leave material for cutting the radius.

Rough and finish turn the small diameter to size.

Mount a radius tool in the holder and set it to centre. Check the tool bit with a radius gauge to be sure that it has the correct radius.

Apply a layout dye or chalk to the small diameter as close as possible to the shoulder location.

Set the lathe spindle speed to approximately one half of the turning speed.

Start the lathe and bring the tool bit in until it just removes the layout dye or chalk.

Note the reading on the graduated collar of the cross-slide screw.

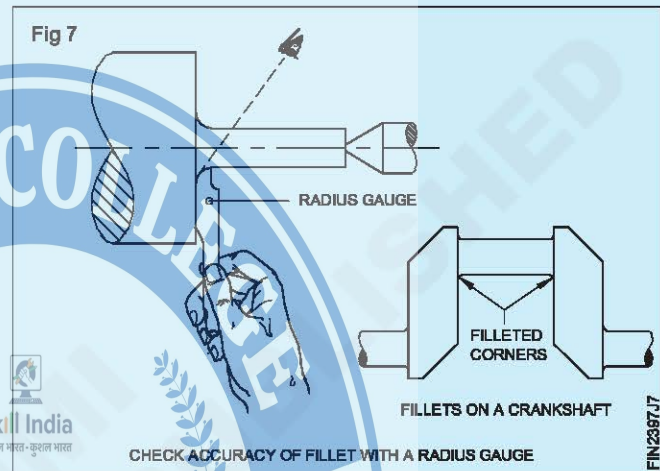
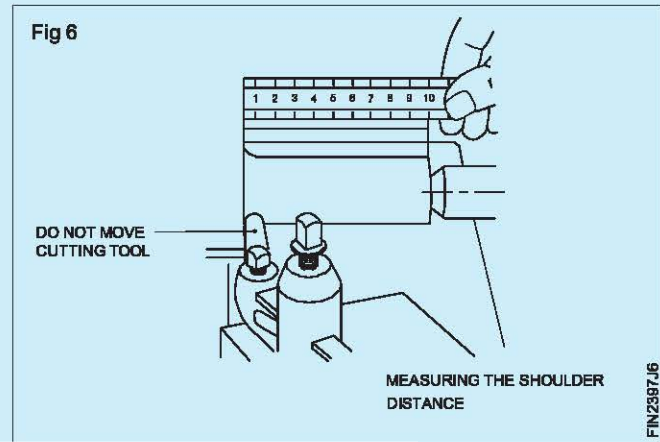
Retract the cutting tool by turning the cross-slide handle anticlockwise one half turn.

Turn the cross-slide handle clockwise until it is within approximately 1 mm of the original collar setting. The point of the round nose tool bit should now be about 1 mm away from the work diameter. This prevents the cutting tool from undercutting while roughing out the filleted corner.

Turn the carriage hand wheel slowly to start the radius tool cutting the filleted shoulder. If chattering occurs while machining the filleted corner, reduce the lathe speed and apply a cutting fluid to improve the finish of the fillet. (Fig 5)

Continue turning the carriage hand wheel slowly and carefully until the length of the shoulder is correct.

When stopping the lathe to measure the shoulder distance, do not move the cutting tool setting by withdrawing it from the diameter. (Fig 6)



Turn the carriage hand wheel to move the cutting tool away from the shoulder slightly.

Turn the cross-slide handle anticlockwise about 1 mm back to the original collar setting.

Finish the filleted corner by carefully advancing the radius tool bit with the carriage hand wheel.

If the radius is too large for a form tool bit, or too much chattering occurs, cut the fillet in steps, using the largest radius tool that does not cause chattering. Check the accuracy of the fillet with a radius gauge. (Fig 7)

Machining an undercut shoulder

Lay out the position of the undercut shoulder along the length of the workpiece.

Rough and finish turn the small diameter to size.

Mount the undercut tool in the tool-holder and set it to the centre.

Apply chalk or layout dye to the small diameter as close as possible to the undercut shoulder location and also on the face of the larger diameter.

Set the lathe spindle to approximately one half of the turning speed.

Bring the point of the tool bit in until it just removes the chalk or layout dye on the face and set the top slide graduated collar to zero.

Apply a cutting fluid to assist the cutting action and produce a good surface finish.

Retract the cutting tool by turning the cross-slide handle anticlockwise.

Repeat the above procedure until the undercut shoulder is machined to the correct depth.

Bring the tool tip clear off the large diameter face and advance the tool axially by 1 division of the top slide.

Feed the tool into the work from the edge of the larger diameter face, till it just removes the chalk mark applied on the small diameter.

Note the cross-slide graduated collar reading and advance the tool into the work to the number of divisions required according to the depth.

Ensure that the tool cutting edge is parallel to the work axis.

Ensure that the carriage is locked during the undercutting operation.

Apply a cutting fluid to assist the cutting action and to produce a good surface finish.

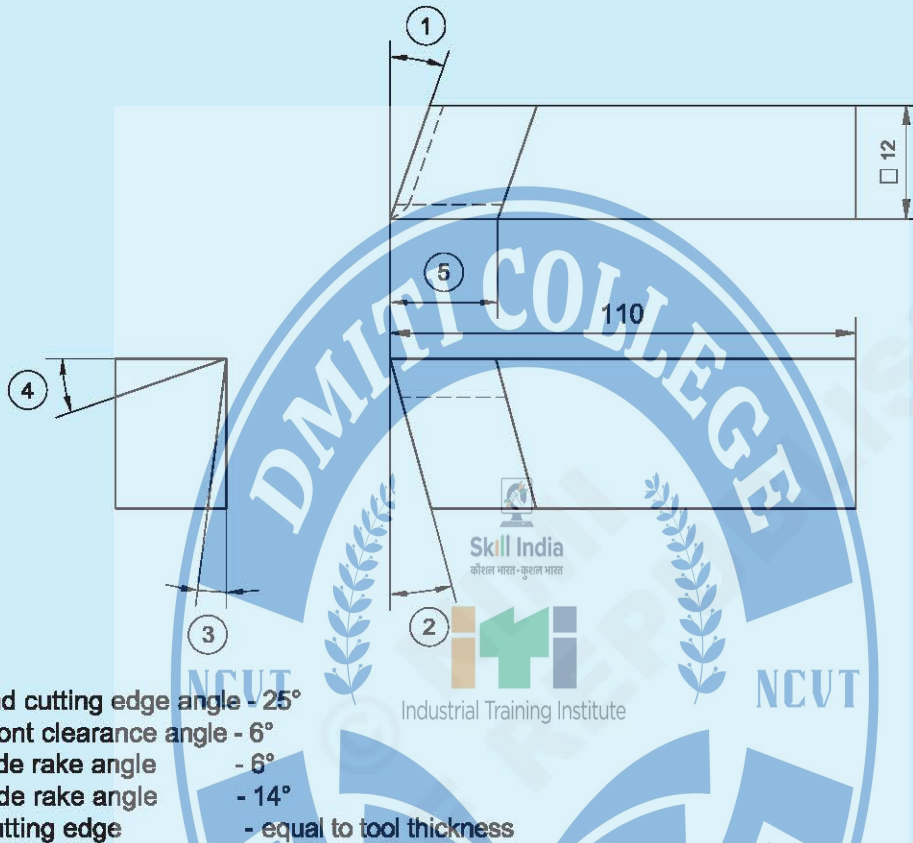
Retract the cutting tool by turning the cross-slide handle anticlockwise.

Repeat the above procedure until the undercut shoulder is machined to the correct depth.



Sharpening of - single point tools

Objective: At the end of this exercise you shall be able to
• grind side cutting tool for machining steel.



- 1 End cutting edge angle - 25°
- 2 Front clearance angle - 6°
- 3 Side rake angle - 6°
- 4 Side rake angle - 14°
- 5 Cutting edge - equal to tool thickness

Job Sequence

- Before starting wear safety goggles.
- Check the gap between the wheel and the tool rest, and maintain the gap 2 to 3 mm.

Damages or any corrections needed should be brought to the notice of the instructor.

- Hold the blank against the wheel to grind the end cutting edge angle 20° to 25° and the front clearance angle between 6° to 8° - simultaneously.

- Grind the side of the tool - for giving 6° to 8° side clearance. The side length should be equal to the width of the tool blank.
- Grind the top of the tool for a side rake angle of 12° to 15°.
- Finish grind all angles and clearances - on a smooth wheel.
- Grind a nose radius of approximately 0.5 mm R.

The ground surfaces should be without steps and should have a uniform smooth finish.

1	SQ12 - 110	-	Fe310	-	-	1.7.98
NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.

SCALE NTS	SHARPENING OF - SINGLE POINT TOOLS	TOLERANCE :±30'	TIME :
		CODE NO. F120N1798E1	

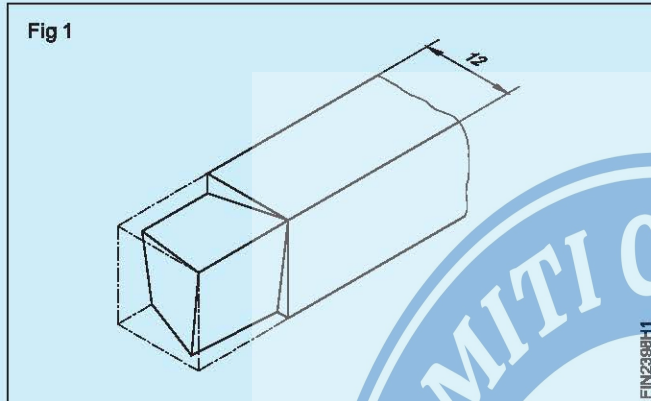
Skill Sequence

Grinding a side cutting tool for machining steel

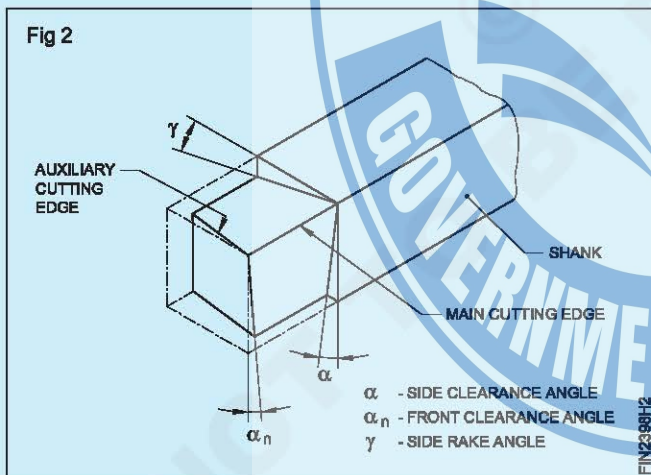
Objective : This shall help you to

- grind a right hand side cutting tool to machine steel.

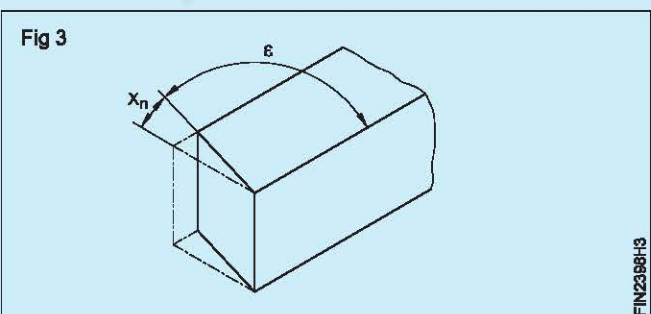
The side cutting tool to be used on steel is illustrated in Figure 1. The portion illustrates the tool blank in dotted lines before grinding, and the ground tool by thick lines. (Fig 1)



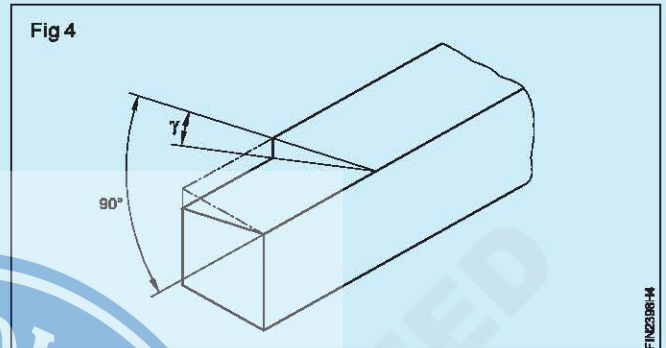
The side cutting edge is in line with the blank edge and the end cutting edge is inclined at an angle of 25° . The side rake angle is 14° . The front and side clearances are ground 6° . The length of the side cutting edge is maintained equal to the size of the square cross-section of the tool blank, i.e. 12 mm. Fig 2 shows the shaded portion to be removed by grinding the tool blank to get the ground tool. The procedure in sequence is as follows.



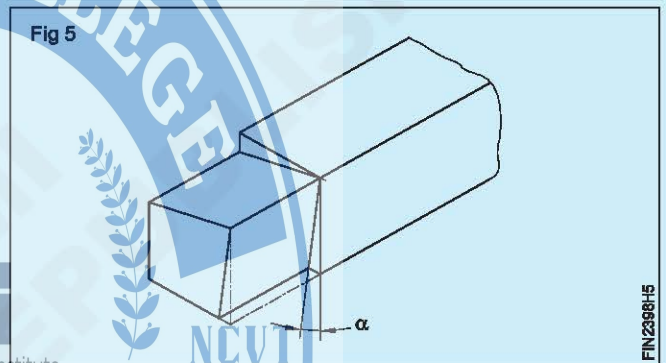
Grind the end cutting edge angle 25° . Angle ' x_n ' (Fig 3)



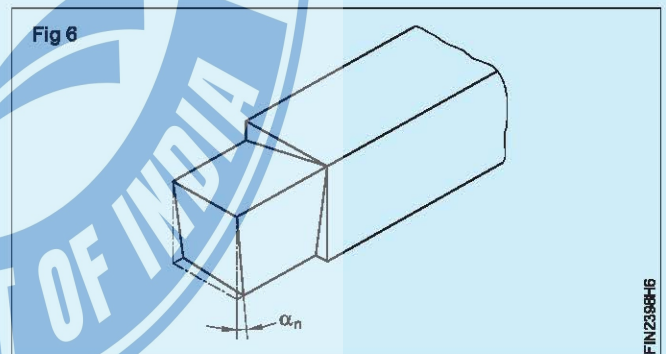
Grind the side rake angle of 14° . Angle ' r '. (Fig 4)



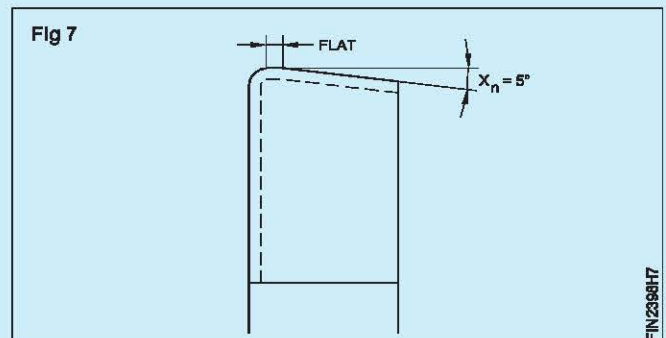
Grind the side clearance angle of 6° . Angle (Fig 5)



Grind the front clearance angle of 6° . Angle α_n (Fig 6)



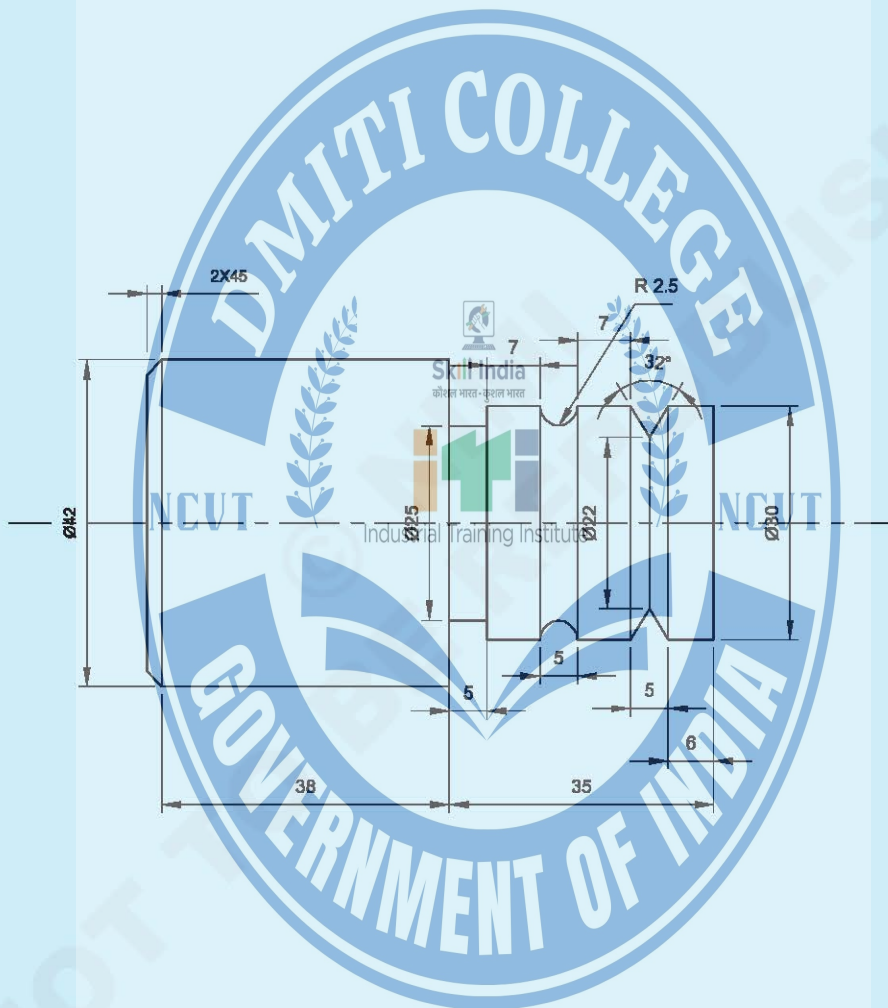
Grind and provide a nose radius of R 0.4 to R 0.6 mm at the point of tool. Grind a flat for a short length of 0.2 to 0.3 mm as shown in Fig 7. For the sake of clarity the figure is magnified.

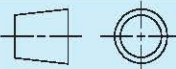


Cut grooves - square, round 'V' groove

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

- hold the job in lathe chuck
- set the turning tool
- set the machine spindle speed
- parallel turn the work piece by hand feed
- set the tool for groove turning -'V' tool, radius tool and square tool.



1	Ø50-80	—	Fe310	—	—	1.7.99
NO.OFF	STOCK SIZE	SEMI PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.
SCALE 1:1						TOLERANCE: ±0.04mm
						TIME:
						CODE NO : FI20N1799E1
CUT GROOVES - SQUARE, ROUND, 'V' GROOVE						

Job Sequence

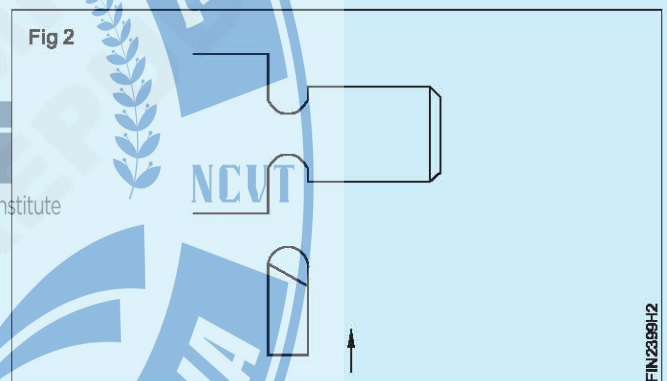
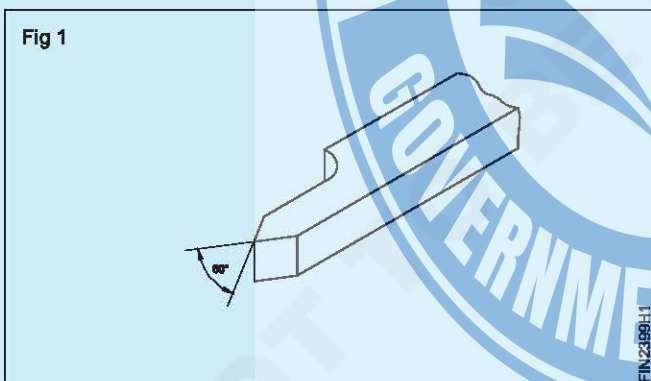
- Check the raw material for its size.
- Hold the job in 3 jaw chuck keeping about 50mm outside the chuck
- Set the tool to the correct centre height.
- Select and set the correct spindle R.P.M.
- Face one side first and turn the outer diameter to $\varnothing 42$ mm for the maximum possible length.
- Turn $\varnothing 30$ mm x 35 mm length.
- Set the under cut tool, radius tool, 'V' groove tool to the correct centre height and hold it rigidly.
- Form a square groove 2.5 mm depth x 5mm width at 30 mm from the end face.
- Form a radius groove 2.5 mm depth x 5mm width at 18 mm from the end face.
- Plunge the 'V' groove tool to form a 'V' groove 5mm width at 6mm from the end face.
- Reverse and hold the job.
- Face the other end to a total length of 75mm.
- Turn $\varnothing 42$ mm x 40 mm length
- Chamfers 2 x 45° at the 2 x 45° end
- Remove the sharp edge
- Check the dimensions.

Skill Sequence

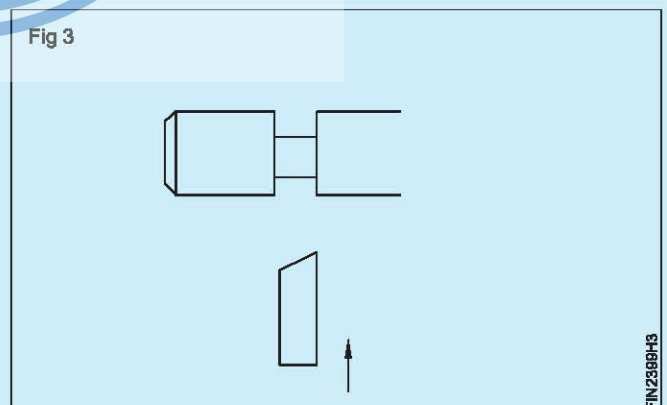
Grind 60° 'V' tool

Objective: This shall help you to
 • grind 60° 'V' tool.

- 1 Grind the tool to the given angle of 60°
 - Mount the tool and set centre height properly
 - Set the speed, lock the carriage
 - Move the cross slide and plunge the tool to the required size.
 - Check the depth of the 'V' groove. (Fig 1)



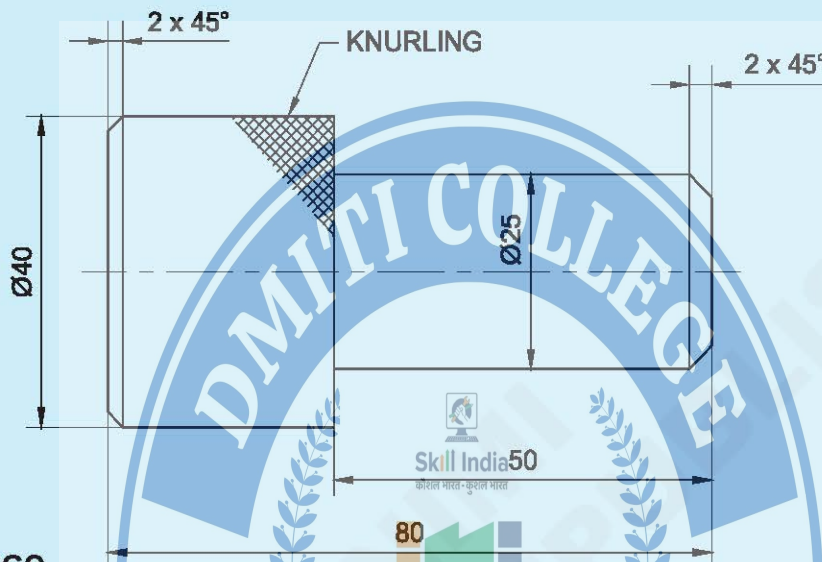
- 3 Grind the tool to the required width of 4 mm
 - Mount the tool and set centre height properly.
 - Set the speed, lock the carriage.
 - Move the cross slide and plunge the tool to the required size. (Fig 3)



Knurl the job

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

- hold the job in lathe chuck
- set a knurling tool in the tool post
- knurl on the cylindrical surface.



Job Sequence

- Check the raw material size
 - Hold the material securely in a 3 jaw chuck projecting 50 mm outside the chuck.
 - Face the one end.
 - Turn the job to $\varnothing 40-0.2$ for more than the required for knurling
 - Hold the diamond knurling tool securely and set it to the centre height.
 - Select the suitable speed for the knurling operation.
 - Knurl the surface till a diamond shape is formed
 - Chamfer $2 \times 45^\circ$ at the end.
 - Reverse and hold the job in the chuck and true the job.
 - Face the end and maintain the length of 80 mm.
- Industrial Training Turn the job $\varnothing 25 \times 50$ with a side knife tool. (Use a vernier caliper for measuring dimensions.)
- Chamfer to $2 \times 45^\circ$ at the end with a 45° chamfering tool.
 - Deburr all sharp edges.
- Remember**
- Avoid overhanging of the tool.
 - Use aluminium pieces for packing, to avoid marks on the knurled surface.
- Safety precautions**
- Never operate a lever when the machine is in motion.
 - Do not keep any tools on the moving parts of the machine.
 - Use a suitable coolant.

1	$\varnothing 45 - 85$	-	Fe 310	-	-	1.7.100
NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.
SCALE 1:1		KNURL THE JOB			TOLERANCE : $\pm 0.04\text{mm}$	TIME :
					CODE NO. FI20N17100E1	

Skill Sequence

Knurling on lathe

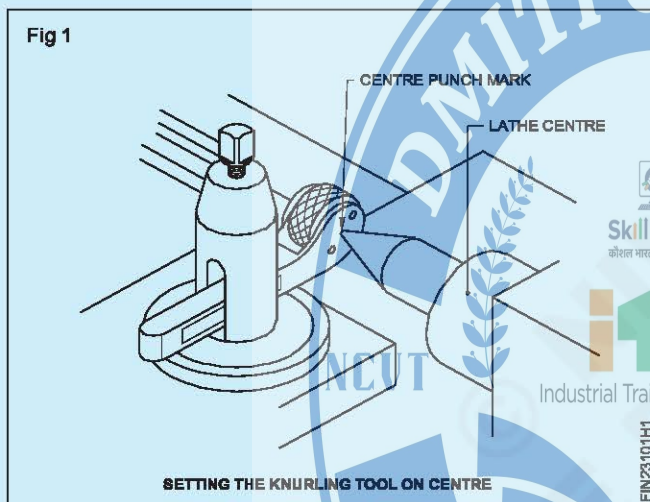
Objectives: This shall help to you

- prepare the work for knurling
- set the speed for knurling
- set the knurling tool in the tool post
- knurl the job using the required grade of knurl.

For better grip and for a good appearance on cylindrical surfaces, a portion of the component is knurled. The procedure of knurling, in sequence, is as follows.

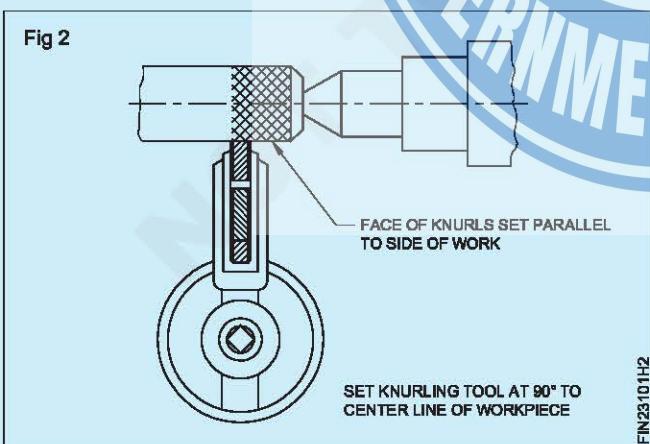
Reduce the diameter of the portion to be knurled depending upon the grade of knurl and material of the job. Reduce 0.1 mm for fine knurling, 0.2 mm for medium knurling and 0.3 mm for coarse knurling approximately.

Set the knurling tool in the tool post and align with the centre or tail stock (Fig 1)



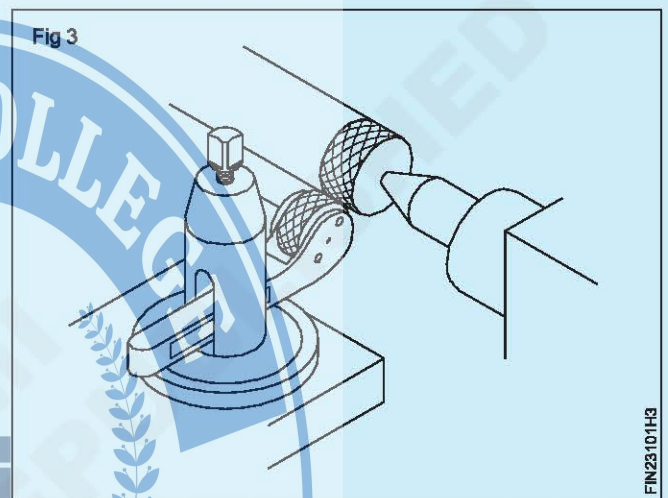
Set the machine for a low speed, preferably 1/3 to 1/4 of the turning speed. Mark off the length to be knurled.

Adjust the knurling tool so that it is at right angles to the axis of the work; tighten it firmly. (Fig 2)



Feed the knurl and make the knurls to contact the work periphery by the cross-slide hand wheel.

Move the carriage until about the face of the knurling roll overlaps the end of the workpiece which helps to produce a true pattern. (Fig 3)



Start the lathe and feed the knurling tool into the work by the cross-slide.

Stop the lathe and reset the knurling tool, if necessary.

Feeding the knurl into the workpiece, before it is rotated, may damage the knurl.

Move the knurling tool longitudinally with a uniform movement by the carriage hand wheel up to the required length of the work to be knurled.

Give the depth by the cross-slide without drawing the tool back. Feed the knurling tool to the other end.

Until the correct pattern is obtained, do not withdraw the knurling tool back.

Ample coolant is to be applied to the workpiece being knurled. This washes away any metal particles, and provides lubrication for the knurling rolls.

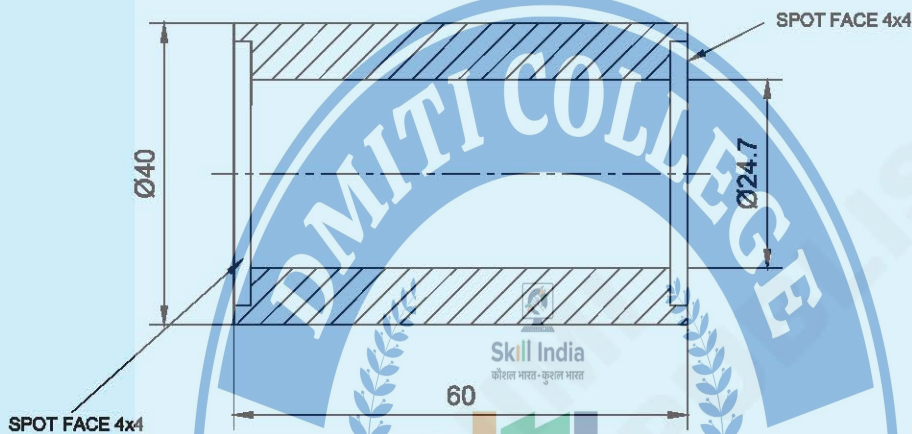
Use a fine feed for knurling hard metals and a coarse feed for knurling soft metals.

Clean the knurl with a brush for subsequent cuts.

Bore holes - spot face, pilot drill, enlarge hole using boring tools

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

- drill through hole
- bore a hole to an accuracy of ± 0.04 mm with boring tool
- measure the bore by using a vernier caliper
- re-shapen a twist drill
- check the twist drill for its performance
- spot face the end of bored hole.



Job Sequence

- Check the raw material for its size.
- Hold the job in a 4 jaw chuck and true it, keeping about 45mm outside the chuck.
- Set the facing tool to the correct centre height.
- Select and set the correct spindle speed, for facing.
- Face one side first, and turn the outer diameter to $\text{Ø}40$ mm for the maximum possible length.
- Centre drill.
- Select the required size of drills including the pilot drill.
- Hold the drill in the tailstock spindle with the help of suitable sleeves after cleaning.
- Select the spindle speed for drilling the pilot hole of 12mm dia.
- Bring the tailstock to a convenient position for drilling, and lock the tailstock on the bed.
- Run the lathe and advance the drill, so that it does the drilling operation on the job held in the chuck.
- Use coolant while drilling and advance the drill slowly.
- Enlarge $\text{Ø}12$ mm hole to $\text{Ø}20$ mm hole by drilling at a reduced spindle speed.
- Set the boring tool in the tool post to the centre height and bore the drilled hole to $\text{Ø}24.7$ mm through.
- Check the bore with vernier caliper.
- Make spot face 4x4 mm by boring tool
- After completion of drilling throughout the job reverse and true the job; face to the required length as per drawing, and turn outer dia $\text{Ø}40$ mm.

1	Ø45 - 65	-	Fe 310	-	-	1.7.101
NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.
SCALE 1:1		BORE HOLES - SPOT FACE, PILOT DRILL, ENLARGE HOLE USING BORING TOOLS.			TOLERANCE : ± 0.04 mm	TIME :
					CODE NO. FI20N17101E1	

- Make spot face by boring tool 4x4 mm

Safety precautions

- Select proper spindle speeds as per size and operation.

- Use pilot drill while drilling more than 20mm drill size.
- Feed the drill slowly while drilling.
- Use coolant while drilling.

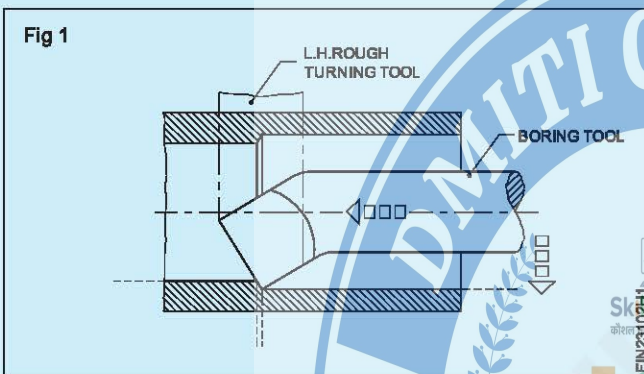
Skill Sequence

Boring a drilled hole

Objectives: This shall help you to

- set the boring tool in the tool post
- bore the drilled hole to the required size
- check the hole with the help of a vernier caliper.

Boring is an internal operation of enlarging a hole with the help of a single point cutting tool. (Fig 1)



To bore the hole the following procedure is to be followed.

Mount the workpiece in a four jaw chuck. True the face of the work and the outer diameter.

Set the lathe to the proper spindle speed for boring.

Mount the boring tool on the tool post of the compound rest.

Fix the boring tool, level and parallel to the centre line of the lathe.

Grip the boring tool as short as possible to reduce chatter.

Use the largest diameter boring tool which can be accommodated in the drilled hole. (Approximately 2/3rd size of the bore)

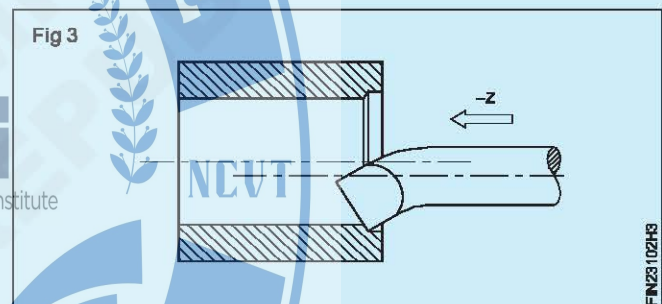
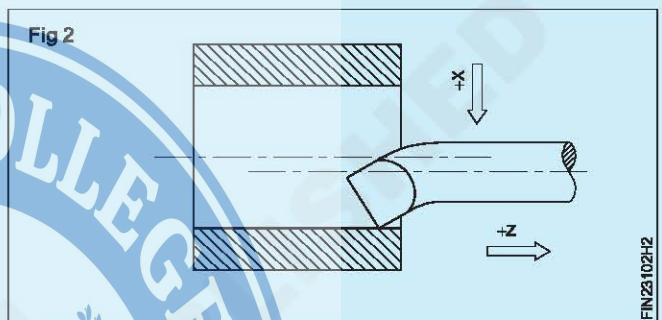
Set the cutting edge of the cutting tool just slightly above the centre line, since there is tendency for the tool to spring downwards when cutting.

Choose a proper feed for rough boring.

The speed for boring is the same as that for turning and is calculated for the diameter of the bore.

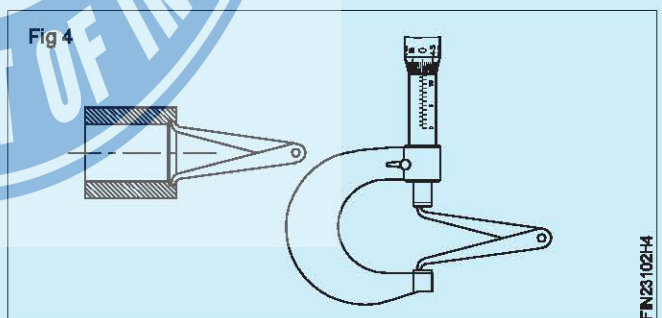
Start the machine and turn the cross-slide handle anticlockwise until the cutting tool touches the inside surface of the hole. (Fig 2)

Take a light trail cut about 0.2 mm deep and about 8 mm long at the right hand end of the work. (Fig 3)



Stop the machine and measure the diameter using a telescopic gauge or inside caliper. (Fig. 4)

Calculate the amount of material to be removed from the hole for the roughing cut.



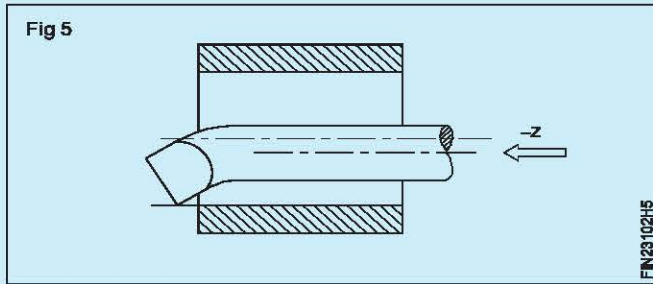
Leave about 0.5 mm undersize for a finish cut.

Take a roughing cut for the required length. (Fig 5)

Keep the machine and move the carriage to the right until the boring tool clears the hole. (Fig 6)

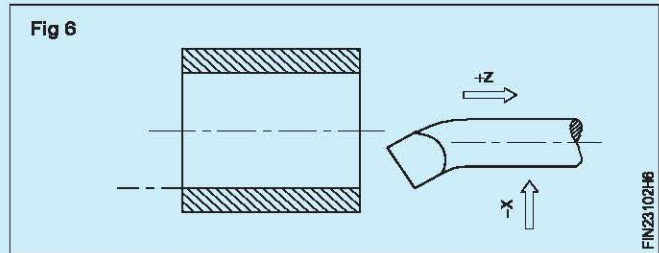
Set a fine feed of about 0.1 mm for the finish cut.

Set the cutting tool for the required depth to get the finished bore size.



Use the cross-slide graduated collar.

Finish the boring operation and measure with a vernier caliper.



To avoid bell mouth, repeat the same cut.

Several cuts taken without adjusting the depth of cut would correct bell mounting.

Remove the sharp corners.

Inside caliper & outside micrometer used for bore measurement

Objective: This shall help you to

- take the measurement of a bored hole with an inside caliper, transfer it to an outside micrometer and read the measurement.

Bores are checked for their dimensional accuracy by using:

- Inside micrometers.
- Universal vernier calipers.
- Inside calipers and outside micrometers (transfer measurement).
- Telescopic gauges and outside micrometers (transfer measurement).

The first two methods give direct reading whereas the 3rd and 4th are by transfer measurement.

For checking the bore diameters using inside calipers and outside micrometers the following sequence is to be followed.

Select the inside caliper according to the size of the bore to be measured.

Select an outside micrometer of suitable range for the size of the hole.

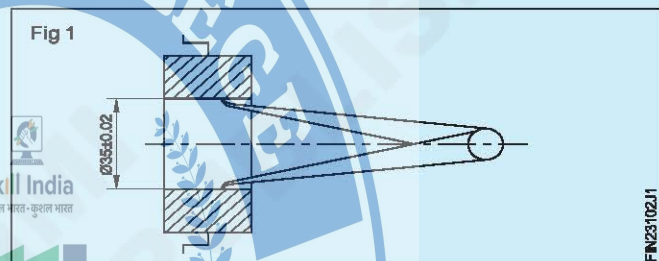
Open the legs of the inside caliper approximately permitting its entrance into the hole.

Position one leg in contact with the bottom of the bore.

Keeping this as the fulcrum, oscillate the other leg in the bore.

Adjust the distance between the legs by gentle tapping to increase or to decrease so as to enable the leg to enter.

Rock the inside caliper with respect to the axis of the work so as to make the leg of the inside caliper contact the bore top surface. (Fig 1)

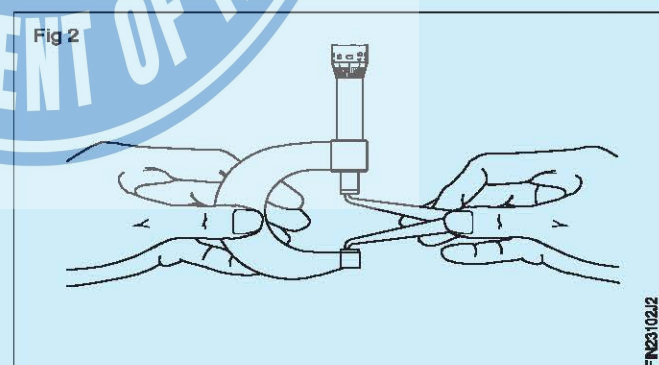


Ensure that the position of the legs is not disturbed, once the correct feel is obtained.

Hold the outside micrometer in one hand, and the spindle away from the anvil face, a little more than the distance between the two legs of the inside caliper.

Hold the inside caliper with the other hand, contacting the tip of one leg with the anvil face of the micrometer.

Oscillate the other leg and rotate the thimble of the outside micrometer to contact the tip of the oscillating leg of the inside caliper. (Fig 2)



Ensure you get the same 'feel' as before.

Note the readings on the barrel and thimble of the outside micrometer, and determine the size of the measurement.

The accuracy depends on the skill. Practice to get the correct feel for the measurement.

If the 'feel' is hard, reduce the distance between the leg tips and if the feel is less or if there is no feel, increase the distance between the leg tips slightly.

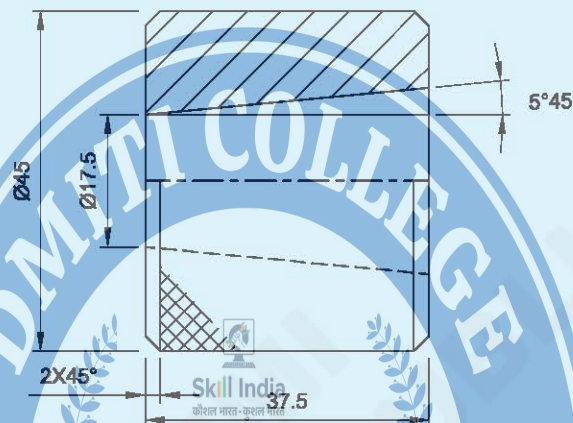
Check once again and repeat till you get the correct feel.

Turn taper (internal and external)

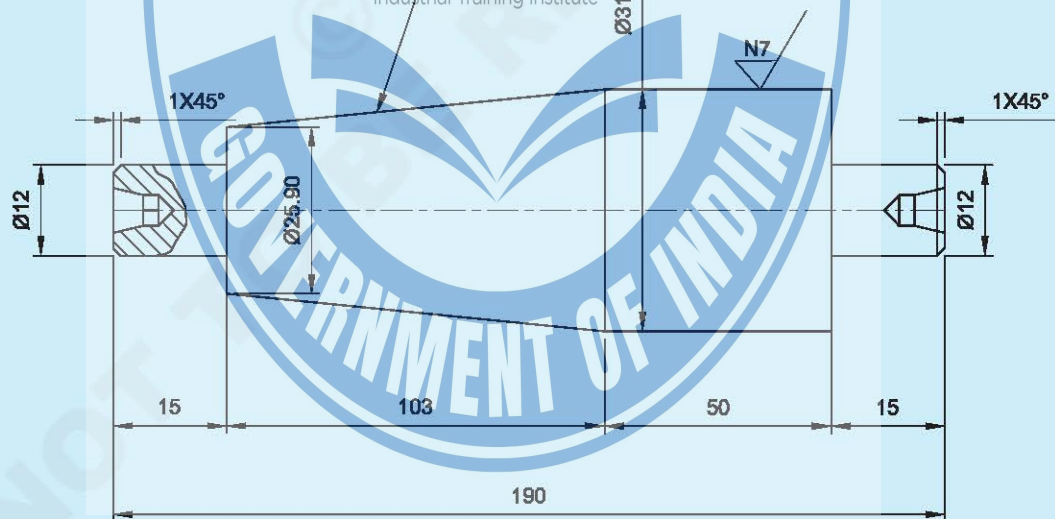
Objectives: At the end of this exercise you shall be able to

- hold the work in between centres
- produce taper bore by compound slide
- set the compound rest to the specified angle
- turn the external taper by the compound rest method
- check the taper with a vernier bevel protector.

TASK 1



TASK 2



1	Ø50 - 45		Fe 310		TASK 1	
1	Ø36 - 200	-	Fe 310	-	TASK 2	1.7.102
NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.
SCALE 1:1	TURN TAPER (INTERNAL AND EXTERNAL)				TOLERANCE : ± 0.04mm	TIME :
					CODE NO. F120N17102E1	

Job Sequence

TASK 1: Taper turning internal

- Hold the job in a 4 jaw chuck and true it.
 - Set the tool to correct centre height.
 - Face one end of the job.
 - Turn \varnothing 45 mm to a length of 45 mm.
 - Drill pilot hole \varnothing 16 mm by drilling
 - Chamfer $2 \times 45^\circ$.
 - Set the parting tool to centre height and cut off to a length of 40 mm.
 - Hold the knurled job and face the ends to maintain a length of 37.5 mm.
 - Chamfer the end to $2 \times 45^\circ$.
- Set the compound rest to the $5^\circ 45'$ with the help of a vernier bevel protractor.
 - Set the boring tool, to the correct centre height.
 - Turn taper as per drawing.
 - Match the taper.

Safety precautions

- Remove all sharp corners.
- Use slow speed while knurling.
- Use plenty of coolant while drilling, taper turning and knurling.

TASK 2: Taper turning external

- Check the raw material size.
 - Hold the job in between centres.
 - Turn the step $\varnothing 12 \times 15$ mm long at the taper end.
 - Reverse and refit between centres.
 - Turn the step $\varnothing 12 \times 15$ mm long from the other end of job.
 - Calculate the setting angle of the compound rest using the formula
- Swivel the compound rest slide to the above angle using a vernier bevel protractor.
 - Turn the taper by using the top slide feed and maintain the major dia. to 31.26 mm. minor dia to 25.90 mm and length to 103 mm.
 - Check the size of the job with a vernier bevel protractor and vernier caliper.

Skill Sequence

Checking a tapered bore using a taper limit plug gauges

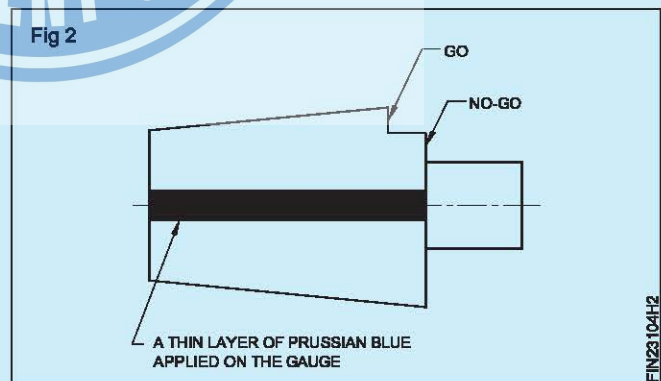
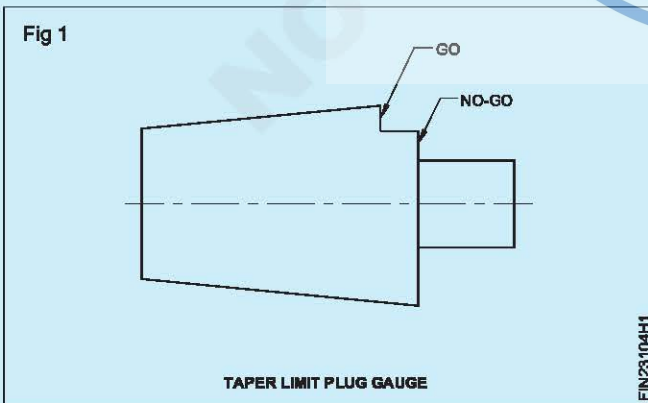
Objective: This shall help to you

- 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)

Clean the tapered bore.

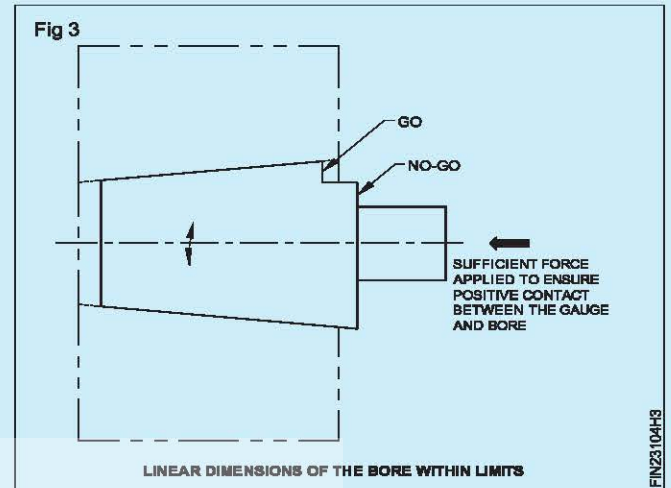
Apply a thin layer of prussion blue on the traper limit plug gauge along its length. (Fig 2)



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 prussion 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 this tapered bore. (Fig 3)

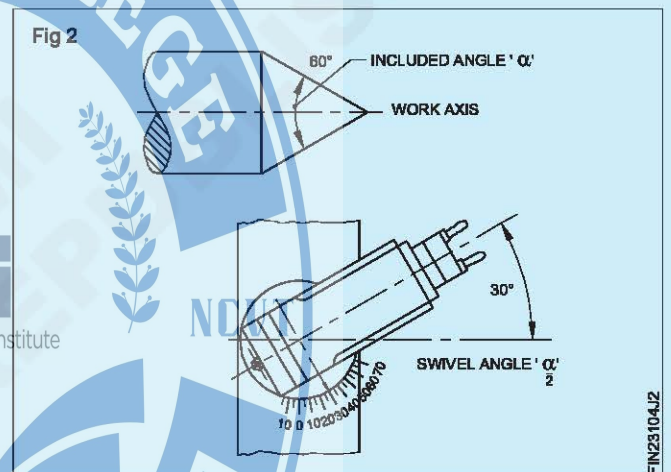
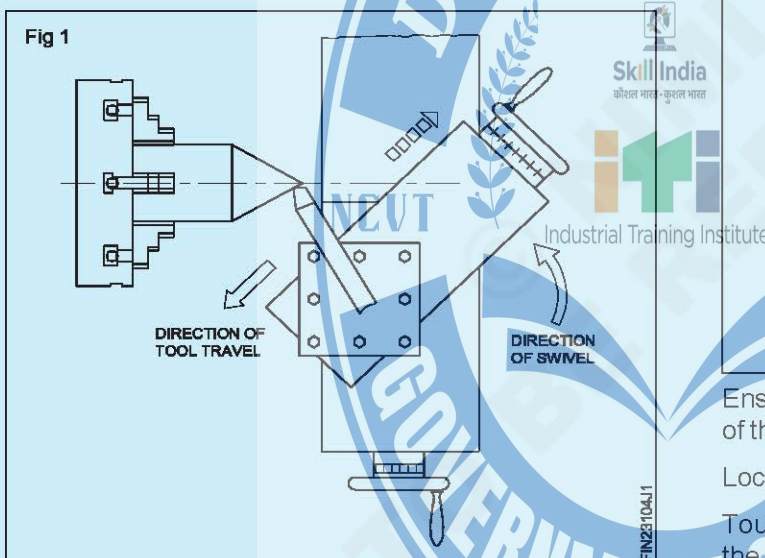


Turning taper by compound slide swivelling

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 swiveling the compound slide and feeding the tool at an angle to the axis of the work by hand feed. (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.

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.

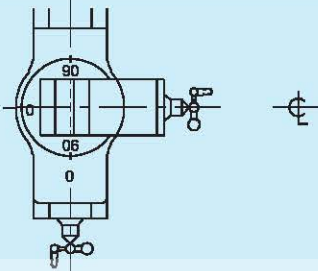
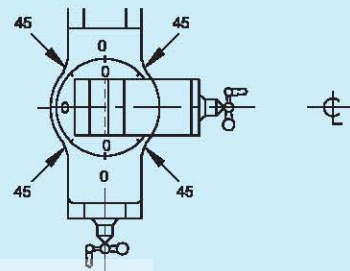
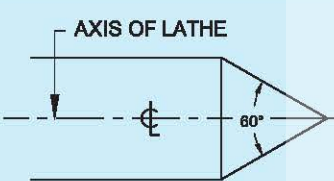


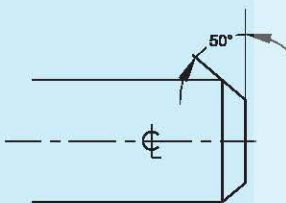
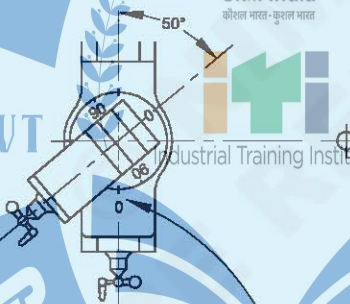
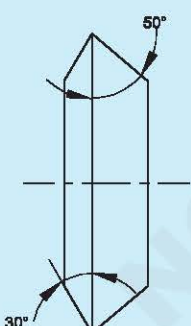
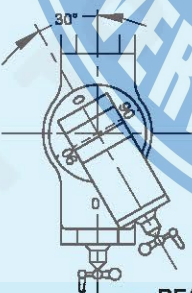

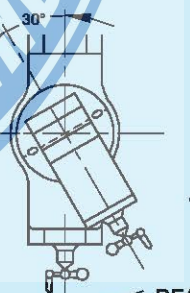
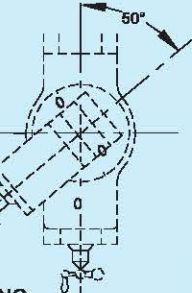
Give successive cuts by the cross-slide and feed the top slide each time.

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 setup for turning various angles

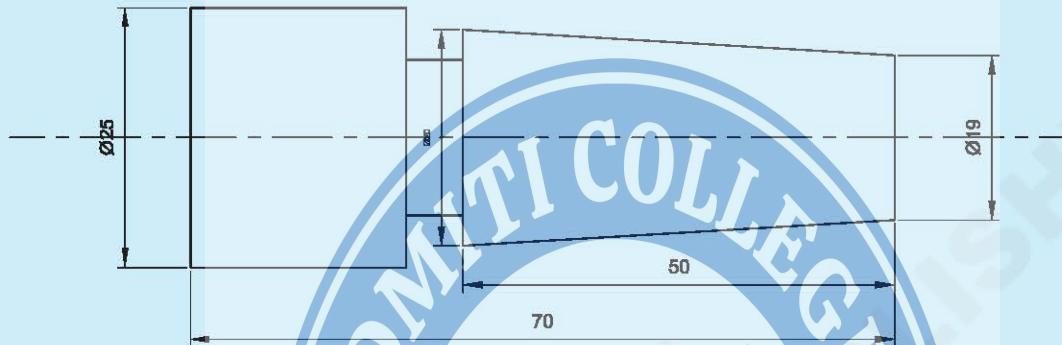
COMPOUND REST SET UP FOR TURNING VARIOUS ANGLES	
	ARRANGEMENT OF GRADUATIONS ON SWIVEL SLIDE
SPECIAL ANGULAR SETTING ON COMPOUND REST	<div style="display: flex; justify-content: space-around;"> <div style="text-align: center;">  <p>GRADUATED FROM 90-0-90</p> </div> <div style="text-align: center;">  <p>GRADUATED FROM 0-45-0</p> </div> </div>
EXAMPLES	READINGS ON GRADUATED SWIVEL SLIDE
<p>AXIS OF LATHE</p>  <p>INCLUDED ANGLE MEASURED IN HORIZONTAL PLANE</p>	<div style="display: flex; justify-content: space-around;"> <div style="text-align: center;">  <p>READING ON SCALE 60°</p> </div> <div style="text-align: center;">  <p>READING ON SCALE 30°</p> </div> </div>
 <p>ANGLE GIVEN FROM A LINE AT 90° TO AXIS OF LATHE</p>	 <p>READING ON SCALE 50°</p>
	<div style="display: flex; justify-content: space-around;"> <div style="text-align: center;">  <p>READING 30°</p> </div> <div style="text-align: center;">  <p>READING 50°</p> </div> <div style="text-align: center;">  <p>READING 60°</p> </div> <div style="text-align: center;">  <p>READING 40°</p> </div> </div>

FIN23104J3

Turn taper pins

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

- set the job on a four jaw chuck
- set the tool in the tool post
- set the taper turning attachment to the required angle
- turn the job in diameter 1:50 taper ratio.



Job Sequence

- Check the raw material size.
- Set the job on a four jaw chuck.
- True the job
- Turn the job Ø 20 mm up to the length of 55 mm
- Calculate the compound rest setting angle the 1:50 taper.
 - Set the angle in compounds slide
 - Turn the diameter taper ration of 1:50
 - Check the diameter of both end as Ø 20 and Ø 19
 - Set the parting tool
 - Feed the cut and remove the length of 50 mm.



$$\frac{\text{opposite side}}{\text{adjacent side}} = \text{Tan } \phi$$

$$\frac{1}{50} = \text{Tan } \phi$$

$$0.02 = \text{Tan } \phi$$

$$\text{Tan}^{-1} .002 = 1.14^\circ$$

convert 0.14 degrees= minute

$$1^\circ = 60'$$

$$0.14 = x$$

$$x = \frac{0.14 \times 60}{1} = 8.4'$$

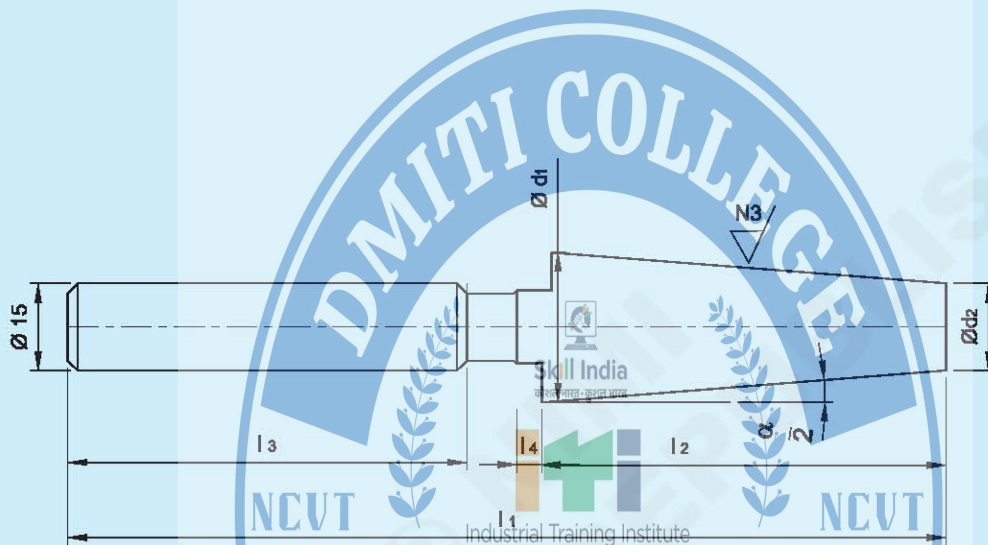
$$\text{setting angle} = 1^\circ, 8'$$

1	Ø25 - 75	-	Fe 310	-	-	1.7.103
NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.
SCALE 1:1		TURN TAPER PINS			TOLERANCE : ± 0.04mm	TIME :
					CODE NO. F120N17103E1	

Turn standard tapers to suit with gauge

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

- set the job on a four jaw chuck
- set taper turning attachment to turn taper
- set the tool in the tool post
- turn standard taper MT3
- check the taper with gauge.



DESIGNATION OF TAPER	d_1 js5	d_2	l_1	l_2 js8	l_3	l_4	Z ± 0.05	$\frac{\alpha}{2}$	AT_D μm
MT3	23.825	17.5	176	81	80	5	1.0	$1^{\circ}26'16''$	+5.1

CONE ANGLE TOLERANCE (AT_D) IS AT_4 GRADE OVER LENGTH ' l_2 ' AS PER IS 7615-1975 SYSTEM OF CORE TOLERANCE

1	$\varnothing 25 - 180$	-	Fe 310	-	-	1.7.104
NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.
SCALE : NTS	TURN STANDARD TAPERS TO SUIT WITH GAUGE				TOLERANCE $\pm 0.04mm$	TIME :
					CODE NO. F120N17104E1	

Job Sequence

- Set the job in four jaw chuck projecting $[(l_1 - l_2 + 10 \text{ mm})]$ outside the chuck.
- True it by universal surface gauge.
- Set the carbide tip tool to the correct centre height for facing with offset facing tool.
- Set offset side cutting tool for turning.
- Set the spindle speed as per the cutting speed chart.
- Face the one end.
- Turn dia 15mm for a length equal to $(l_1 - l_2)$.
- Form grooving, after leaving l_3 from the end and maintain dia.
- Chamfer the two ends of $\varnothing 15$ to $1 \times 45^\circ$.
- Reverse the job and hold turned dia 15mm by giving aluminium/copper sheet as a packing.
- True the job by using surface gauge.
- Face the end to maintain a length of l_1 .
- Turn dia d_1 and check by using vernier micrometer.
- Set the taper turning attachment to turn a taper of $1^\circ 26' 16''$.
- Turn taper MT3 and check the dimensions as per the drawing by using vernier micrometer and vernier bevel protractor.
- Check the taper with gauge.

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.

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.

Lock the clamping bracket to the lathe bed to secure the taper turning attachment in this position.

When using a plain taper turning attachment, follow the steps given below at this stage.

Adjust the top slide so that it is parallel with the cross-slide, i.e. at 90° to the work.

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 work surface.

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

Use the blinding 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 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 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 of about 2 mm long and check the end 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 a light cut and recheck the taper.

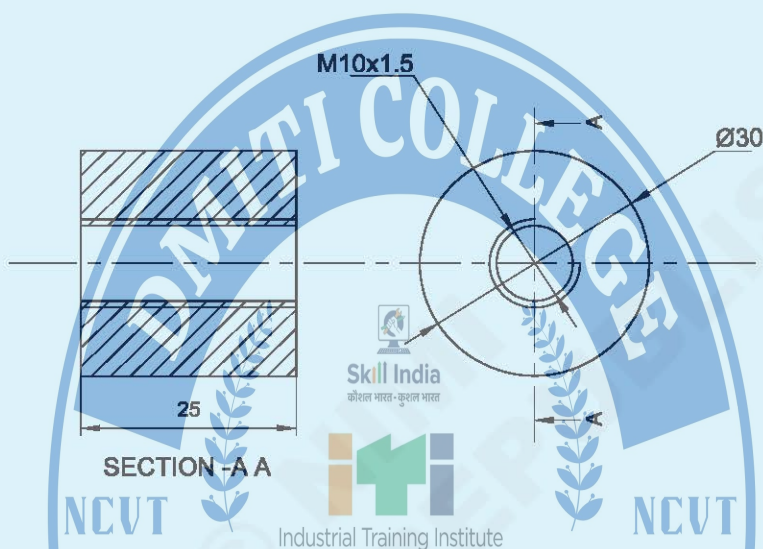
Finish the taper to size and fit it to the taper gauge.

Practice threading using taps, dies on lathe by hand

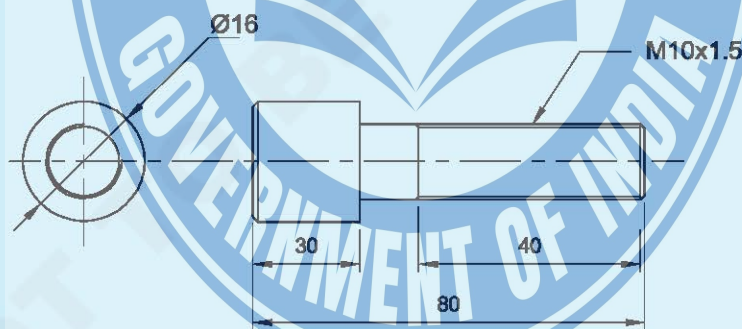
Objectives: At the end of this exercise you shall be able to

- set the job in a three jaw chuck
- drill through hole
- cut internal thread in a lathe using tap and tap wrench
- set the pre machined round rod with three jaw chuck
- cut external thread in a lathe using die and die stock.

TASK 1



TASK 2



1	Ø16 - 85	-	PRE-MACHINED ROUND ROD	-	TASK 2	
1	Ø30 - 30	-	PRE-MACHINED ROUND ROD	-	TASK 1	1.7.105
NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.
SCALE 1:1		PRACTICE THREADING USING TAPS,DIES ON LATHE BY HAND			TOLERANCE : ± 0.04mm	TIME :
					CODE NO. F120N17105E1	

Job Sequence

TASK 1:

- Check the raw material size.
- Hold the job in a 3 jaw chuck
- Turn and finish out side dia meter and length
- Centre drill and drill \varnothing 8.5 mm for M10
- Chamfer the drilled hole on both sides.
- Fix the tap wrench to the square end of first tap
- Place the first tap taper lead in hole and support other end with tail stock dead centre.
- Form thread by first tap, second tap and third tap one by one by hand rotate clock wise slowly and half rotation to release chips till you get the full formation of internal thread.
- Apply oil and clean burrs
- Check the thread hole by M10 bolt.

TASK 2:

- Check the raw material for its size.
- Hold the job in a 3 jaw chuck
- Turn the job for blank size of \varnothing 9.85 mm to 50 mm length
- Chamfer the end of the job.
- Hold the die parallel to job face.
- Rotate the die for a thread forward and for half thread backward with appropriate push to cut thread and remove chips.
- Increase the depth of cut gradually and cut thread to match M10 nut by adjusting the screws provided in the die stock.
- Check the thread with the matching round nut (Task 1).
- Clean the threads without burrs.
- Apply a little oil and preserve it for evaluation.

Note: The tap wrench and die stock handle must be short enough to ensure to rotate on lathe bed.

Skill sequence

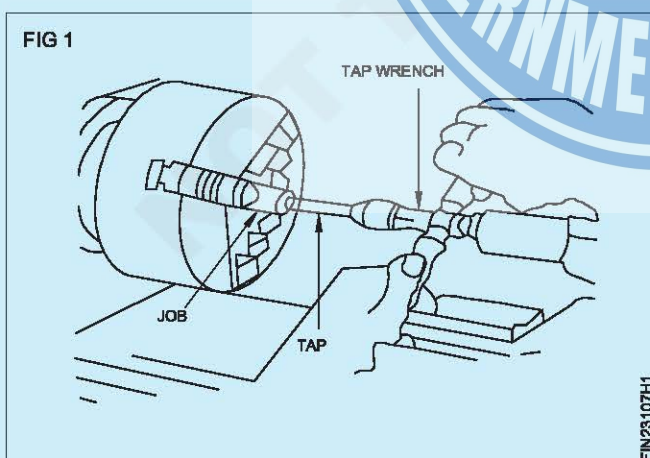
Cutting internal and external thread using

Objective: This shall help you to

- **cut internal and external thread in lathe using tap and die.**

TASK 1:

Cutting internal thread using tap and tap wrench in lathe. (Fig 1)



TASK 2:

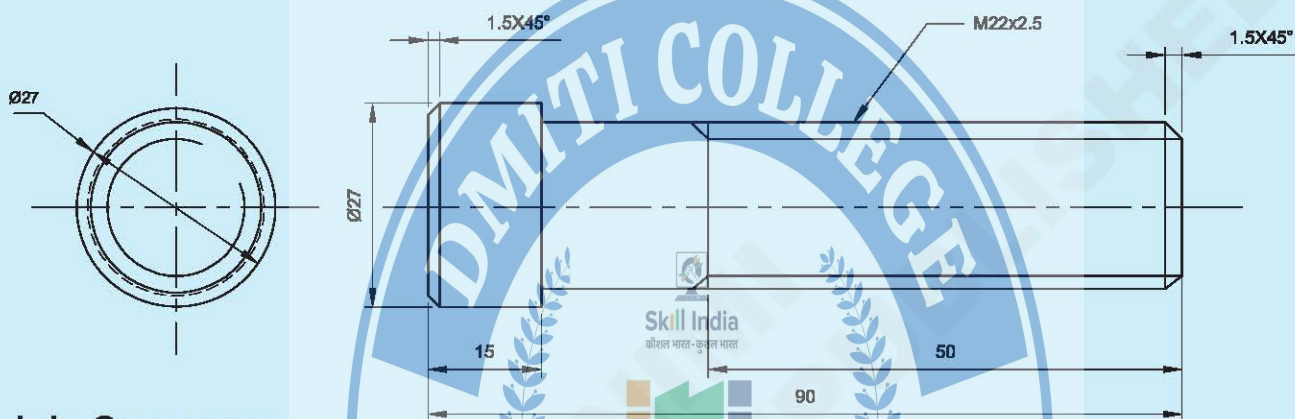
Cutting external thread using die and die stock in lathe. (Fig 2)



Make external 'V' thread

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

- hold the job in lathe machine
- turn and chamfer as per drawing
- grind threading tool to cut metric thread on lathe
- cut metric thread on lathe by single point tool
- check the metric thread using thread ring gauge.



Job Sequence

- Check the raw material size.
- Hold the job in the chuck with 40 mm overhang and true it.
- Face end and turn to Ø 27 mm to maximum length possible.
- Chamfer 1.5×45° at the end.
- Reverse and hold the job in the chuck with 75 mm overhang, face and centre drill.
- Chamfer 1.5×45° at the end.
- Turn the job to Ø 22 mm to length of 75 mm.
- Chamfer 1 x 45° at the end.
- Set the metric 'V' threading tool in the tool post and with the help of centre gauge, set threading tool perpendicular to the axis.
- Set the machine for 2.5 mm pitch to cut right hand thread.
- Set across slide graduation collar to size.
- Move the tail stock with revolving centre close to the job and support the job in centre drilled portion
- Cut right hand metric 'V' thread, giving depth of cut by the cross slide for successive cuts.
- Withdraw the tool at the end of each cut by the cross slide. Again advance to zero before giving depth of cut by the cross slide.
- Rough and finish the thread and check with a thread ring gauge.

1	Ø30 - 100	-	Fe 310	-	-	1.7.106
NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.
SCALE 1:1					TOLERANCE : ± 0.04mm	
<p style="text-align: center;">MAKE EXTERNAL 'V' THREAD</p>					TIME :	
					CODE NO. F120N17106E1	

Skill Sequence

Chamfering on lathe

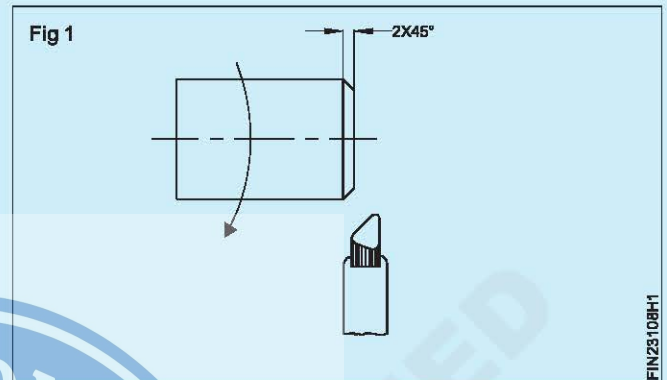
Objective: This shall help you to

- chamfer the end to required size.

Grind the tool to the given angle usually 45° .
 Mount the tool and set centre height properly.
 Set the speed, lock the carriage.
 Move cross slide and plunge the tool to the required size.
 Check the length of chamfer by vernier caliper.

If the protruding length is greater, support with centre.

Make sure the tool is perpendicular to the lathe axis.

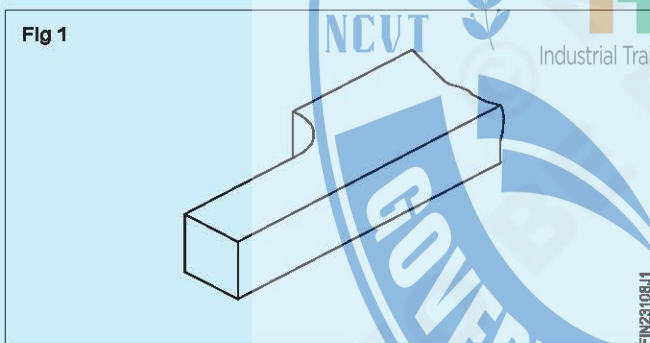


Grinding 60° threading tool

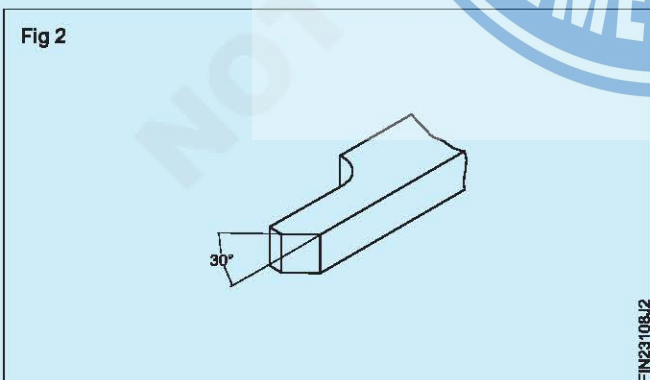
Objective: This shall help you to

- grind 60° threading tool.

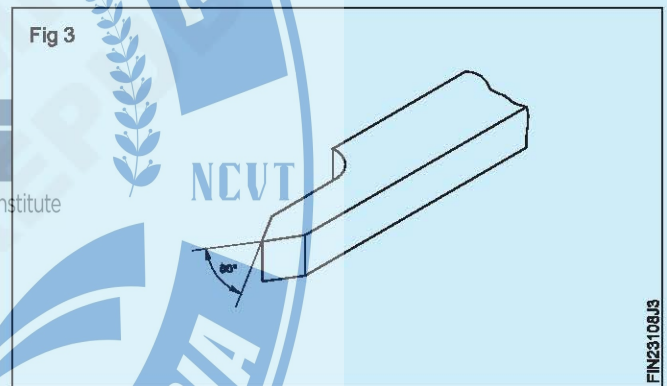
Set the pedestal grinder for tool grinding.
 Remove excess material on right hand side of the tool to length equal to thickness of tool and width being half of the thickness of tool on rough grinding wheel. (Fig 1)



Hold the tool at an angle of 60° to the face of the wheel, grind 30° on left hand side of the tool. (Fig.2)



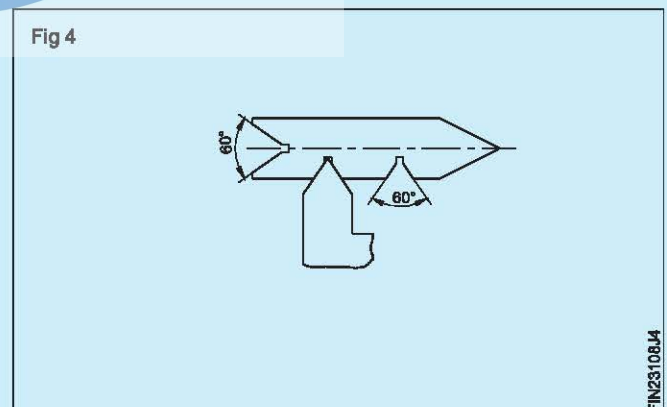
Repeat the above procedure on the right side of the tool to get an included angle of 60° on the tool. (Fig.3)



Grind 6° to 8° side clearance angle on each side of the tool.
 Grind 4° to 6° front clearance angle.
 Finish all slides by using smooth grinding wheel.

Do not Grind Rake Angle

Check the tool by centre gauge, there light should not pass through gauge and cutting edge of the tool. (Fig.4)



Cutting point is curved to $0.14 \times \text{pitch}$ by carefully grinding in smooth wheel.

Finally Lap the tool by applying oil stone on cutting edges.

Safety precautions

Ensure grinding wheels are properly guarded.

Keep 2 mm gap between tool rest and grinding wheel face.

Ensure cutting edge is visible to the operator while grinding.

Do not give too much pressure on the wheel face.

Frequently cool the tool in coolant.

Cutting 'V' thread by plunge cut method

Objective: This shall help you to

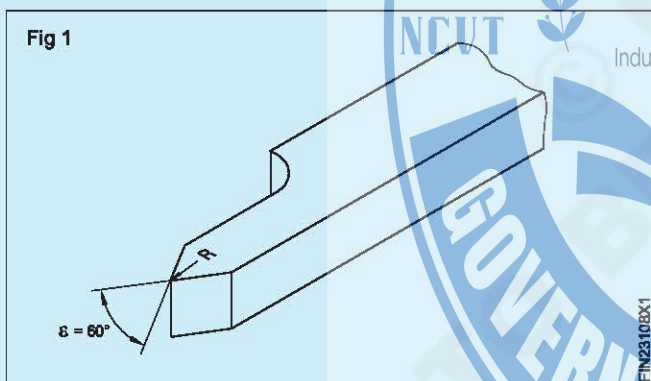
- cut 'V' thread using a single point tool on a lathe by the plunge cut method.

Thread has coarse and fine pitches according to their usage. Standard fine pitch threads, both external and internal, are generally cut by using taps and dies. When they are produced in large quantities, different methods are adopted on different machine tools. However, at times, it may be necessary to cut threads by a single point tool on a centre lathe.

The plunge cut method of threading by a single point tool is done by plunging the tool into the work to produce the thread form. The tip of the tool, as well as, the two flanks of the tool will remove metal during thread cutting and hence the load on the tool will be more. As the possibility of obtaining a good finish on the thread is limited, this method is applicable to fine pitch thread cutting.

The following is the procedural sequence in cutting the 'V' thread by the plunge cut.

Grind a 'V' thread tool for the required thread angle. (Fig 1)



Ensure that the thread angle ground is symmetrical with respect to the axis of the tool.

Arrange the change gear train and set the quick change gearbox levers for the required pitch and hand of thread.

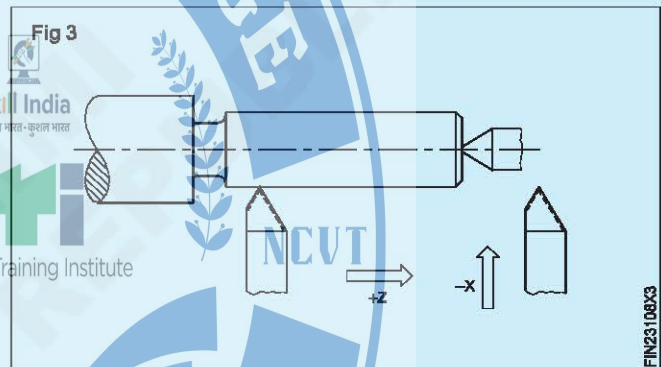
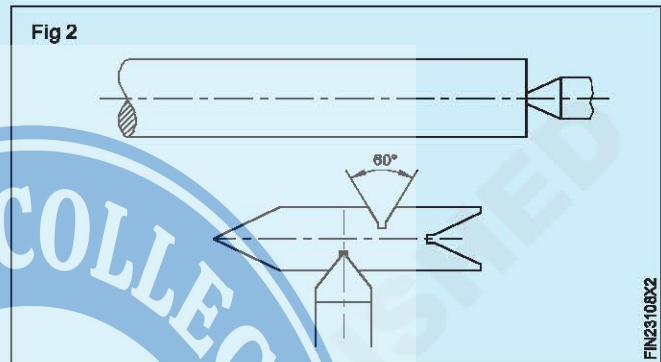
Clamp the tool in the tool-post and set the tool to centre height.

Set the tool perpendicular to the lathe axis by using centre gauge. (Fig 2)

Ensure that the top slide is set at 0° , and slackness is removed by gib adjustment.

Set the machine to about $1/3$ rd of the rough turning r.p.m.

Start the machine and touch the tip to work. (Fig 3) set the cross-slide and the compound slide graduated collars to zero, eliminating backlash.

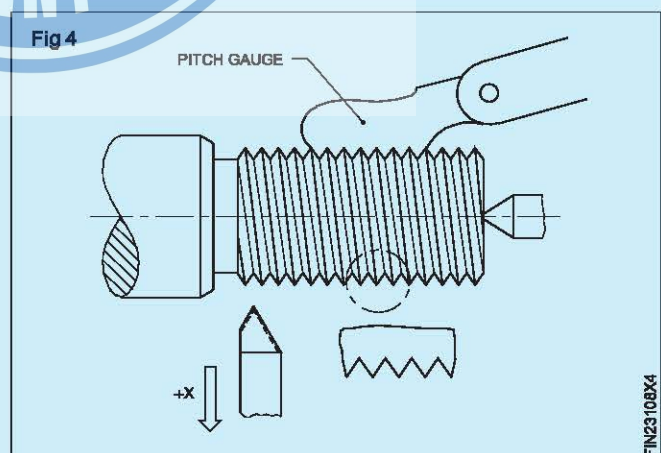


Bring the tool to the starting point and engage the half nut.

Allow the tool to take the trial cut, the depth being given 0.05 mm divisions of the cross-slide graduated collar.

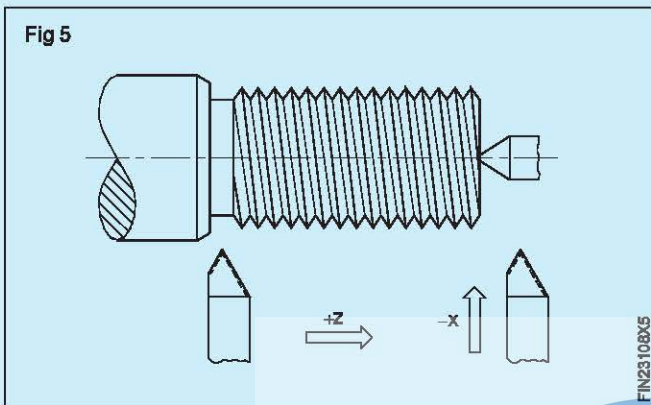
Withdraw the tool at the end of the cut and stop the machine. (Fig 4)

Check with the screw pitch gauge to confirm the gear box setting. (Fig 4)

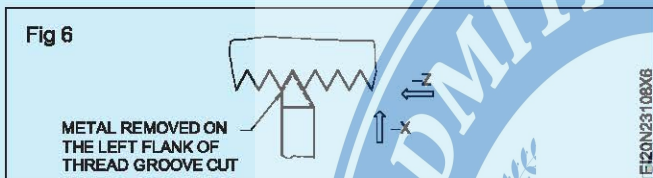


Reverse the machine to bring the carriage to the starting point. (Fig 5)

Give successive cuts.



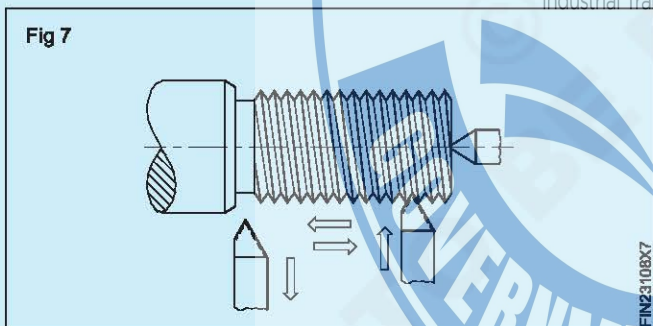
For every 3 depths of cuts by the cross-slide, give one axial cut by feeding the tool axially by half division of the compound slide. This relieves the load on the tool. (Fig 6)



Continue the sequence till the thread profile is formed. (Fig 7)

Check with the screw pitch gauge for the thread form.

Match the mating component to ensure the class of fit.



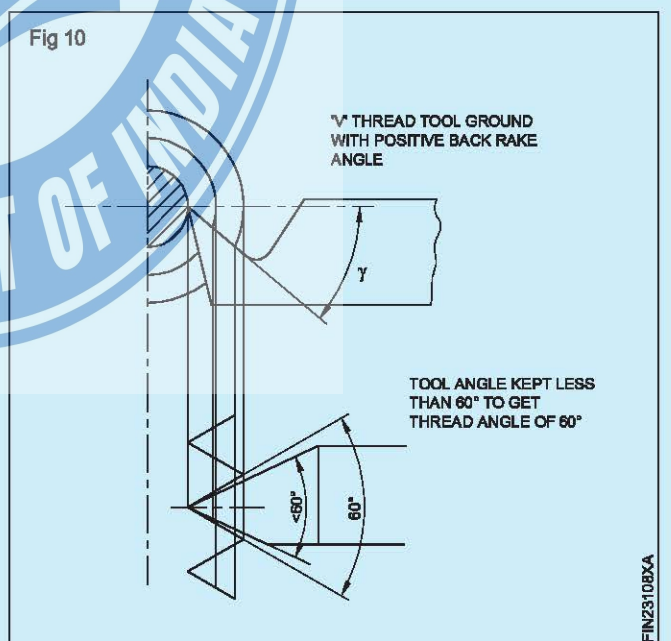
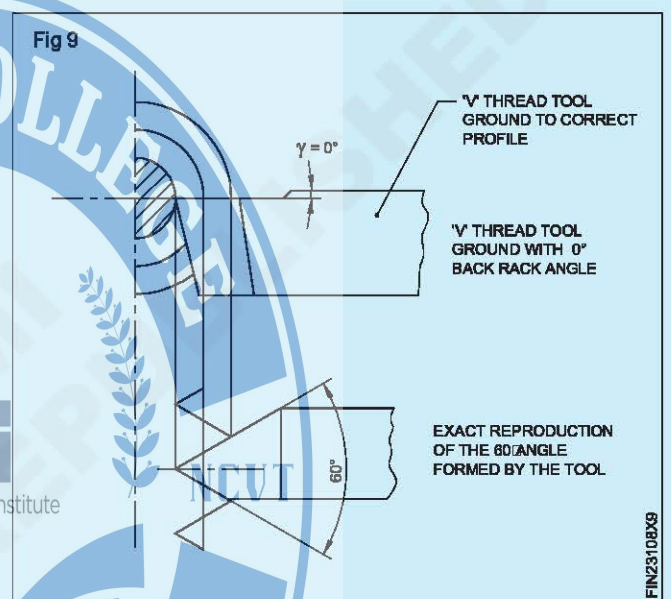
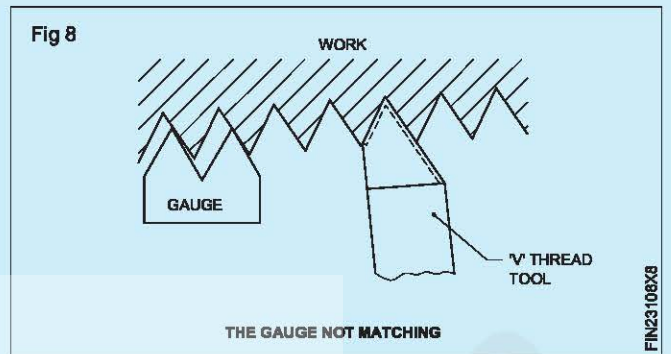
If the tool is not set square to the axis of the work, the gauge will not match with the thread. (Fig 8)

In the plunge cut method of thread cutting with a single point tool on a lathe, the accuracy of the thread is greatly influenced by:

- The correctness of the tool profile.
- The accuracy with which the tool is set square to the axis of the work.
- The number of plunge cuts (depth of cut) given

- The relative number of side cuts (preferably on both flanks) given.

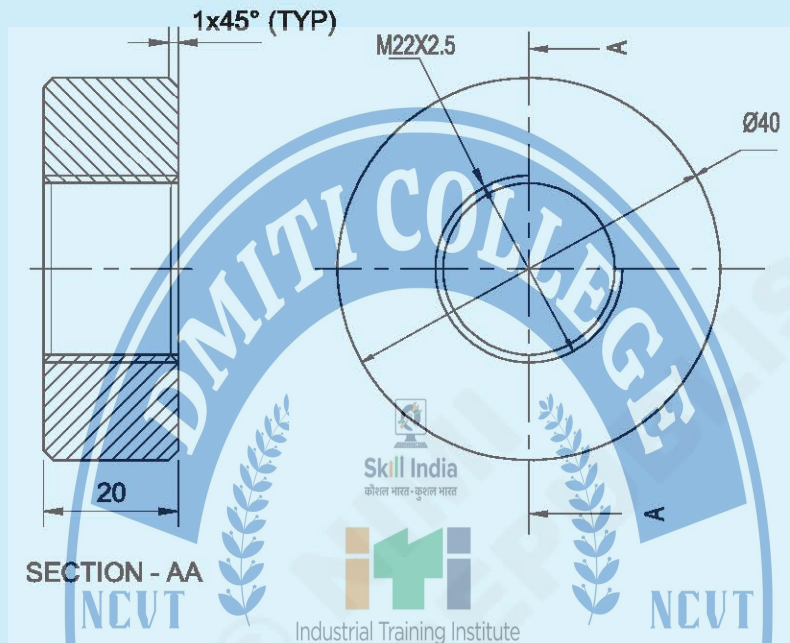
Effect of grinding positive back rake angle of 'V' thread tool and threads cut. (Figs 9 & 10)



Prepare a nut and match with the bolt

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

- cut internal 'V' thread by single point threading tool
- check the metric thread using thread plug gauge
- match the nut and bolt.



Job Sequence

- Check the given material for its size by steel rule.
- Hold the work in a three jaw chuck about 10 mm inside the chuck.
- Turn the outer dia to 40 mm to possible length.
- Chamfer the edge 1x45° by chamfering tool.
- Centre drill, and drill a pilot of Ø 10 mm through hole.
- Enlarge the drilled hole dia 10 mm Ø 18 mm by drilling.
- Bore the drilled hole to the core (root) diameter of the thread i.e. 19.2 mm.
- Set the machine to cut 2.5 mm pitch internal thread.
- Cut the internal thread.
- Check the thread with screw pitch gauge.
- Check the thread with external thread mating parts Ex.106
- Reverse and hold the work on Ø 40 mm and true it.
- Face the end of the work, and maintain a total length of 20 mm.
- Chamfer 1x45° on the outer edge.
- Remove the sharp edges and have a final check.

1	Ø45 - 25	-	Fe310	-	-	1.7.107
NO.OFF	STOCK SIZE	SEMI-PRODUCT	MATERIAL	PROJECT NO.	PART NO.	EX. NO.
SCALE 1:1		PREPARE A NUT AND MATCH WITH THE BOLT			ACCURACY ±0.04mm	TIME:
					CODE NO. FI20N17107E1	

Skill Sequence

Cutting an internal thread

Objective: This shall help you to

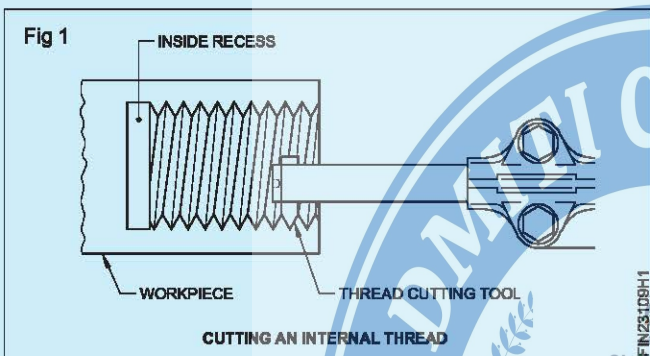
- cut an internal thread on a centre lathe.

Mount the job on four jaw chuck / three jaw chuck/ collet.

Drill and bore the job to the core diameter of the thread to required length/ through hole.

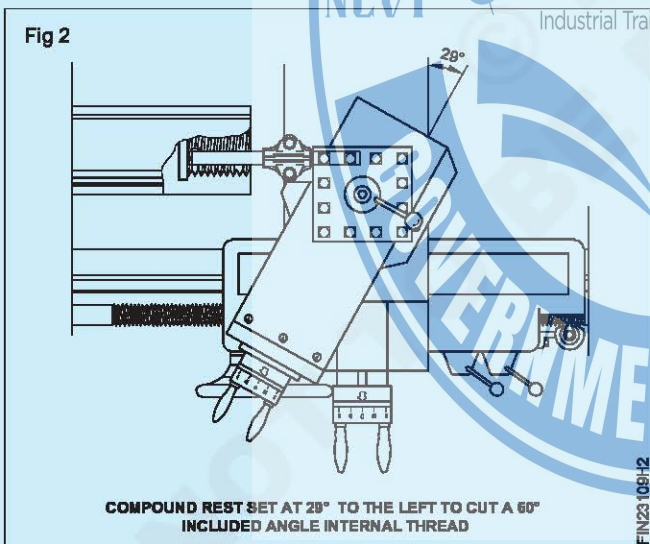
For a blind hole, cut a recess at the end of the bore enough to permit the cutting tool to clear thread.

The recess must be larger than the major diameter of the thread. (Fig 1)



Chamfer the front end to $2 \times 45^\circ$.

Set the compound rest at 29° to cut 60° included angle as shown in Fig 2.

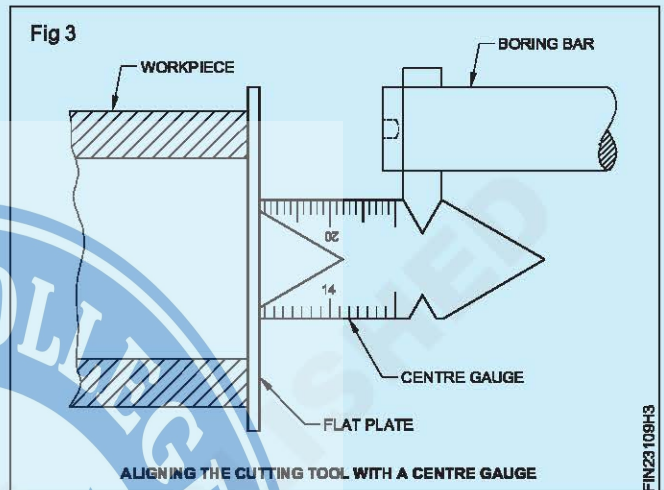


Set the gear box levers to the required pitch.

Fix the correctly ground threading tool in a boring bar.

Fix the boring bar parallel to the lathe centre line and set the point of the cutting tool to lie on the centre.

Align the cutting tool with a help of centre gauge as shown in Fig 3.



Mark the boring bar to indicate the required depth of entry into the bore.

Ensure that the boring bar does not foul anywhere on the job.

Reverse the cross slide until the tool point just touches the bore.

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

Withdraw the cutting tool from the bore.

Set the spindle speed to $1/3$ of the calculated r.p.m.

Start the machine.

Adjust the depth of cut to 0.1 mm.

Engage the half nut.

At the end of the cut, simultaneously reverse the chuck and clear the tool just away from the thread.

Ensure that the tool should not touch the thread in both side of the bore.

When cutting tool comes out of the bore stop the machine.

Give the depth of cut and run the machine in forward direction. Similarly finish the thread until final depth is achieved.

Check the finished thread with a thread plug gauge or a threaded bolt.