METHODIST COLLEGE OF ENGINEERING & TECHNOLOGY

UGC AUTONOMOUS Institution Affiliated to Osmania University, Accredited by NBA & Naac with A+

Estd: 2008

Abids, Hyderabad, Telangana, 500001

DEPARTMENT OF MECHANICAL ENGINEERING

LABORATORY MANUAL

CAD/CAM LABORATORY

BE V Semester

AUTONOMOUS

Name: ....

Roll No: ....

Branch:...SEM:......................

Academic Year: ....

Empower youth- Architects of Future World

Estd: 2008

METHODIST

COLLEGE OF ENGINEERING & TECHNOLOGY

Approved by AICTE New Delhi | Affiliated to Osmania University, Hyderabad

Abids, Hyderabad, Telangana, 500001

VISION

To produce ethical, socially conscious and innovative professionals who would contribute to sustainable technological development of the society.

MISSION

To impart quality engineering education with latest technological developments and interdisciplinary skills to make students succeed in professional practice.

To encourage research culture among faculty and students by establishing state of art laboratories and exposing them to modern industrial and organizational practices.

To inculcate humane qualities like environmental consciousness, leadership, social values, professional ethics and engage in independent and lifelong learning for sustainable contribution to the society.

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DEPARTMENT OF MECHANICAL ENGINEERING

LABORATORY MANUAL

CAD/CAM
(6PC557ME)

Prepared by

Mrs. Shazia Anwar, Assistant Professor. Mech. Engg.

Mr. Shaik Shoeb, CAD Lab Assistant. Mech. Engg.

DEPARTMENT OF MECHANICAL ENGINEERING

VISION

To be a reputed centre of excellence in the field of mechanical engineering by synergizing innovative technologies and research for the progress of society.

MISSION

- To impart quality education by means of state-of-the-art infrastructure.
- To involve in trainings and activities on leadership qualities and social responsibilities.
- To inculcate the habit of life-long learning, practice professional ethics and service the society.
- To establish industry-institute interaction for stake holder development.

DEPARTMENT OF MECHANICAL ENGINEERING

After 3-5 years of graduation, the graduates will be able to:

PEO1: Excel as engineers with technical skills, and work with complex engineering systems.

PEO2: Capable to be entrepreneurs, work on global issues, and contribute to industry and society through service activities and/or professional organizations.

PEO3: Lead and engage diverse teams with effective communication and managerial skills.

PEO4: Develop commitment to pursue life-long learning in the chosen profession and/or progress towards an advanced degree

DEPARTMENT OF MECHANICAL ENGINEERING

PROGRAM OUTCOMES

Engineering Graduates will be able to:

Po1. Engineering knowledge: Apply the basic knowledge of mathematics, science and engineering fund a mentals along with the specialized knowledge of mechanical engineering to understand complex engineering problems.

PO2. Problem analysis: Identify, formulate, design and analyse complex mechanical engineering problems using knowledge of science and engineering.

Po3. Design/development of solutions: Develop solutions for complex engineering problems, design and develop system components or processes that meet the specified needs with appropriate consideration of the public health and safety, and the cultural, societal, and environmental considerations.

PO4. Conduct investigations of complex problems: Formulate engineering problems, conduct investigations and solve using research-based knowledge.

PO5. Modern tool usage: Use the modern engineering skills, techniques and tools that include IT tools necessary for mechanical engineering practice.

Po6.Theengineerandsociety: Apply the contextual knowledge to assess societal, health, safety, legal and cultural issues and the consequent responsibilities relevant to the professional engineering practice.

PO7. Environment and sustainability: Understand the impact of the professional engineering solutions in societal and environmental contexts, and demonstrate the knowledge of, and need for sustainable development.

PO8. Ethics: Apply ethical principles and commit to professional ethics and responsibilities during professional practice.

PO9. Individual and team work: Function effectively as an individual, and as a member or leader in diverse teams, and in multidisciplinary settings.

PO10.Communication: Communicate effectively on complex engineering activities to various groups, ability to write effective reports and make effective presentations.

PO11. Project management and finance: Demonstrate and apply the knowledge to understand the management principles and financial aspects in multidisciplinary environments.

PO12. Life-long learning: Recognize the need for, and have the preparation and ability to engage in Independent and life-long learning in the broadest context of technological change.

PROGRAM SPECIFIC OUTCOMES

Mechanical Engineering Graduates will be able to:

PSO1: Apply the knowledge of CAD/CAM/CAE tools to analyse, design and develop the products and processes related to Mechanical Engineering.

PSO 2: Solve problems related to mechanical systems by applying the principles of modern manufacturing technologies.

PSO 3: Exhibit the knowledge and skill relevant to HVAC and IC Engines.

CODE OF CONDUCT

1. Students should report to the concerned labs as per the time table schedule.
2. Students who turn up late to the labs will in no case be permitted to perform the experiment scheduled for the day.
3. After completion of the experiment, certification of the concerned staff in-charge in the observation book is necessary.
4. Staff member in-charge shall award marks based on continuous evaluation for each experiment out of maximum 15 marks and should be entered in the evaluation sheet/attendance register.
5. Students should bring a note book of about 100 pages and should enter the readings/observationsintothenotebookwhileperformingtheexperiment.
6. The record of observations along with the detailed experimental procedure of the experiment performed in the immediate last session should be submitted and certified by the staff member in-charge.
7. Not more than three students in a group are permitted to perform the experiment on a setup for conventional labs and one student in case of computer labs.
8. The components required pertaining to the experiment should be collected from stores in-charge after duly filling in the requisition form.
9. When the experiment is completed, students should disconnect the setup made by them, and should return all the components/instruments taken for the purpose.
10. Any damage of the equipment or burn-out of components will be viewed seriously either by putting penalty or by dismissing the total group of students from the lab for the semester/year.
11. Students should be present in the labs for the total scheduled duration.
12. Students are required to prepare thoroughly to perform the experiment before coming to Laboratory.

DO'S

1. Leave footwear & bag outside the laboratory at their designated place.
2. Enter the system number in the register & use the system alone.
3. Report any broken plugs, exposed electrical wires or any unsafe conditions to your lecturer/laboratory staff immediately.
4. Read and understand the procedure from Lab Manual as how to carry out an activity thoroughly before coming to the laboratory.
5. Always keep anti-virus in active mode
6. Students must carry their Identity Cards & Observation Notes in the Lab.
7. Enter or Leave the lab only with the permission of the lab in charge.
8. Turn off the respective system and arrange the chairs properly before leaving the laboratory.
9. Training: Ensure that all personnel operating CNC machines have received proper training. This includes understanding machine controls, safety protocols, and basic troubleshooting.
10. Check Tools: Inspect and properly mount cutting tools before starting the machine. Dull or damaged tools can lead to poor quality cuts and potential hazards.
11. Program Review: Double-check the CNC program before running it. Ensure accuracy in code, tool paths, and coordinates to prevent errors.

DON'TS

1. Do not install, uninstall or alter any software on computer.
2. Do not touch electrical fittings nor connect or disconnect any plug or cable.
3. Do not plug in external drives like pen drive, external hard disk or mobile phone
4. Students are not allowed to work in the Lab without the presence of faculty or instructor.
5. Do not leave your place, misbehave or make noise while in the Lab.
6. Don't scatter around unwanted things while doing an experiment.
7. Do not eat or drink in the laboratory.
8. Untrained Operation: Never allow anyone without proper training to operate CNC machines. Inexperienced users can pose risks to themselves and the equipment.
9. Unattended Operation: Never leave CNC machines running unattended, especially for extended periods. Unexpected errors can occur, and leaving the machine running could lead to damage or accidents.
10. Ignore Warning Signs: Pay attention to any unusual sounds, vibrations, or warning messages from the machine. Ignoring these signs could lead to breakdowns or safety hazards.

COURSE OBJECTIVES

The objectives of this course are

1	To learn design criteria of machine components, selection of materials and manufacturing process
2	To familiarize with NC features, part programming using G and M codes, APT, CNC, DNC and FMS etc.

COURSE OUTCOMES

CO No.	Course Outcomes	PO
CO 1	Create the model of the components.	1,2,5,6,7,8,9,10,12
CO 2	Demonstrate the documentation and presentation skills	1,5,7,8,9,10,12
CO 3	Prepare the production drawings of the parts from the given assembly drawing.	1,5,7,8,9,10,12
CO 4	Generate the bill of materials and indicate details pertaining to manufacturing requirements.	1,5,7,8,9,10,12
CO 5	To recognize the importance of Computer Aided Manufacturing and prepare a simple part program to perform machining on a CNC machine and to produce various machine components by performing different machining operations.	1,2,5,6,7,8,9,10,12

COURSE OUTCOMES VS POs MAPPING

S. NO	PO1	PO2	PO3	PO4	PO5	PO6	PO7	PO8	PO9	PO10	PO11	PO12	PSO1	PSO2	PSO3
6PC557ME.1	3.0	2	-	-	3	1	2	2	3	3	-	2	3.0	-	-
6PC557ME.2	3.0	-	-	-	3	-	2	2	3	3	-	2	3.0	-	-
6PC557ME.3	3.0	-	-	-	3	-	2	2	3	3	-	2	3.0	-	-
6PC557ME.4	3.0	-	-	-	3	-	2	2	3	3	-	2	3.0	-	-
6PC557ME.5	3.0	2	-	-	3	1	2	2	3	3	-	2	3.0	1	-
Avg	3.0	2	-	-	3	1	2	2	3	3	-	2	3.0	1	-

LIST OF EXPERIMENTS

List of Additional Experiments

Note: The test is for the ability of the student to read and interpret drawing. The drawing should include part list in standard format.

METHODIST COLLEGE OF ENGINEERING & TECHNOLOGY

(AUTONOMOUS)

Accredited by NAAC with 'A+' and NBA for BE (Civil, CSE, EEE, ECE and Mechanical)

Web: www.methodist.edu.in

Certificate

This is to certify that this is a bonafide record of the work done by

Mr./Ms. ______________________________________ bearing

Roll No. ____________________ of B.E. ______________ ______________
Year Semester

______________________________ in the ______________ Laboratory
Branch

during the Academic year ______________________________

Number of experiments conducted : ______________________

Internal Examiner HOD External Examiner

INDEX

Experiment No	Experiment Name	Date	Page No	Marks				Remarks/ Signature
				P	R	V	T	

Experiment No	Experiment Name	Date	Page No	Marks				Remarks/ Signature
				P	R	V	T	

Introduction- SOLIDWORKS

It is a Parametric CAD Modeling Package developed by Dassault Systems.
It consists of different modules each one meant for specific task as explained below:

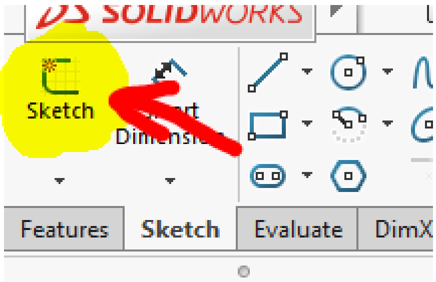
Part Module

It is used to model 3D parts. Each part needs to be modelled & saved in a different file.

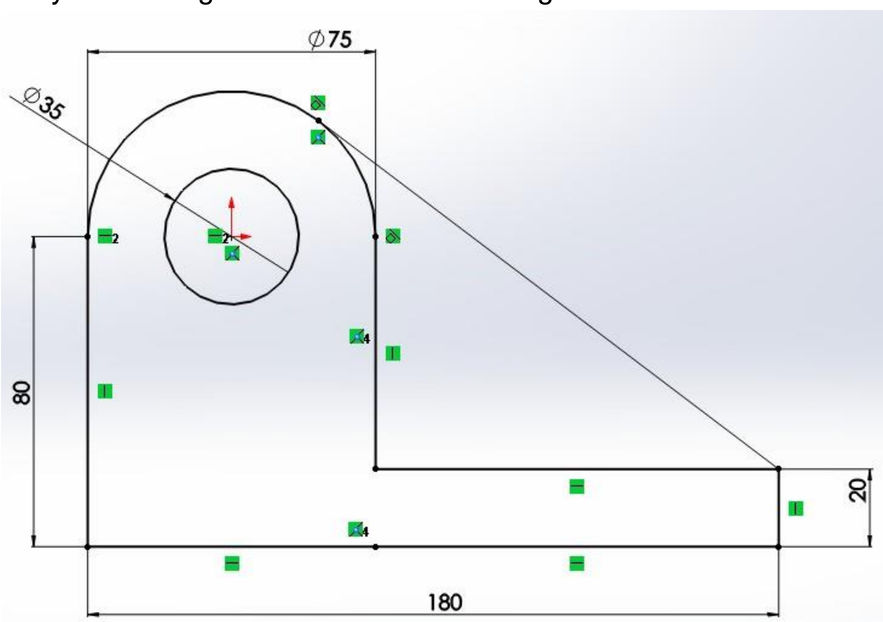
Sketch

It is a feature present within the part module & is used to create 2D profiles which can be then converted to 3D using other features like Extrude, Revolve etc… Steps for using sketch are described below:

1. Click on the sketch button at the left top corner.
2. Select a plane on which you need the sketch.
    a. You may choose the Front, Top or Right reference planes to begin with.
    b. You can also choose the flat faces of a solid as sketch plane.
3. A sketch can be drawn using commands like line, circle, rectangle, polygon, arc etc...



4. Geometric & dimensional constraints can be used to define relations between geometry.
5. A fully constrained sketch will turn black as shown in the below diagram.
6. If any part of the sketch is blue it means that its not completely constrained & can move. It can be dragged by left clicking to check where its moving & corrective action taken.



Extrude Feature

It is used to add solid thickness to a cross section, closed sketch or region.

Assembly Module

It is used to assembly 3D parts by importing the parts saved using Part Module & then applying constraints over them.

Drawing Module

It is used to place projected views of the modelled parts & mentioning dimensions, tolerances, notes, bill of materials, assembly with ballooned numbering etc…

Experiment No 1

Part Modelling-I from given assembly drawings using any solid Modelling package.

Aim:

To create all the components of the Wheel Support assembly and then assemble them, as shown in Figure 1 using dimensions of the components are shown in Figures 2 through 6. Also, draft the 3 view of each component showing all dimensions & Isometric view of the assembly with balloons & Bill of materials table.

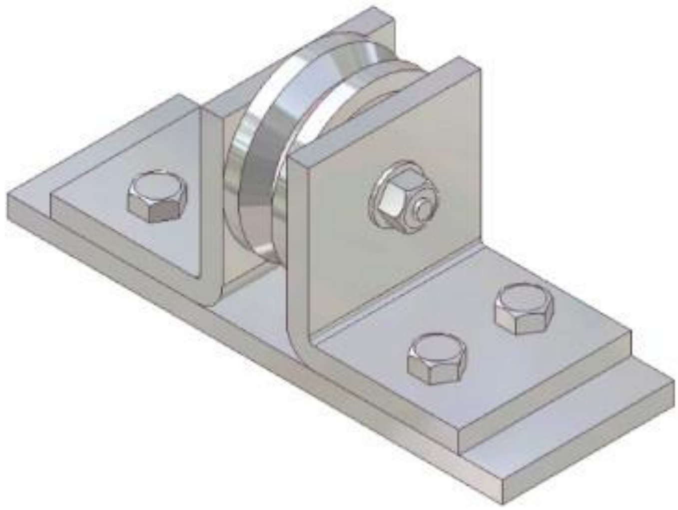


Figure 1 *The Wheel Support assembly*

Software Package Used:

SOLIDWORKS 2020

Hardware Specifications of System:

Processor:__________________________, RAM:__________________, Hard Disk: _________________

Modules used:

Part, Assembly, Drawing

Features used:

Sketch, Extruded Boss/Base, Extruded Cut, Revolved Boss/Base, Fillet
Mate connectors in Assembly: Concentric, Coincidental

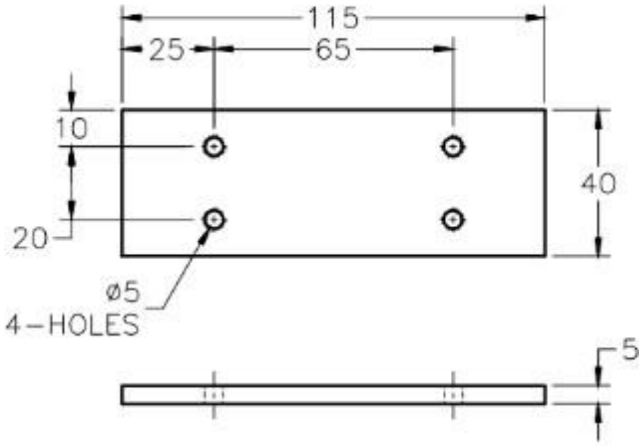


Figure 2 Dimensions of the Base

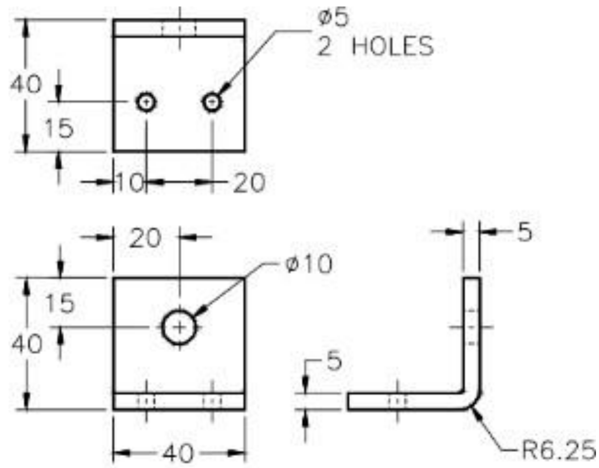


Figure 9-3 Dimensions of the Support

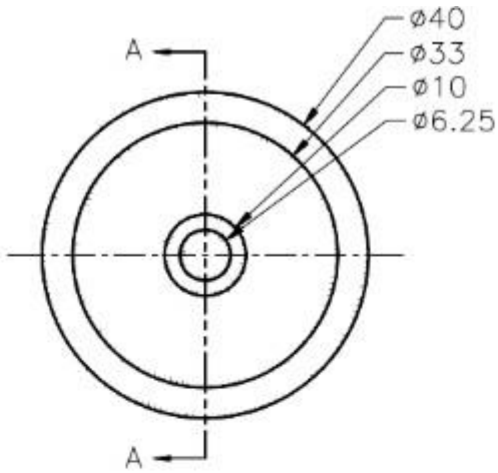


Figure 4 Front view of the Wheel

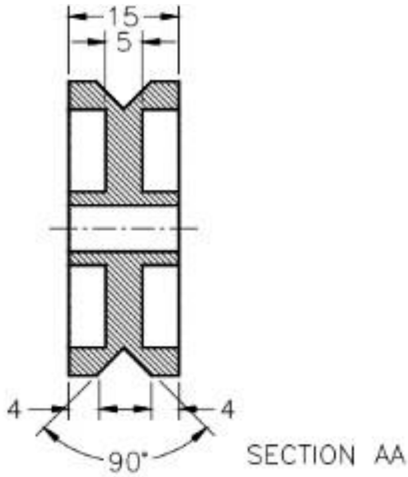


Figure 5 Sectioned side view of the Wheel

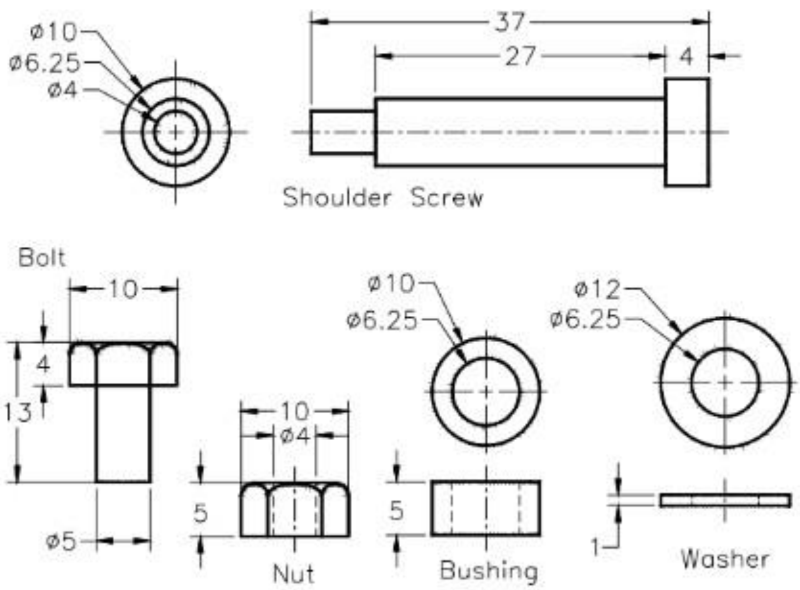


Figure 6 Dimensions of the Shoulder Screw, Bolt, Nut, Bushing, and Washer

Viva Questions

1. What are the different design approaches? Explain their advantages & limitations with examples.
   a. Top Down
   b. Bottom Up
2. What are the modules present in Solidworks? Explain their functions.
   a. Part
   b. Assembly
   c. Drawing
3. What is the function of following features? Note some of their applications.
   a. Sketch
   b. Extruded Boss/Base
   c. Extruded Cut
   d. Revolved Boss/Base
4. What are constraints in a sketch? Explain the steps to add constraints in sketch.
   a. Horizontal
   b. Vertical
   c. Parallel
   d. Perpendicular
   e. Equal
   f. = Curve
   g. Coradial
   h. Fix
   i. Concentric
   j. Tangent
   k. Collinear
   l. Coincide
   m. Merge
5. How can you identify which entities are completely constrained?
6. What are the options in trim? Explain
   a. Power Trim
   b. Corner
   c. Trim away inside
   d. Trim away outside
   e. Trim to closest
7. What are the types of splines / curves available in Solidworks?
8. Explain slot & Polygon features in sketch of Solidworks.
9. What is the use of "Convert to entities" feature in sketch?
10. Explain the transformation tools available in sketch.
    a. Move
    b. Copy
    c. Rotate
    d. Scale
    e. Stretch
11. What is the difference between Mirror in Sketch & Mirror in Pattern?

Experiment No 2

Part Modelling-II from given assembly drawings using any solid Modelling package.

Aim:

To create all the components of the Wheel Support assembly and then assemble them, as shown in Figure 1 using dimensions of the components are shown in Figures 2 through 6. Also, draft the 3 view of each component showing all dimensions & Isometric view of the assembly with balloons & Bill of materials table.

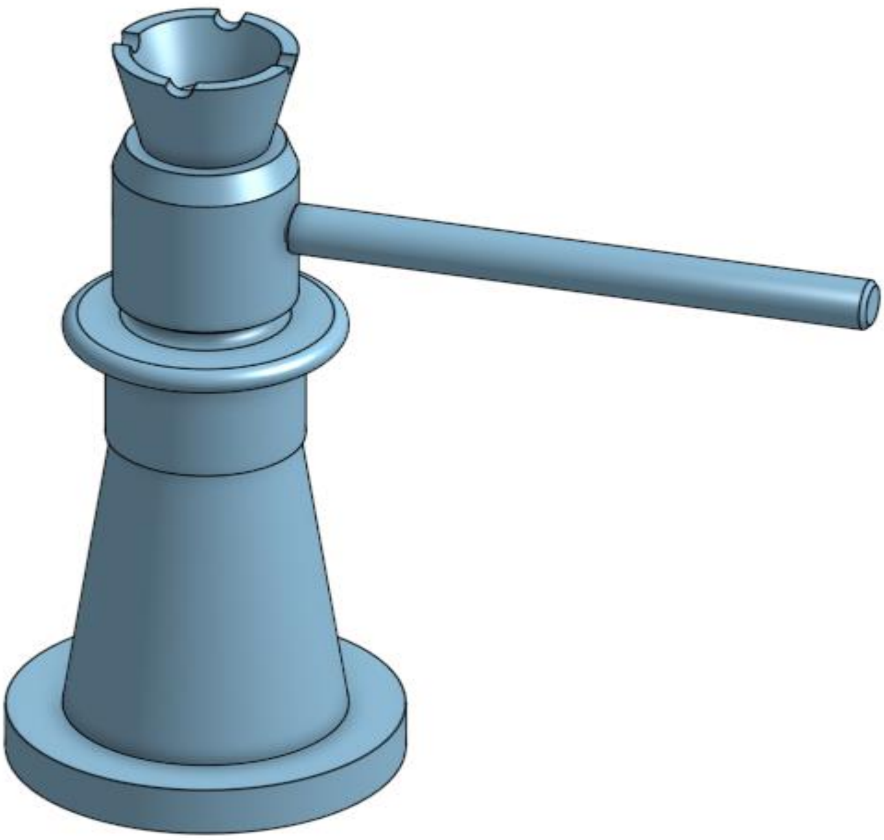


Fig 1 Screw Jack

Software Package Used:

SOLIDWORKS 2020

Hardware Specifications of System:

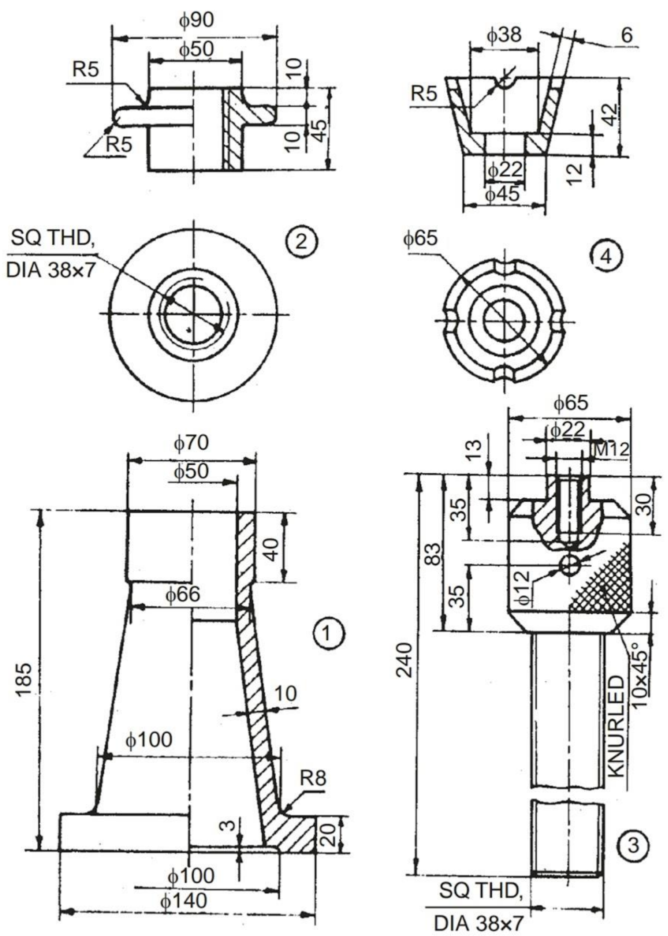
Processor:_________________________, RAM:__________________, Hard Disk: _________________

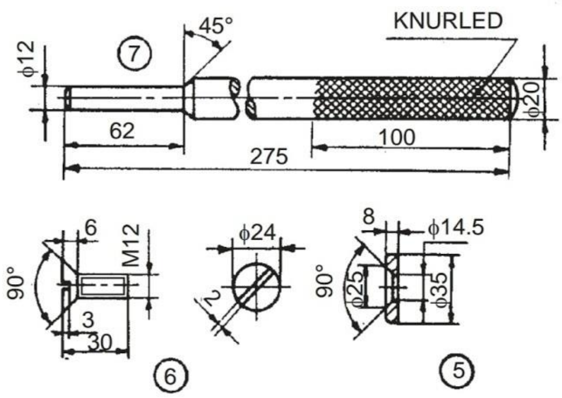
Modules used:

Part, Assembly, Drawing

Features used:

Sketch, Extruded Boss/Base, Extruded Cut, Revolved Boss/Base, Fillet
Mate connectors in Assembly: Concentric, Coincidental





Parts list

Part No.	Name	Matl	Qty
1	Body	CI	1
2	Nut	GM	1
3	Screw	MS	1
4	Cup	CS	1
5	Washer	MS	1
6	Screw	MS	1
7	Tommy bar	MS	1

Fig. 18.51 Scr

Viva Questions:

1. What are the applications of Screw Jack?
2. What constraints are used in the sketches of the parts?
3. What features are used to model each of the parts?
4. Explain reference geometries used in the package.
5. Explain the difference between solid modelling & surface modelling?
6. What are top-down & bottom-up approaches of product development?
7. What is the format used for part files?
8. What are the different formats in which geometry can be exported?

Experiment No 3

Geometric dimensioning and tolerance representation on part drawings.

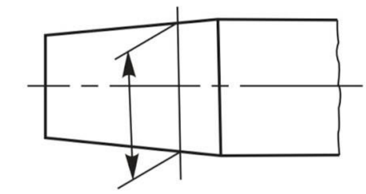


Fig. 2.30

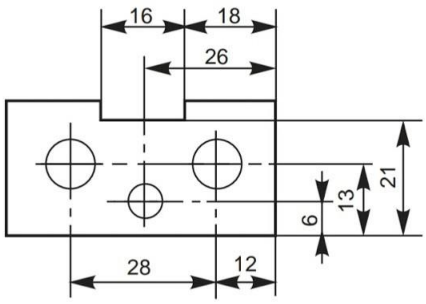


Fig. 2.31

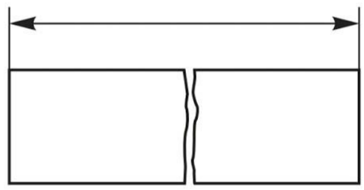


Fig. 2.32

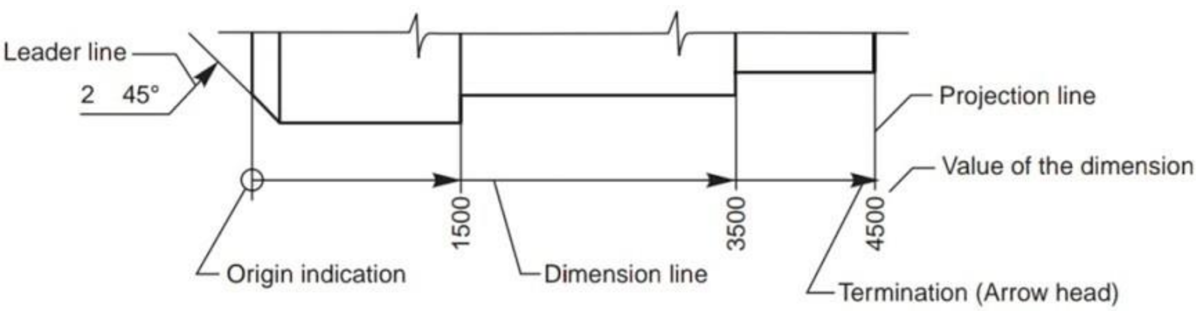
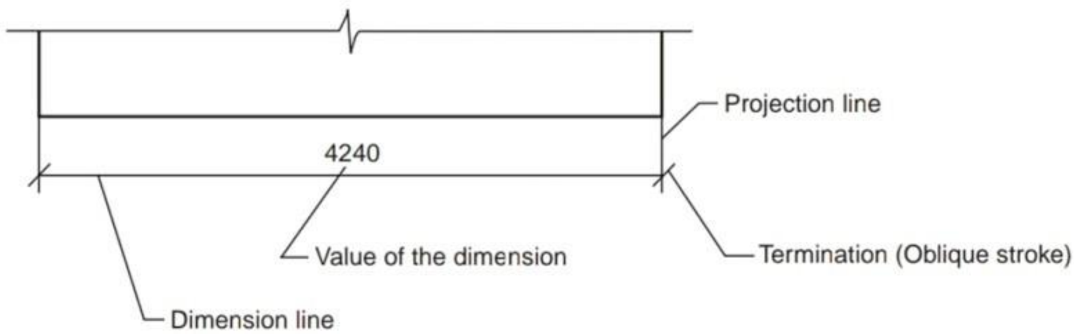


Fig. 2.28 Elements of dimensioning



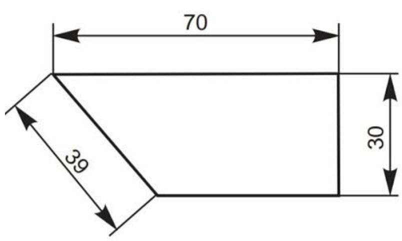


Fig. 2.36

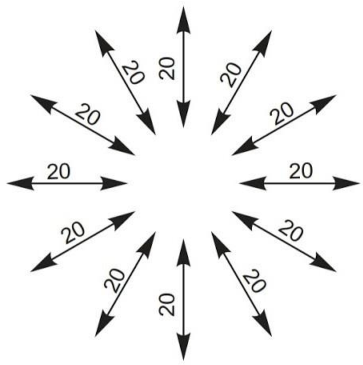


Fig. 2.37 Oblique dimensioning

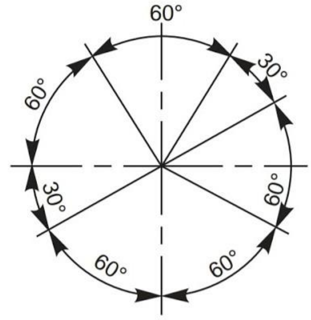
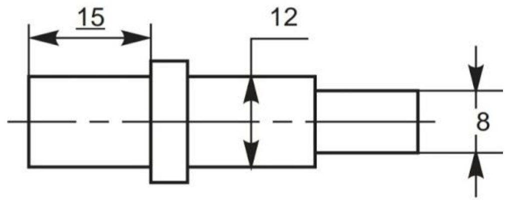
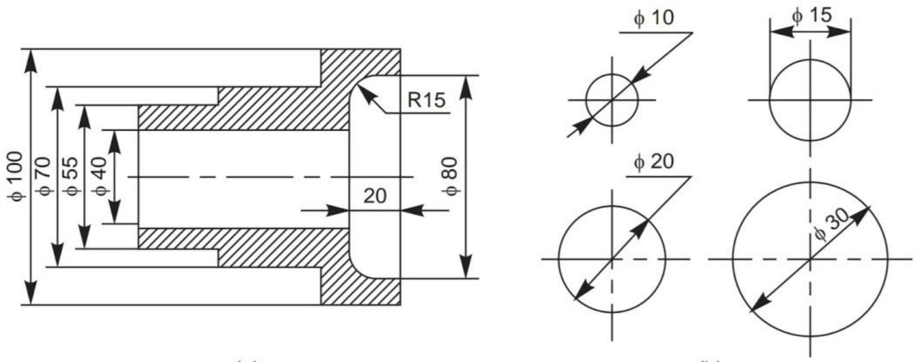
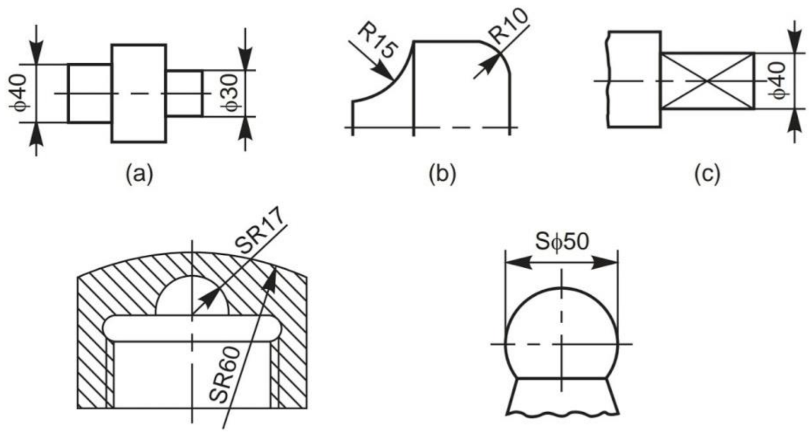


Fig. 2.38 Angular dimensioning





(a) (b)



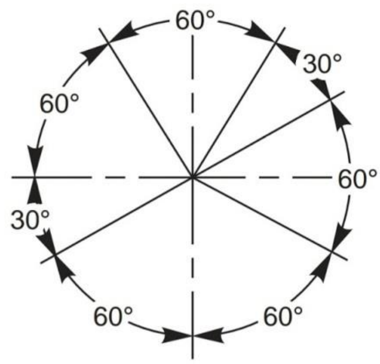
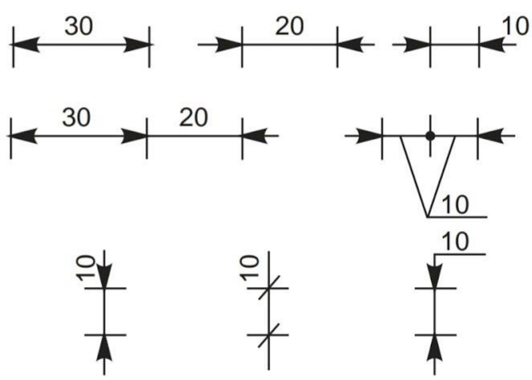
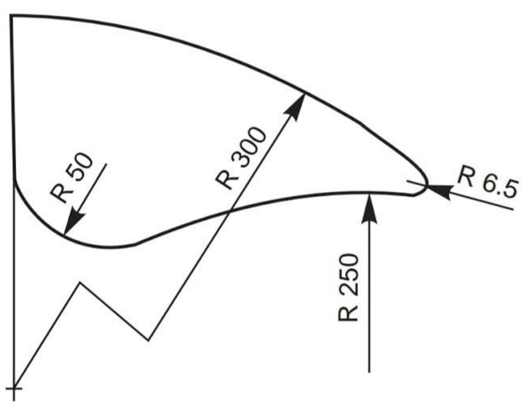


Fig. 2.42 Shape identification symbols





Method of Placing Limit Dimensions (Tolerancing Individual Dimensions)

There are three methods used in industries for placing limit dimensions or tolerancing individual dimensions.

Method 1

In this method, the tolerance dimension is given by its basic value, followed by a symbol, comprising of both a letter and a numeral. The following are the equivalent values of the terms given in Fig. 15.4:

$$\phi\, 25H7 = \phi\, 25^{+0.021}_{+0.000}$$

$$10H10 = 10^{+0.058}_{+0.000}$$

$$40C11 = 40^{+0.280}_{+0.120}$$

$$10h9 = 10^{-0.000}_{-0.036}$$

$$\phi\, 25h9 = \phi\, 25^{-0.000}_{-0.052}$$

$$\phi\, 40h11 = \phi\, 40^{-0.000}_{-0.160}$$

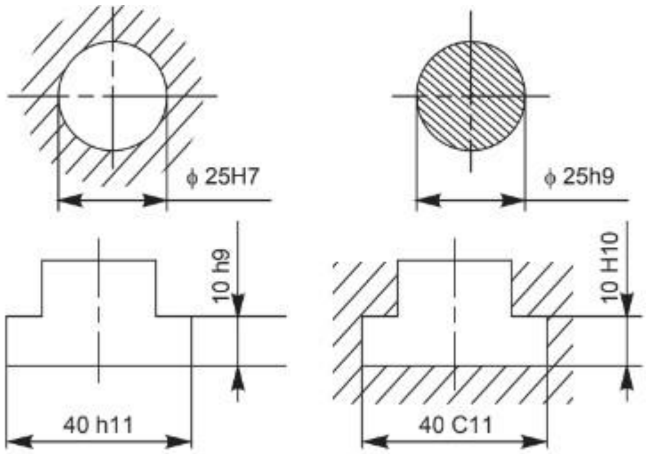


Fig. 15.4 Toleranced dimensions for internal and external features

The terms φ 25H7, 10H10 and 40C11 refer to internal features, since the terms involve capital letter symbols. The capital letter ‘H’ signifies that the lower deviation is zero and the number symbol 7 signifies the grade, the value of which is 21 microns (Table 15.1) which in-turn is equal to the upper deviation. The capital letter C signifies that the lower deviations is 120 microns (Table 15.3). The value of the tolerance, corresponding to grade 11 is 160 microns (Table 15.1). The upper deviation is obtained by adding 160 to 120 which is equal to 280 microns or 0.28 mm.

The terms φ40H11 and 10h9 refer to external features, since the terms involve lower case letters. The letter ‘h’ signifies that the upper deviation is zero (Fig. 15.3) and the number symbol 11 signifies the grade, the value of which is 160 microns (Table 15.1), which in-turn is equal to the lower deviation.

Method 2

In this method, the basic size and the tolerance values are indicated above the dimension line; the tolerance values being in a size smaller than that of the basic size and the lower deviation value being indicated in line with the basic size.

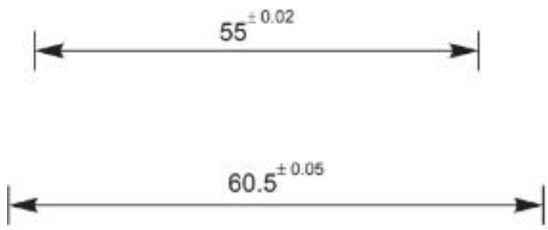


Fig. 15.5 Bilateral tolerance of equal variation

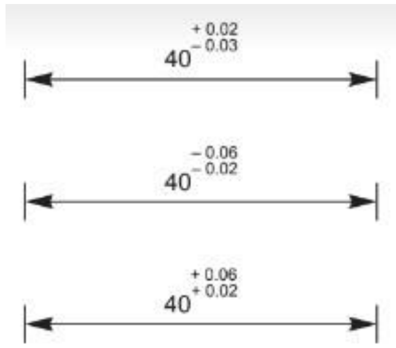


Fig. 15.6 Bilateral tolerance of unequal variation

Figure 15.5 shows dimensioning with a bilateral tolerance; the variation form the basic size being equal on either side.

Figure 15.6 shows dimensioning with a bilateral tolerance; the variation being unequal.

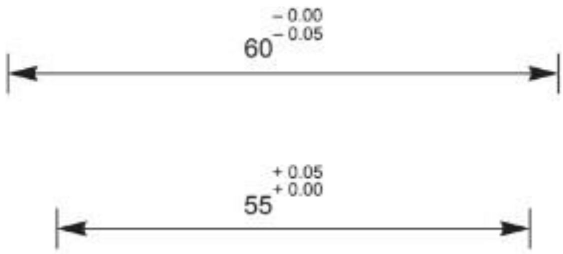


Fig. 15.7 Unilateral tolerance with zero variation in one direction

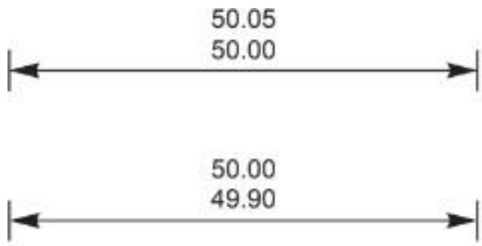


Fig. 15.8 Maximum and minimum size directly indicated

Figure 15.7 shows dimensioning with a unilateral tolerance; the variation being zero in one direction.

Method 3

In this method, the maximum and minimum sizes are directly indicated above the dimension line (Fig. 15.8).

When assembled parts are dimensioned, the fit is indicated by the basic size common to both the components, followed by the hole tolerance symbol first and then by the shaft tolerance symbol (e.g., φ 25 H7/h6, etc., in Fig. 15.9).

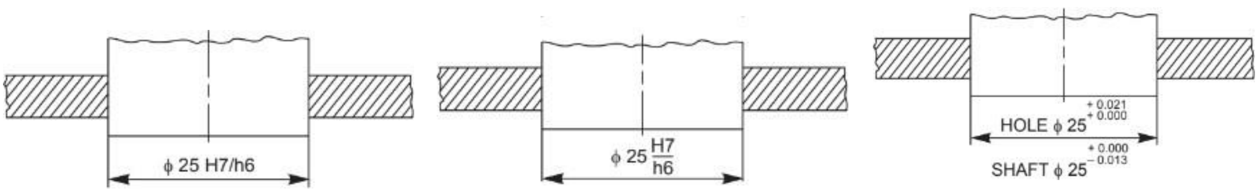


Fig 15.9 Toleranced dimensioning of assembled parts.

Table 15.1A Relative magnitude of IT tolerances for grades 5 to 16 in terms of tolerance unit i for sizes upto 500 mm

Grade	*IT 5*	*IT 6*	*IT 7*	*IT 8*	*IT 9*	*IT 10*	*IT 11*	*IT 12*	*IT 13*	*IT 14*	*IT 15*	*IT 16*
Tolerance values	$7i$	$10i$	$16i$	$25i$	$40i$	$64i$	$100i$	$160i$	$250i$	$400i$	$640i$	$1000i$

Thus, the fundamental tolerance values for different grades (IT) may be obtained either from Table 15.1 or calculated from the relations given in Table 15.1A.

Example 1 *Calculate the fundamental tolerance for a shaft of 100 mm and grade 7.*

The shaft size, 100 lies in the basic step, 80 to 120 mm and the geometrical mean is

$$D = \sqrt{80 \times 120} = 98 \text{ mm}$$

The tolerance unit, $i = 0.45\sqrt[3]{98} + 0.001 \times 98 = 2.172$ microns

For grade 7, as per the Table 15.1A, the value of tolerance is,

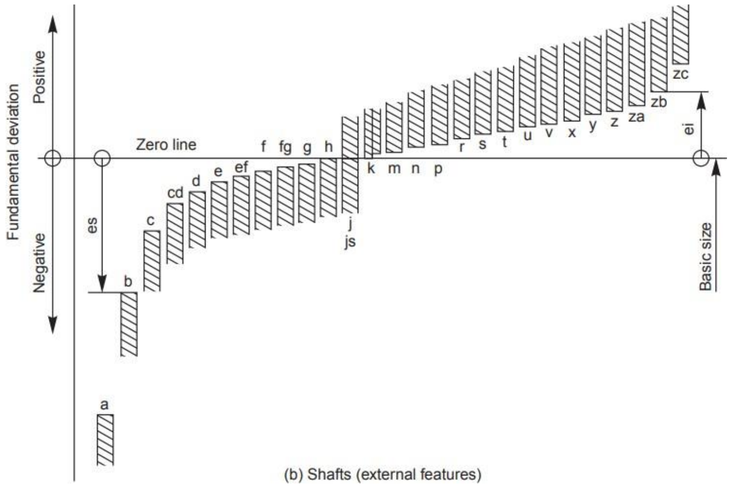
$$16i = 16 \times 2.172 = 35 \text{ microns}$$

(tallies with the value in Table 15.1).

Table 15.1 Fundamental tolerances of grades 01, 0 and 1 to 16 (values of tolerances in microns) (1 micron = 0.001 mm)

Diameter steps in mm	*Tolerance Grades*																	
	01	*0*	*1*	*2*	*3*	*4*	*5*	*6*	*7*	*8*	*9*	*10*	*11*	*12*	*13*	*14**	*15**	*16**
To and inc 3	0.3	0.5	0.8	1.2	2	3	4	6	10	14	25	40	60	100	140	250	400	600
Over 3 To and inc 6	0.4	0.6	1	1.5	2.5	4	5	8	12	18	30	48	75	120	180	300	480	750
Over 6 To and inc 10	0.4	0.6	1	1.5	2.5	4	6	9	15	22	36	58	90	150	220	360	580	900
Over 10 To and inc 18	0.5	0.8	1.2	2	3	5	8	11	18	27	43	70	110	180	270	430	700	1100
Over 18 To and inc 30	0.6	1	1.5	2.5	4	6	9	13	21	33	52	84	130	210	330	520	840	1300
Over 30 To and inc 50	0.6	1	1.5	2.5	4	7	11	16	25	39	62	100	160	250	390	620	1000	1600
Over 50 To and inc 80	0.8	1.2	2	3	5	8	13	19	30	46	74	120	190	300	460	740	1200	1900
Over 80 To and inc 120	1	1.5	2.5	4	6	10	15	22	35	54	87	140	220	350	540	870	1400	2200
Over 120 To and inc 180	1.2	2	3.5	5	8	12	18	25	40	63	100	160	250	400	630	1000	1600	2500
Over 180 To and inc 250	2	3	4.5	7	10	14	20	29	46	72	115	185	290	460	720	1150	1850	2900
Over 250 To and inc 315	2.5	4	6	8	12	16	23	32	52	81	130	210	320	520	810	1300	2100	3200
Over 315 To and inc 400	3	5	7	9	13	18	25	36	57	89	140	230	360	570	890	1400	2300	3600
Over 400 To and inc 500	4	6	8	10	15	20	27	40	63	97	155	250	400	630	970	1550	2500	4000

*Upto 1 mm, Grades 14 to 16 are not provided.



(b) Shafts (external features)

Table 15.2 Fundamental deviations for shafts of types **a** to **k** of sizes upto 500mm *(contd.)*

Fundamental deviation in microns															(1 micron = 0.001 mm)
Diameter steps in mm		*Upper deviation (es)*								js⁺	*Lower deviation (ei)*				
		a	*b*	*c*	*d*	*e*	*f*	*g*	*h*		*j*				*k*
over	*upto*	*All grades*									5.6	7	8	4 to 7	≤ 3, > 7
—	*3	– 270	– 140	– 60	– 20	– 14	– 6	– 2	0		– 2	– 4	– 6	– 0	– 0
3	6	– 270	– 140	– 70	– 30	– 20	– 10	– 4	0		– 2	– 4	—	+ 1	0
6	10	– 280	– 150	– 80	– 40	– 25	– 13	– 5	0		– 2	– 5	—	+ 1	0
10	14	– 290	– 150	– 95	– 50	– 32	– 16	– 6	0	± IT/2	– 3	– 6	—	+ 1	0
14	18														
18	24	– 300	– 160	– 110	– 65	– 40	– 20	– 7	0		– 4	– 8	—	+ 2	0
24	30														
30	40	– 310	– 170	– 120	– 80	– 50	– 25	– 9	0		– 5	– 10	—	+ 2	0
40	50	– 320	– 180	– 130											
50	65	– 340	– 190	– 140	– 100	– 60	– 30	– 10	0		– 7	– 12	—	+ 2	0
65	80	– 360	– 200	– 150											
80	100	– 380	– 220	– 170	– 120	– 72	– 36	– 12	0		– 9	– 15	—	+ 3	0
100	120	– 410	– 240	– 180											
120	140	– 460	– 260	– 200											
140	160	– 520	– 280	– 210	– 145	– 85	– 43	– 14	0		– 11	– 18	—	+ 3	0
160	180	– 580	– 310	– 230											
180	200	– 660	– 340	– 240											
200	225	– 740	– 380	– 260	– 170	– 100	– 50	– 15	0	± IT/2	– 13	– 21	—	+ 4	0
225	250	– 820	– 420	– 280											
250	280	– 920	– 480	– 300	– 190	– 110	– 56	– 17	0		– 16	– 26	—	+4	0
280	315	– 1050	– 540	– 330											
315	355	– 1200	– 600	– 360	– 210	– 125	– 62	– 18	0		– 18	– 28	—	+ 4	0
355	400	– 1350	– 680	– 400											
400	450	– 1500	– 760	– 440	– 230	– 135	– 68	– 20	0		– 20	– 32	—	+ 5	0
450	500	– 1650	– 840	– 480											

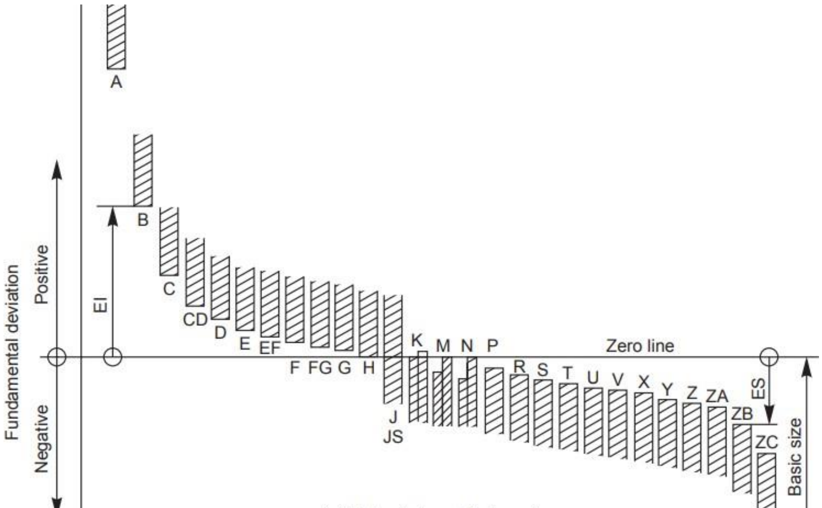
*The deviations of shafts of types a and b are not provided for diameters upto 1 mm

+ For types js in the particular Grades 7 to 11, the two symmetrical deviations ± IT/2 may possibly be rounded, if the IT value in microns is an odd value; by replacing it by the even value immediately below.

Table 15.2 Fundamental deviations for shafts of types **m** to **zc** of sizes upto 500 mm (*contd.*)

Fundamental deviation in microns															(1 micron = 0.001 mm)
Diameter steps in mm		*Lower deviations (ei)*													
		m	*n*	*p*	*r*	*s*	*t*	*u*	*v*	*x*	*y*	*z*	*za*	*zb*	*zc*
Over	*Upto*	*All grades*													
—	3	+ 2	+ 4	+ 6	+ 10	+ 14	—	+ 18	—	+ 20	—	+ 26	+ 32	+ 40	+ 60
3	6	+ 4	+ 8	+ 12	+ 15	+ 19	—	+ 23	—	+ 28	—	+ 35	+ 42	+ 50	+ 80
6	10	+ 6	+ 10	+ 15	+ 19	+ 23	—	+ 28	—	+ 34	—	+ 42	+ 52	+ 67	+ 97
10	14	+ 7	+ 12	+ 18	+ 23	+ 28	—	+ 33	—	+ 40	—	+ 50	+ 64	+ 90	+ 130
14	18								+ 39	+ 45	—	+ 60	+ 77	+ 108	+ 150
18	24	+ 8	+ 15	+ 22	+ 28	+ 35	—	+ 41	+ 47	+ 54	+ 63	+ 73	+ 98	+ 136	+ 188
24	30						+ 41	+ 48	+ 55	+ 64	+ 75	+ 88	+ 118	+ 160	+ 218
30	40	+ 9	+ 17	+ 26	+ 34	+ 43	+ 48	+ 60	+ 68	+ 80	+ 94	+ 112	+ 148	+ 200	+ 274
40	50						+ 54	+ 70	+ 81	+ 97	+ 114	+ 136	+ 180	+ 242	+ 325
50	65	+ 11	+ 20	+ 32	+ 41	+ 53	+ 66	+ 87	+ 102	+ 122	+ 144	+ 172	+ 226	+ 300	+ 405
65	80				+ 43	+ 59	+ 75	+ 102	+ 120	+ 146	+ 174	+ 210	+ 274	+ 360	+ 480
80	100	+ 13	+ 23	+ 37	+ 51	+ 71	+ 91	+ 124	+ 146	+ 178	+ 214	+ 258	+ 335	+ 445	+ 585
100	120				+ 54	+ 79	+ 104	+ 144	+ 172	+ 210	+ 254	+ 310	+ 400	+ 525	+ 690
120	140				+ 63	+ 92	+ 122	+ 170	+ 202	+ 248	+ 300	+ 365	+ 470	+ 620	+ 800
140	160	+ 15	+ 27	+ 43	+ 65	+ 100	+ 134	+ 190	+ 228	+ 280	+ 340	+ 415	+ 535	+ 700	+ 900
160	180				+ 68	+ 108	+ 146	+ 210	+ 252	+ 310	+ 380	+ 465	+ 600	+ 780	+ 1000
180	200				+ 77	+ 122	+ 166	+ 236	+ 274	+ 350	+ 425	+ 520	+ 670	+ 880	+ 1150
200	225	+ 17	+ 31	+ 50	+ 80	+ 130	+ 180	+ 258	+ 310	+ 385	+ 470	+ 575	+ 740	+ 960	+ 1250
225	250				+ 84	+ 140	+ 196	+ 284	+ 340	+ 425	+ 520	+ 640	+ 820	+ 1050	+ 1350
250	280				+ 94	+ 158	+ 218	+ 315	+ 385	+ 475	+ 580	+ 710	+ 920	+ 1200	+ 1550
280	315	+ 20	+ 34	+ 56	+ 98	+ 170	+ 240	+ 350	+ 425	+ 525	+ 650	+ 790	+ 1000	+ 1300	+ 1700
315	355				+ 108	+ 190	+ 268	+ 390	+ 475	+ 590	+ 730	+ 900	+ 1150	+ 1500	+ 1900
355	400	+ 21	+ 37	+ 62	+ 114	+ 208	+ 294	+ 435	+ 530	+ 660	+ 820	+ 1000	+ 1300	+ 1650	+ 2100
400	450				+ 126	+ 232	+ 330	+ 490	+ 595	+ 740	+ 920	+ 1100	+ 1450	+ 1850	+ 2400
450	500	+ 23	+ 40	+ 68	+ 132	+ 252	+ 360	+ 540	+ 660	+ 820	+ 1000	+ 1250	+ 1600	+ 2100	+ 2600

1. Convert the following tolerances into
   a. Second method of representing the tolerances (with relative upper & lower limits in small letters)
   b. the absolute values of upper & lower limits.



(a) Holes (Internal features)

Table 15.3 Fundamental deviations for holes of types **A** to **N** for sizes upto 500 mm *(contd.)*

A to N

Fundamental deviation in microns																				(1 micron = 0.001 mm)
Diameter steps in mm		*Lower deviations (EI)*														*Upper deviations (ES)*				
		*A**	**B*	*C*	*D*	*E*	*F*	*G*	*H*	*Js+*	*J*			*K*		*M*		*N*		
Over	*Upto*	*All grades*									6	7	8	≤ 8	> 8	≤ 8 ‡	> 8	≤ 8	> 8*	≤ 7
—	3*	+ 270	+ 140	+ 60	+ 20	+ 14	+ 6	+ 2	0		+ 2	+ 4	+ 6	0	0	– 2	– 2	– 4	– 4	
3	6	+ 270	+ 140	+ 70	+ 30	+ 20	+ 10	+ 4	0		+ 5	+ 6	+ 10	– 1 + Δ	—	– 4 + Δ	– 4+Δ	– 8 + Δ	0	
6	10	+ 280	+ 150	+ 80	+ 40	+ 25	+ 13	+ 5	0		+ 5	+ 8	+ 12	– 1 + Δ	—	– 6+ Δ	– 6+Δ	– 10 + Δ	0	
10	14	+ 290	+ 150	+ 95	+ 50	+ 32	+ 16	+ 6	0		+ 6	+ 10	+ 15	– 1+ Δ	—	– 7 + Δ	– 7	– 12 + Δ	0	
14	18									± IT/2										
18	24	+ 300	+ 160	+ 110	+ 65	+ 40	+ 20	+ 7	0		+ 8	+ 12	+ 20	– 2 + Δ	—	– 8 + Δ	– 8	– 15 + Δ	0	Same deviation as for grades > 7 + Δ
24	30																			
30	40	+ 310	+ 170	+ 120	+ 80	+ 50	+ 25	+ 9	0		+ 10	+ 14	+ 24	– 2 + Δ	—	– 9 + Δ	– 9	– 17 + Δ	0	
40	50	+ 320	+ 180	+ 130																
50	65	+ 340	+ 190	+ 140	+ 100	+ 60	+ 30	+ 10	0		+ 13	+ 18	+ 28	– 2 + Δ	—	– 11+ Δ	– 11	– 20 + Δ	0	
65	80	+ 360	+ 200	+ 150																
80	100	+ 380	+ 220	+ 170	+ 120	+ 72	+ 36	+ 12	0		+ 16	+ 22	+ 34	– 3 + Δ	—	– 13 + Δ	– 13	– 23 + Δ	0	
100	120	+ 410	+ 240	+ 180																
120	140	+ 460	+ 260	+ 200																
140	160	+ 520	+ 280	+ 210	+ 145	+ 85	+ 43	+ 14	0		+ 18	+ 26	+ 41	– 3 + Δ	—	– 15 + Δ	– 15	– 27 + Δ	0	
160	180	+ 580	+ 310	+ 230																
180	200	+ 660	+ 340	+ 240																
200	225	+ 740	+ 380	+ 260	+ 170	+ 100	+ 50	+ 15	0		+ 22	+ 30	+ 47	– 4+ Δ	—	– 17 + Δ	– 17	– 31 + Δ	0	
225	250	+ 820	+ 420	+ 280																
250	280	+ 920	+ 480	+ 300	+ 190	+ 110	+ 56	+ 17	0	± IT/2	+ 25	+ 36	+ 55	– 4+ Δ	—	– 20 + Δ	– 20	– 34 + Δ	0	Same deviation as for grades > 7 + Δ
280	315	+ 1050	+ 540	+ 330																
315	355	+ 1200	+ 600	+ 360	+ 210	+ 125	+ 62	+ 18	0		+ 29	+ 39	+ 60	– 4+ Δ	—	– 21 + Δ	– 21	– 37 + Δ	0	
355	400	+ 1350	+ 680	+ 400																
400	450	+ 1500	+ 760	+ 440	+ 230	+ 135	+ 68	+ 20	0		+ 33	+ 43	+ 66	– 5+ Δ	—	– 23 + Δ	– 23	– 40 + Δ	0	
450	500	+ 1650	+ 840	+ 480																

* The deviation of holes of types A and B in all grades >8 are not for diameters upto 1 mm.

+For the hole of type Js in the grades 7 and 11, the two symmetrical ± deviations IT/2 may possibly rounded. If the IT value in microns is an odd value, replace it by the even value immediately below.

‡ Special case: For the hole M6, ES = 9 from 250 to 315 (instead of – 11).

Table 15.3 Fundamental deviations for holes of types P to ZC for sizes upto 500mm (*Contd.*)

P to ZC

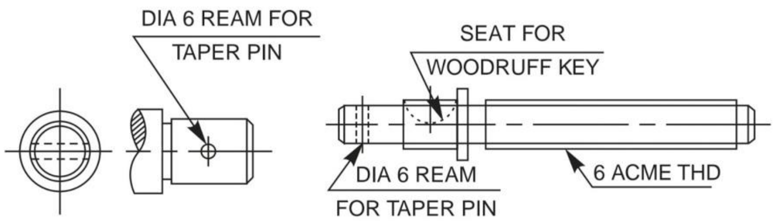
Fundamental deviation in microns														(1 micron = 0.001 mm)					
Diameter steps in mm		*Upper deviations (ES)*																	
		P	*R*	*S*	*T*	*U*	*V*	X	*Y*	Z	ZA	*ZB*	ZC	Δ in microns*					
Over	*Upto*	>7												3	4	5	6	7	8
—	3	– 6	– 10	– 14	—	– 18	—	– 20	—	– 26	– 32	– 40	– 60	Δ = 0					
3	6	– 12	– 15	– 19	—	– 23	—	– 28	—	– 35	– 42	– 50	– 80	1	1.5	1	3	4	6
6	10	– 15	– 19	– 23	—	– 28	—	– 34	—	– 42	– 52	– 67	– 97	1	1.5	2	3	6	7
10	14	– 18	– 23	– 28	—	– 33	—	– 40	—	– 50	– 64	– 90	– 130	1	2	3	3	7	9
14	18						– 39	– 45	—	– 60	– 77	– 109	– 150						
18	24	– 22	– 28	– 35	—	– 41	– 47	– 54	– 63	– 73	– 93	– 136	– 188	1.5	2	3	4	8	12
24	30				– 41	– 48	– 55	– 64	– 75	– 88	– 118	– 160	– 218						
30	40	– 26	– 34	– 43	– 48	– 60	– 68	– 80	– 94	– 112	– 148	– 200	– 274	1.5	3	4	5	9	14
40	50				– 54	– 70	– 81	– 97	– 114	– 136	– 180	– 242	– 325						
50	65	– 32	– 41	– 53	– 65	– 87	– 102	– 122	– 144	– 172	– 226	– 300	– 405	2	3	5	6	11	16
65	80		– 43	– 59	– 75	– 102	– 120	– 146	– 174	– 210	– 274	– 360	– 480						
80	100	– 37	– 51	– 71	– 91	– 124	– 146	– 178	– 214	– 258	– 335	– 445	– 585	2	4	5	7	13	19
100	120		– 54	– 79	– 104	– 144	– 172	– 210	– 254	– 310	– 400	– 525	– 690						
120	140		– 63	– 92	– 122	– 170	– 202	– 248	– 300	– 365	– 470	– 620	– 800	3	4	6	7	15	23
140	160	– 43	– 65	– 100	– 134	– 190	– 228	– 280	– 340	– 415	– 535	– 700	– 900						
160	180		– 68	– 108	– 146	– 210	– 252	– 310	– 380	– 465	– 600	– 780	– 1000						
180	200		– 77	– 122	– 166	– 236	– 284	– 350	– 425	– 520	– 670	– 880	– 1150						
200	225	– 50	– 80	– 130	– 180	– 256	– 310	– 385	– 470	– 575	– 740	– 960	– 1250	3	4	6	9	17	26
225	250		– 84	– 140	– 196	– 284	– 340	– 425	– 520	– 640	– 820	– 1050	– 1350						
250	280	– 56	– 94	– 158	– 218	– 315	– 385	– 475	– 580	– 710	– 920	– 1200	– 1550	4	4	7	9	20	29
280	315		– 98	– 170	– 240	– 350	– 425	– 525	– 650	– 790	– 1000	– 1300	– 1700						
315	355	– 62	– 108	– 190	– 268	– 390	– 475	– 590	– 730	– 900	– 1150	– 1500	– 1900	4	5	7	11	21	32
355	400		– 114	– 208	– 294	– 435	– 530	– 650	– 820	– 1000	– 1300	– 1650	– 2100						
400	450	– 68	– 126	– 232	– 330	– 490	– 595	– 740	– 920	– 1100	– 1450	– 1850	– 2400	5	5	7	13	23	34
450	500		– 132	– 252	– 360	– 540	– 660	– 820	– 1000	– 1250	– 1600	– 2100	– 2600						

*In determining K, M, N upto grade 8 and P to ZC upto grade 7, take the Δ values from the columns on the right.
Example: For P7, from diameters 18 to 30 mm, Δ = 8; hence ES = – 14.

Methods of indicating notes on drawing

Term	Abbreviations	Term	Abbreviations	Material	Abbreviation
Across corners	A/C	Manufacture	MFG	Aluminium	AL
Across flats	A/F	Material	MATL	Brass	BRASS
Approved	APPD	Maximum	max.	Bronze	BRONZE
Approximate	APPROX	Meter	m	Cast iron	CI
Assembly	ASSY	Mechanical	MECH	Cast steel	CS
Auxiliary	AUX	Millimeter	mm	Chromium steel	CrS
Bearing	BRG	Minimum	min.	Copper	Cu
Centimeter	Cm	Nominal	NOM	Forged steel	FS
Centers	CRS	Not to scale	NTS	Galvanized iron	GI
Centre line	CL	Number	No.	Gray iron	FG
Centre to centre	C/L	Opposite	OPP	Gunmetal	GM
Chamfered	CHMED	Outside diameter	OD	High carbon steel	HCS
Checked	CHD	Pitch circle	PC	High speed steel	HSS
Cheese head	CH HD	Pitch circle dia	PCD	High tensile steel	HTS
Circular pitch	CP	Quantity	QTY	Low carbon steel	LCS
Circumference	OCE	Radius	R	Mild steel	MS
Continued	CONTD	Radius in a note	RAD	Nickel steel	Ni S
Counter bore	C BORE	Reference	REF	Pearlitic malleable iron	PM
Countersunk	CSK	Required	REQD	Phosphor bronze	PHOS.B
Cylinder	CYL	Right hand	RH	Sheet steel	Sh S
Diameter	DIA	Round	RD	Spring steel Spring	S
Diametric pitch	DP	Screw	SCR	Structure steel	St
Dimension	DIM	Serial number	Sl. No.	Tungsten carbide steel	TCS
Drawing	DRG	Specification	SPEC	Wrought iron	WI
Equii-spaced	EQUI-SP	Sphere/Spherical	SPHERE	White metal	WM
External	EXT	Spot face	SF		
Figure	FIG.	Square	SQ		
General	GNL	Standard	STD		
Ground level	GL	Symmetrical	SYM		
Ground	GND	Thick	THK		
Hexagonal	HEX	Thread	THD		
Inspection	INSP	Through	THRU		
Inside diameter	ID	Tolerance	TOL		
Internal	INT	Typical	TYP		
Left hand	LH	Undercut	U/C		
Machine	M/C	Weight	WT		

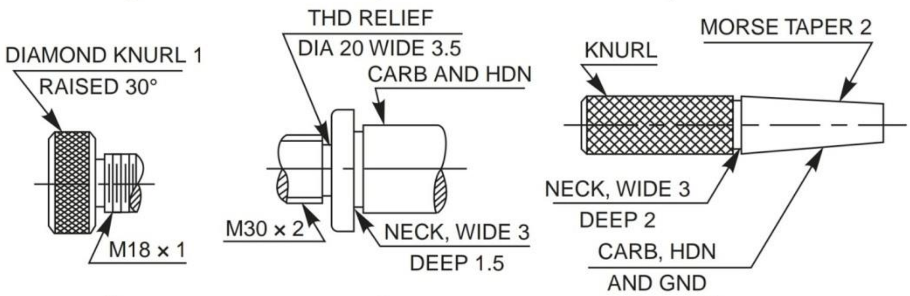
S.No.	*Note*	*Meaning / Instruction*
1.	DIA 25 DEEP 25	Drill a hole of diameter 25 mm, to a depth of 25 mm.
2.	DIA 10 CSK DIA 15	Drill a through hole of diameter 10 mm and countersink to get 15 mm on top.
3.	4 HOLES, DIA 12 C BORE DIA 15 DEEP 8	Dirll through hole of ϕ 12 mm, counterbore to a depth of 8 mm, with a ϕ 15 mm, the number of such holes being four.
4.	6 HOLES, EQUI–SP DIA 17 C BORE FOR M 16 SOCKET HD CAP SCR	Drill a through hole of ϕ 17 and counterbore to insert a socket headed cap screw of M 16. Six holes are to be made equi-spaced on the circle.
5.	KEYWAY, WIDE 6 DEEP 3	Cut a key way of 6 mm wide and 3 mm depth.
6.	KEY SEAT, WIDE 10 DEEP 10	Cut a key seat of 10 mm wide and 10 mm deep to the length shown.
7.	U/C, WIDE 6 DEEP 3	Machine an undercut of width 6 mm and dpeth 3 mm.
8.	(*a*) DIAMOND KNURL 1 RAISED 30°	Make a diamond knurl with 1 mm pitch and end chamfer of 30°.
	(*b*) M 18 × 1	Cut a metric thread of nominal diameter 18 mm and pitch 1 mm.
9.	(*a*) THD RELIEF, DIA 20 WIDE 3.5	Cut a relief for thread with a diameter of 20.8 mm and width 3.5 mm.
	(*b*) NECK, WIDE 3 DEEP 1.5	Turn an undercut of 3 mm width and 1.5 mm depth
	(*c*) CARB AND HDN	Carburise and harden.
10.	(*a*) CARB, HDN AND GND	Carburise, harden and grind.
	(*b*) MORSE TAPER 2	Morse taper No. 1 to be obtained.
11.	DIA 6 REAM FOR TAPER PIN	Drill and ream with taper reamer for a diameter of 6 mm to suit the pin specified.
12.	6 ACME THD	Cut an ACME thread of pitch 6 mm.



11 12

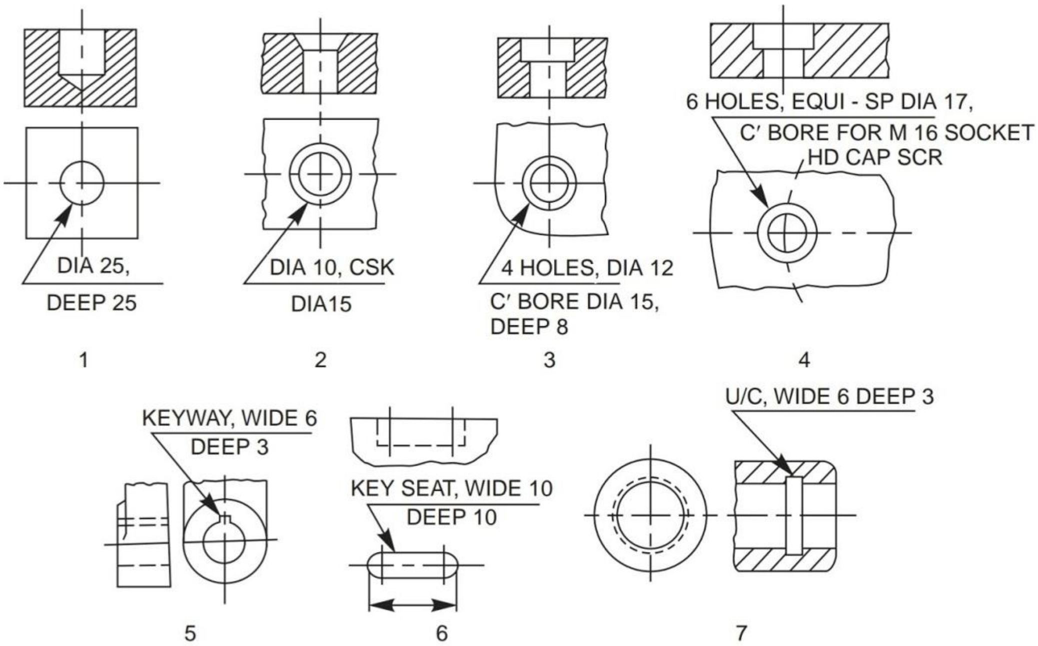
Fig. 2.55 Method of indicating notes

5 6 7



8 9 10

Fig. 2.55 Method of indicating notes (*Contd.*)


DIA 25,
DEEP 25
DIA 10, CSK
DIA15
4 HOLES, DIA 12
C′ BORE DIA 15,
DEEP 8
6 HOLES, EQUI - SP DIA 17,
C′ BORE FOR M 16 SOCKET
HD CAP SCR
1
2
3
4
KEYWAY, WIDE 6
DEEP 3
KEY SEAT, WIDE 10
DEEP 10
U/C, WIDE 6 DEEP 3
5
6
7

Experiment No 4

Conventional practices indicating Dimensional, Form & Position tolerances.

Tolerances of size are not always sufficient to provide the required control of form. For example, in Fig. 15.15 at the shaft has the same diameter measurement in all possible positions but is notcircular; in Fig. 15.15 b, the component has the same thickness throughout but is not flat and in Fig.
15.15 c, the component is circular in all cross-sections but is not straight. The form of these components can be controlled by means of geometrical tolerances.

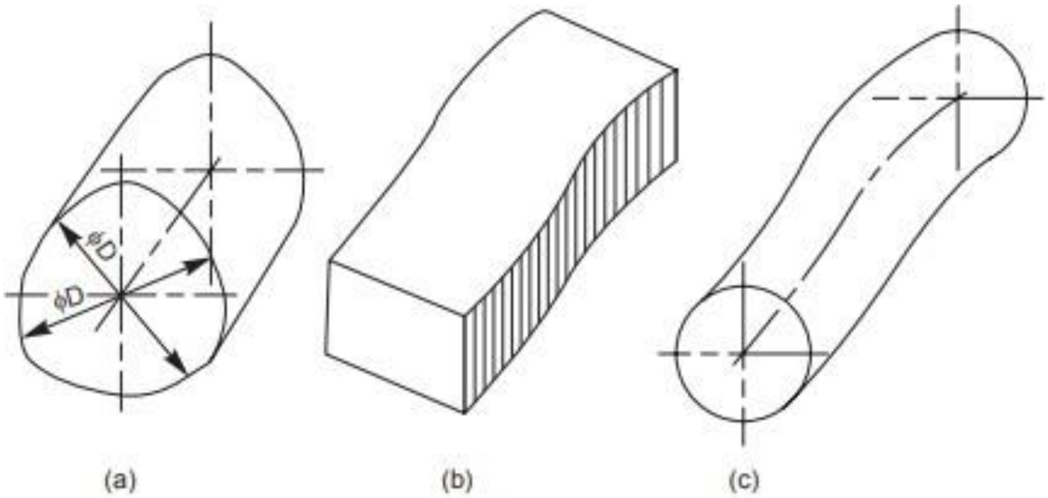


Fig. 15.15 Errors of form

Form variation:

It is a variation of the actual condition of a form feature (surface, line) from geometrically ideal form.

Position Variation:

It is a variation of the actual position of the form feature from the geometrically ideal position, with reference to another form (datum) feature.

Geometrical tolerance:

It is defined as the maximum permissible overall variation of form or position of a feature. Geometrical tolerances are used,

- (i) to specify the required accuracy in controlling the form of a feature,
- (ii) to ensure correct functional positioning of the feature,
- (iii) to ensure the interchangeability of components, and
- (iv) to facilitate the assembly of mating components.

Tolerance Zone:

It is an imaginary area or volume within which the controlled feature of the manufactured component must be completely contained (Figs. 15.16 a and b).

Datum:

It is a theoretically exact geometric reference (such as axes, planes, straight lines, etc.) to which the tolerance features are related (Fig. 15.17).

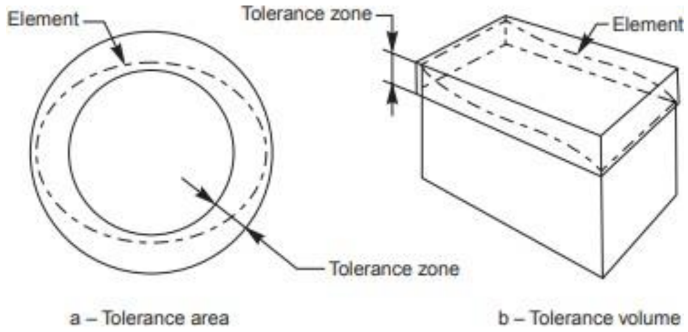


Fig. 15.16

Datum feature:

It is a feature of a part, such as an edge, surface, or a hole, which forms the basis for a datum or is used to establish its location (Fig. 15.17).

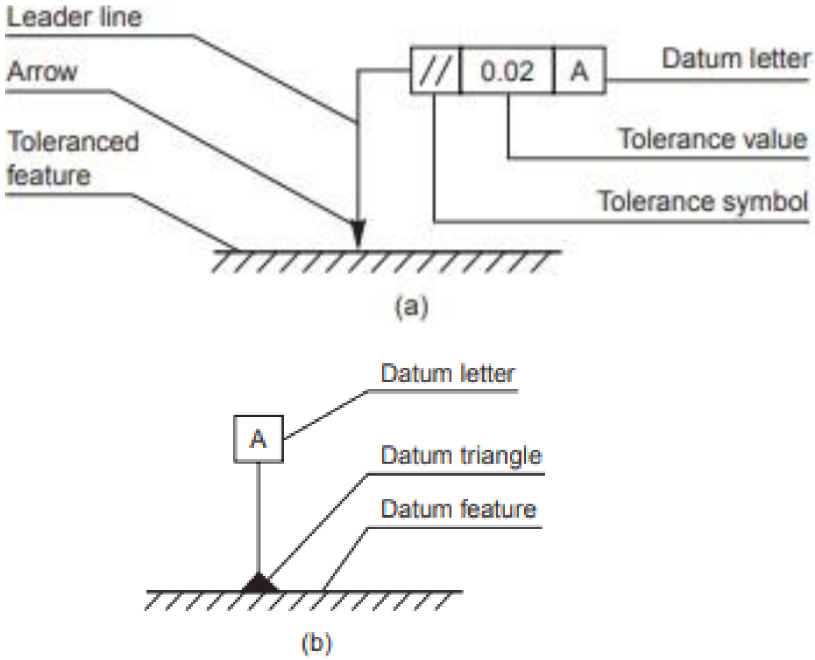


Fig. 15.17

Parts of datum feature:

1. The datums are indicated by a leader line, terminating in a filled or an open triangle (Fig. 15.17).
2. Datum Letter: To identify a datum for reference purposes, a capital letter is enclosed in a frame, connected to the datum triangle (Fig. 15.17).
3. The datum feature is the feature to which tolerance of orientation, position and run-out are related. Further, the form of a datum feature should be sufficiently accurate for its purpose and it may therefore be necessary in some cases to specify tolerances of form from the datum features. Table 15.7 gives symbols, which represent the types of characteristics to be controlled by the tolerance.

Indicating geometric tolerances on the drawing

To eliminate the need for descriptive notes, geometrical tolerances are indicated on drawings by symbols, tolerances and datums, all contained in compartments of a rectangular frame, as shown in Fig. 15.17.

Table 15.7 Symbols representing the characteristics to be toleranced

Characteristics to be toleranced		*Symbols*
Form of single features	Straightness	⏤
	Flatness	⏥
	Circularity (roundness)	○
	Cylindricity	⌭
	Profile of any line	⌒
	Profile of any surface	⌓
Orientation of related features	Parallelism	//
	Perpendicularity (squareness)	⊥
	Angularity	∠
Position of related features	Position	⌖
	Concentricity and coaxiality	◎
	Symmetry	⌯
	Run-out	↗

The tolerance frame is connected to the tolerance feature by a leader line, terminating with an arrow in the following ways:

- 1. On the outline of the feature or extension of the outline, but not a dimension line, when the tolerance refers to the line or surface itself (Figs. 15.18 a to c), and
- 2. On the projection line, at the dimension line, when the tolerance refers to the axis or median plane of the part so dimensioned (Fig. 15.18 d) or on the axis, when the tolerance refers to the axis or median plane of all features common to that axis or median plane (Fig. 15.18 e).

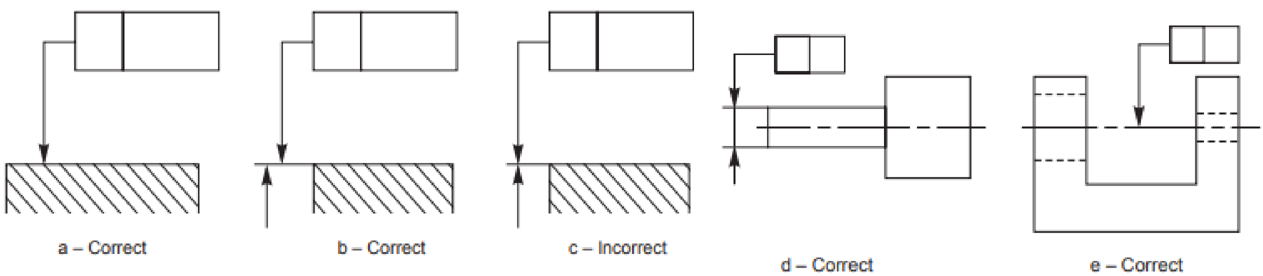


Fig. 15.18 Indication of feature controlled (outline or surface only)

Experiment No 5

Calculation of limits, suggestion of suitable fits for mating parts with Interference detection.

Fits:

The relation between the two mating parts is known as a fit. Depending upon the actual limits of the hole or shaft sizes, fits may be classified as clearance fit, transition fit and interference fit.

Clearance Fit:

It is a fit that gives a clearance between the two mating parts.

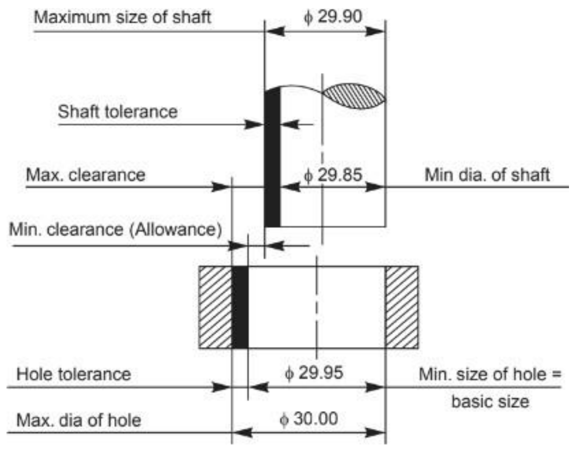


Fig. 15.10 Clearance fit

Minimum clearance:

It is the difference between the minimum size of the hole and the maximum size of the shaft in a clearance fit.

Maximum Clearance:

It is the difference between the maximum size of the hole and the minimum size of the shaft in a clearance or transition fit.

The fit between the shaft and hole in Fig. 15.10 is a clearance fit that permits a minimum clearance (allowance) value of 29.95 – 29.90 = + 0.05 mm and a maximum clearance of + 0.15 mm.

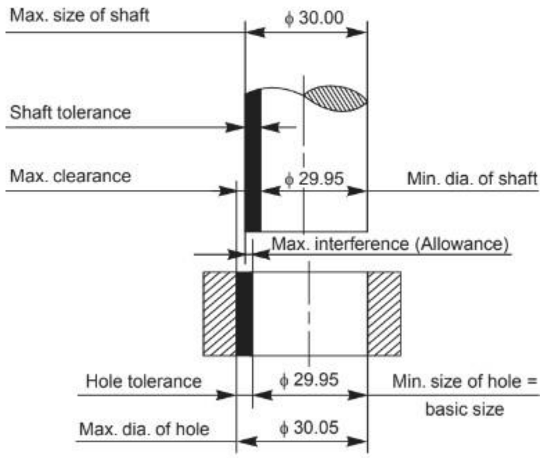


Fig. 15.11 Transition fit

Transition Fit:

This fit may result in either an interference or a clearance, depending upon the actual values of the tolerance of individual parts. The shaft in Fig. 15.11 may be either smaller or larger than the hole and still be within the prescribed tolerances. It results in a clearance fit when shaft diameter is 29.95 and hole diameter is 30.05 (+ 0.10 mm) and interference fit, when shaft diameter is 30.00 and hole diameter 29.95 (— 0.05 mm).

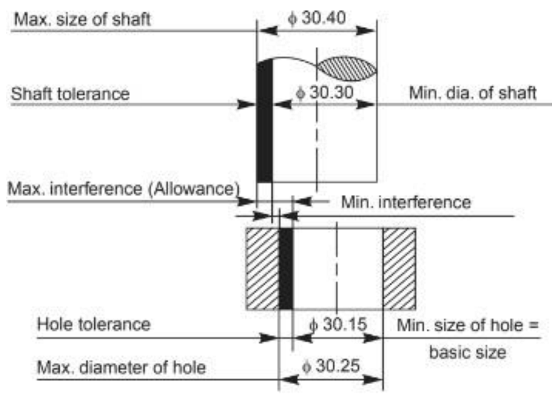


Fig. 15.12 Interference fit

Interference Fit:

If the difference between the hole and shaft sizes is negative before assembly; an interference fit is obtained.

Minimum Interference:

It is the magnitude of the difference (negative) between the maximum size of the hole and the minimum size of the shaft in an interference fit before assembly.

Maximum Interference:

It is the magnitude of the difference between the minimum size of the hole and the maximum size of the shaft in interference or a transition fit before assembly.

Clearance		*Transition*		*Interference*	
Hole basis	*Shaft basis*	*Hole basis*	*Shaft basis*	*Hole basis*	*Shaft basis*
H7 – c8	C8 – h7	H6 – j5	J6 – h5	H6 – n5	N6 – h5
H8 – c9	C9 – h8	H7 – j6	J7 – h6		
H11 – c11	C11 – h11	H8 – j7	J8 – h7	H6 – p5	P6 – h5
				H7 – p6	p7 – h6
H7 – d8	D8 – h7	H6 – k5	K6 – h5		
H8 – d9	D9 – h8	H7 – k6	K7 – h6	H6 – r5	R6 – h5
H11 – d11	D11 – h11	H8 – k7	K8 – h7	H7 – r6	R7 – h6
H6 – e7	E7 – h6	H6 – m5	M6 – h5	H6 – s5	S6 – h5
H7 – e8	E8 – h7	H7 – m6	M7 – h6	H7 – s6	S7 – h6
H8 – e8	E8 – h8	H8 – m7	M8 – h7	H8 – s7	S8 – h7
H6 – f6	F6 – h6	H7 – n6	N7 – h6	H6 – t5	T6 – h5
H7 – f7	F7 – h7	H8 – n7	N8 – h7	H7 – t6	T7 – h6
H8 – f8	F8 – h8			H8 – t7	T8 – h7
		H8 – p7	P8 – h7		
H6 – g5	G6 – h5			H6 – u5	U6 – h5
H7 – g6	G7 – h6	H8 – r7	R8 – h7	H7 – u6	U7 – h6
H8 – g7	G8 – h7			H8 – u7	U8 – h7

Table 15.6. Types of fits with symbols and applications

Type of fit	*Symbol of fit*	*Examples of application*
Interference fit		
Shrink fit	H8/u8	Wheel sets, tyres, bronze crowns on worm wheel
Heavy drive fit	H7/s6	hubs, couplings under certain conditions, etc.
Press fit	H7/r6	Coupling on shaft ends, bearing bushes in hubs, valve
Medium press fit	H7/p6	seats, gear wheels.
Transition fit		
Light press fit	H7/n6	Gears and worm wheels, bearing bushes, shaft and wheel assembly with feather key.
Force fit	H7/m6	Parts on machine tools that must be changed without damage, *e.g.,* gears, belt pulleys, couplings, fit bolts, inner ring of ball bearings.
Push fit	H7/k6	Belt pulleys, brake pulleys, gears and couplings as well as inner rings of ball bearings on shafts for average loading conditions.
Easy push fit	H7/j6	Parts which are to be frequently dismantled but are secured by keys, *e.g.,* pulleys, hand-wheels, bushes, bearing shells, pistons on piston rods, change gear trains.
Clearance fit		
Precision sliding fit	H7/h6	Sealing rings, bearing covers, milling cutters on milling mandrels, other easily removable parts.
Close running fit	H7/g6	Spline shafts, clutches, movable gears in change gear trains, etc.
Normal running fit	H7/f7	Sleeve bearings with high revolution, bearings on machine tool spindles.
Easy running fit	H8/e8	Sleeve bearings with medium revolution, grease lubricated bearings of wheel boxes, gears sliding on shafts, sliding blocks.
Loose running fit	H8/d9	Sleeve bearings with low revolution, plastic material bearings.
Slide running fit	H8/c11	Oil seals (Simmerrings) with metal housing (fit in housing and contact surface on shaft), multi-spline shafts.

Exercise: Solve the following questions manually in the Lab Records...

15.1 Calculate the maximum and minimum limits for both the shaft and hole in the following; using the tables for tolerances and name the type of fit obtained:
(*a*) 45H8/d7 (*b*) 180H7/n6 (*c*) 120H7/s6
(*d*) 40G7/h6 (*e*) 35 C11/h10

15.2 The dimensions of a shaft and a hole are given below:
Shaft, Basic size = 60mm and given as 60 – 0.020
Hole, Basic size = 60mm and given as 60 – 0.005
Find out:
(*a*) Tolerance of shaft (*b*) Tolerance of hole (*c*) Maximum allowance
(*d*) Minimum allowance (*e*) Type of fit

15.3 A schematic representation of basic size and its deviations are given in Fig. 15.19. Calculate the following in each case for a shaft of 50 mm basic size:
(*a*) Upper deviation (*b*) Lower deviation (*c*) Tolerance
(*d*) Upper limit size (*e*) Lower limit size

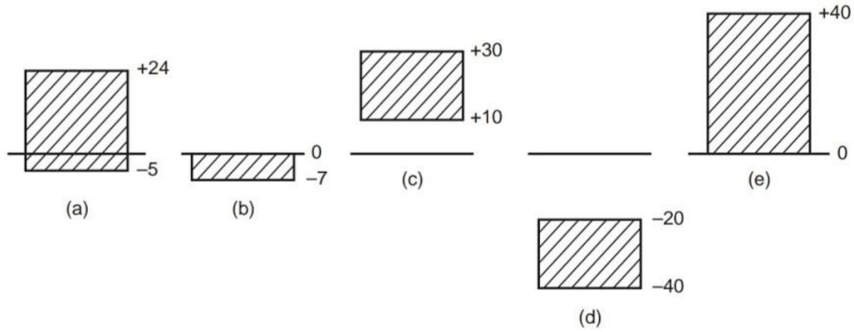


Fig. 15.19

A schematic representation of basic size and its deviationsare given in Fig. 15.20. Identify them.

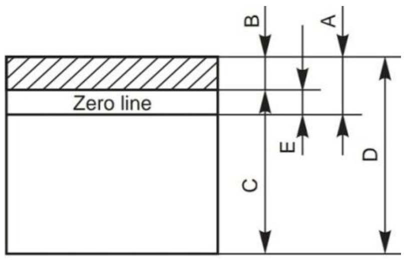


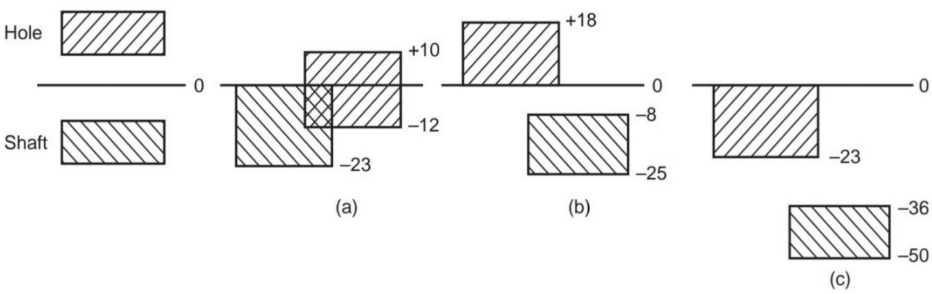
Fig. 15.20

A 30mm diameter hole is made on a turret lathe to the limits, 30.035 and 30.00. The following two grades of shaftsare used to fit in the hole:
(a) φ29·955mm and 29.925mm, and
(b) ϕ30.055mm and 30.050mm
Calculate the maximum tolerance, clearance and indicate the type of fit in each case by a sketch

Compute the limit dimensions for a clearance fit on the hole basis system, for a basic size of 40 mm diameter, with a minimum clearance of 0.05 mm; with the tolerance on the hole being 0.021 and the tolerance on the shaft being 0.15 mm.

Find the limit dimensions for an interference fit on the shaft basis system for the above problem and compare the dimensions of the shaft and hole.

Determine the type of fit and calculate the clearance and or interference for the schematic tolerance zones shown in Fig. 15.21

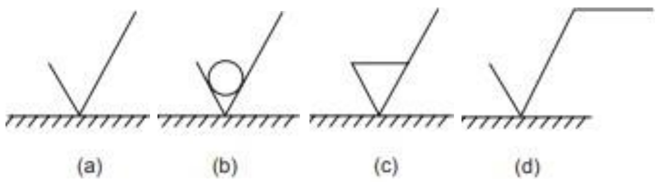


Experiment No 6

Surface finish, surface treatments - specification and indication methods on the drawings.

Machining Symbols

This article deals with the symbols and other additional indications of surface texture, to be indicated on production drawings. The basic symbol consists of two legs of unequal length, inclined at approximately 60° to the line, representing the surface considered (Fig. 16.2a). This symbol may be used where it is necessary to indicate that the surface is machined, without indicating the grade of roughness or the process to be used. If the removal of material is not permitted, a circle is added to the basic symbol, as shown in Fig. 16.2b. This symbol may also be used in a drawing, relating to a production process, to indicate that a surface is to be left in the state, resulting from a preceding manufacturing process, whether this state was achieved by removal of material or otherwise. If the removal of material by machining is required, a bar is added to the basic symbol, as shown in Fig. 16.2c. When special surface characteristics have to be indicated, a line is added to the longer arm of the basic symbol, as shown in Fig. 16.2d.



16.2

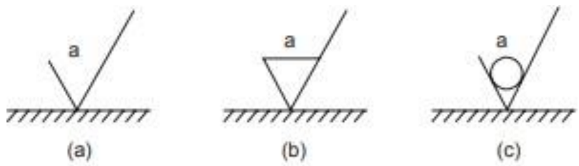


Fig. 16.3

The value or values, defining the principal criterion of roughness, are added to the symbol as shown in Fig. 16.3.

A surface texture specified, as in Fig. 16.3a, may be obtained by any production method. as in Fig. 16.3b, must be obtained by removal of material by machining. as in Fig. 16.3c, must be obtained without removal of material. When only one value is specified to indicate surface roughness, it represents the maximum permissible value. If it is necessary to impose maximum and minimum limits of surface roughness, both the values should be shown, with the maximum limit, a1, above the minimum limit, a2 (Fig. 16.4a).

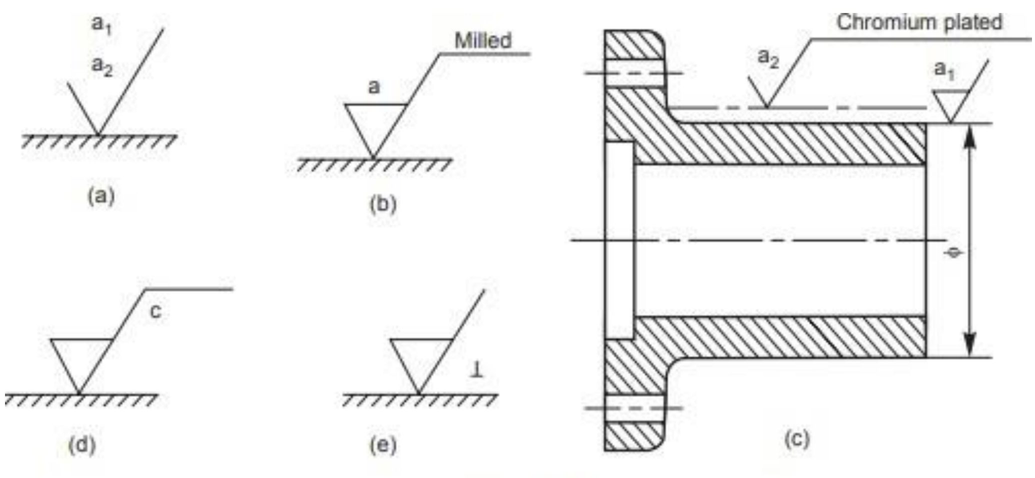


Fig. 16.4

The principal criterion of surface roughness, Ra may be indicated by the corresponding roughness grade number, as shown in Table 16.2

Table 16.2 Equivalent surface roughness symbols

Roughness values R_a *µm*	*Roughness grade number*	*Roughness grade symbol*
50	N12	~
25	N11	▽
12.5	N10	
6.3	N9	▽▽
3.2	N8	
1.6	N7	
0.8	N6	▽▽▽
0.4	N5	
0.2	N4	
0.1	N3	▽▽▽▽
0.05	N2	
0.025	N1	

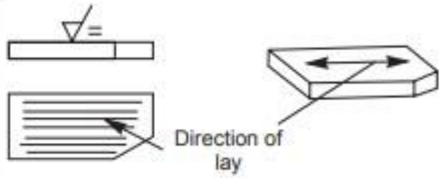
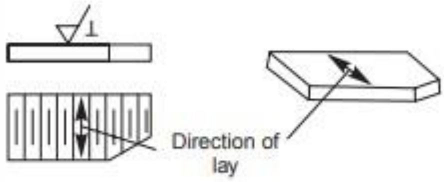
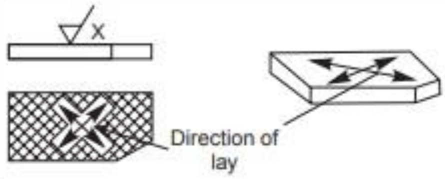
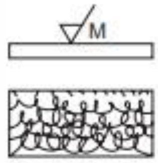
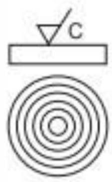
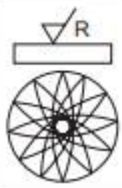
Indication of special surface roughness characteristics

In certain circumstances, for functional reasons, it may be necessary to specify additional special requirements concerning surface roughness. If it is required that the final surface texture produced by one particular production method, this method should be indicated on an extension of the longer arm of the symbol as shown in Fig. 16.4b. Also, any indications relating to the treatment of coating may be given on the extension of the longer arm of the symbol. Unless otherwise stated, the numerical value of the roughness, applies to the surface roughness after treatment or coating. If it is necessary to define surface texture, both before and after treatment, this should be explained by a suitable note or as shown in Fig. 16.4c.

If it is necessary to indicate the sampling length, it should be selected from the series given in ISO/R 468 and be stated adjacent to the symbol, as shown in Fig. 16.4d. If it is necessary to control the direction of lay, it is specified by a symbol added to the surface roughness symbol, as shown in Fig. 16.4e

NOTE The direction of lay is the direction of the predominant surface pattern, ordinarily determined by the production method employed. Table 16.3 shows the symbols which specify the common directions of lay.

Table 16.3 Symbols specifying the directions of lay

Symbol	*Interpretation*	
=	Parallel to the plane of projection of the view in which the symbol is used	
⊥	Perpendicular to the plane of projection of the view in which the symbol is used	
X	Crossed in two slant directions relative to the plane of projection of the view in which the symbol is used	
M	Multi-directional	
C	Approximately circular, relative to the centre of the surface to which the symbol is applied	
R	Approximately radial, relative to the centre of the surface to which the symbol is applied	

Indication of Machining Allowances

When it is necessary to specify the value of the machining allowance, this should be indicated on the left of the symbol, as shown in Fig. 16.5a. This value is expressed normally in millimetres. Figure 16.5b shows the various specifications of surface roughness, placed relative to the symbol.

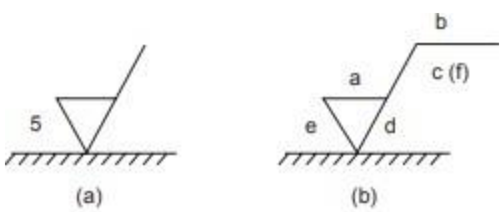


Fig. 16.5

Indications of Surface Roughness Symbols in Drawings

The symbol and the inscriptions should be so oriented, that they may be read from the bottom or the right hand side of the drawing (Fig. 16.6a). If it is not practicable to adopt this general rule, the symbol may be drawn in any position (Fig. 16.6b), provided that it does not carry any indications of special surface texture characteristics.

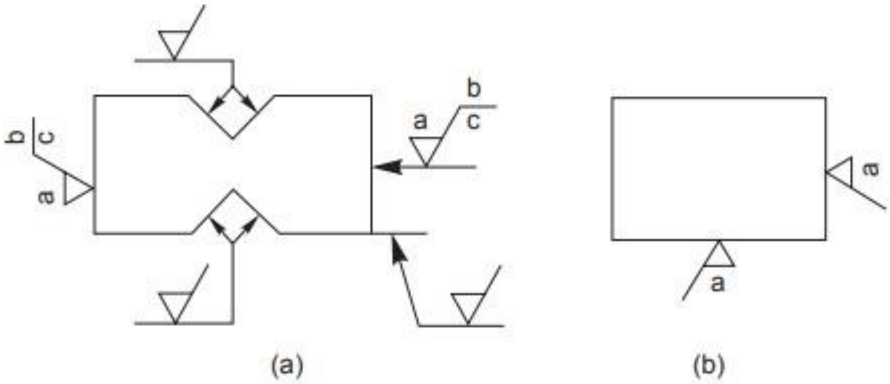


Fig. 16.6

The symbol may be connected to the surface by a leader line, terminating in an arrow. The symbol or the arrow should point from outside the material of the piece, either to the line representing the surface, or to an extension of it (Fig. 16.6a) In accordance with the general principles of dimensioning, the symbol is only used once for a given surface and, if possible, on the view which carries the dimension, defining the size or position of the surface (Fig. 16.7).

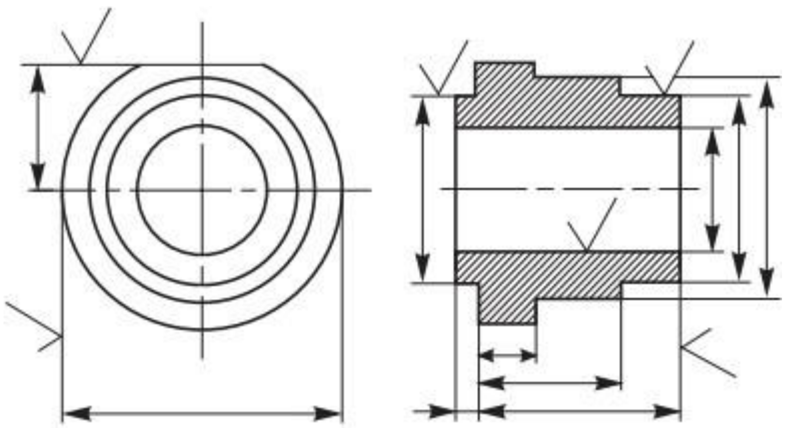


Fig. 16.7

If the same surface roughness is required on all the surfaces of a part, it is specified, either by a note near a view of the part (Fig. 16.8), near the title block, or in the space devoted to general notes, or following the part number on the drawing.

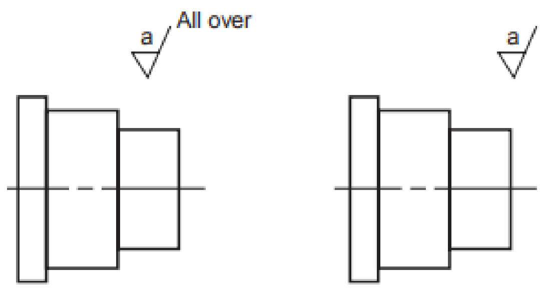


Fig. 16.8

If the same surface roughness is required on the majority of the surfaces of a part, it is specified with the addition of, the notation, except where otherwise stated (Fig. 16.9a), or a basic symbol (in brackets) without any other indication (Fig. 16.9b), or the symbol or symbols (in brackets) of the special surface roughness or roughness's (Fig. 16.9c).

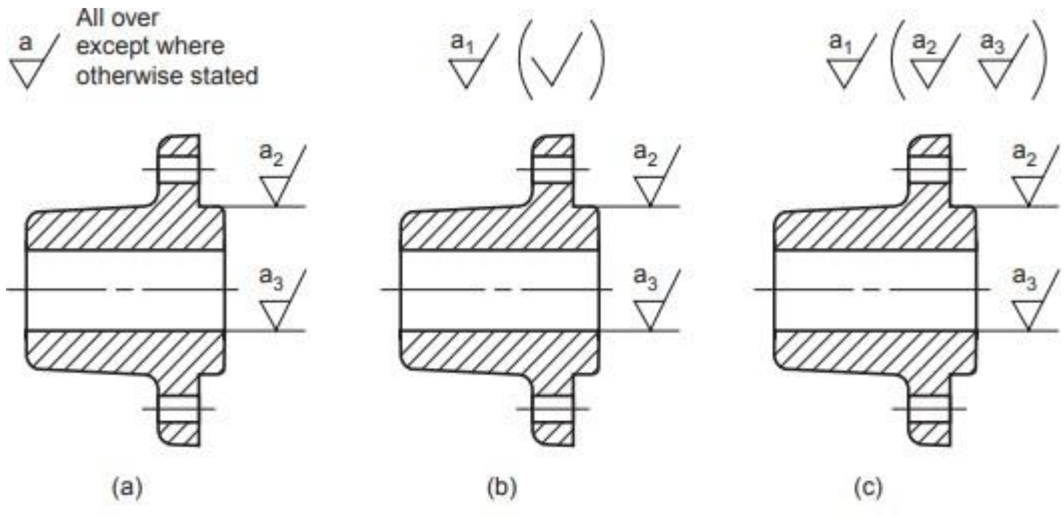


Fig. 16.9

To avoid the necessity of repeating a complicated specification a number of times, or where space is limited, a simplified specification may be used on the surface, provided that its meaning is explained near the drawing of the part, near the title block or in the space devoted to general notes (Fig. 16.10).

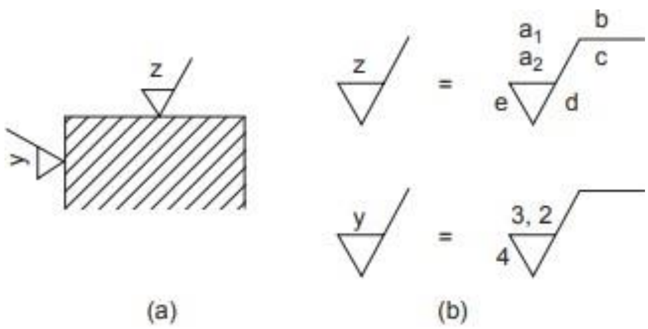


Fig. 16.10

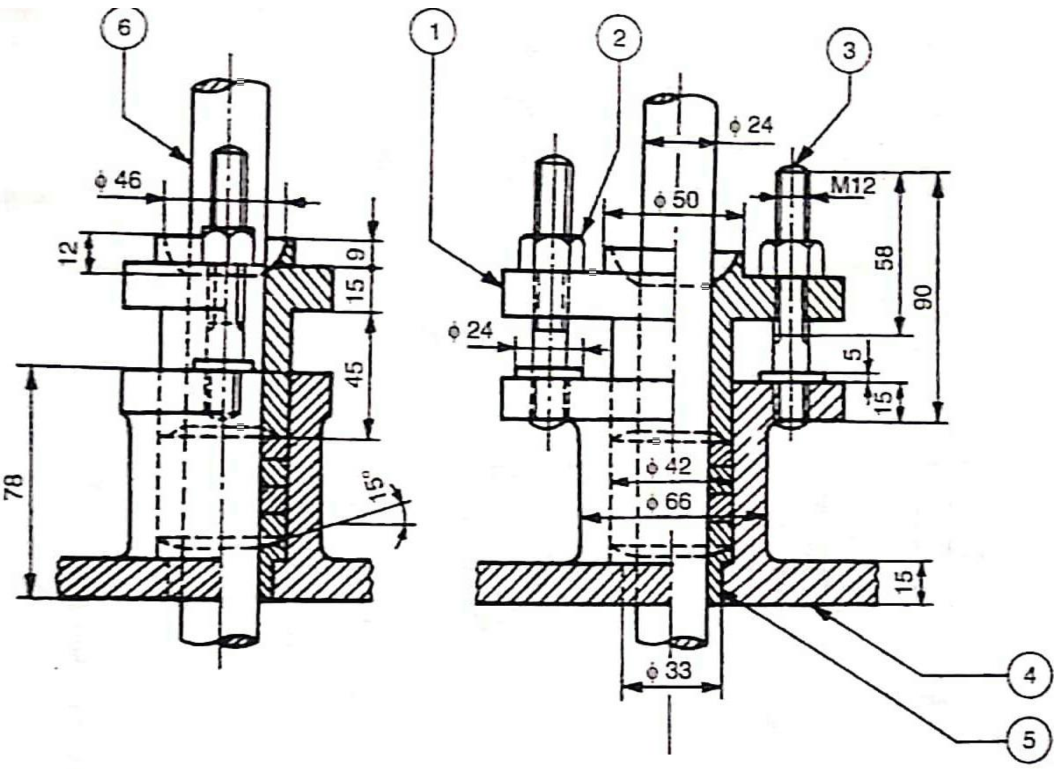
Experiment No 7

Generation of production drawings in 2D from part models representing Limits, fits. tolerances, Surface finish, geometrical and form tolerance etc.

1. Stuffing box

Aim:

Create a 3D model of each part from the assembly given below. Create the production drawing of each part along with tolerance (Dimensional, Geometric & Surface Finish) information. Create a Process chart for at least one component.



Parts List

Part No.	Name	Matl.	Qty.
1	Gland	Brass	1
2	Nut, M12	MS	2
3	Stud	MS	2
4	Body	CI	1
5	Bush	Brass	1
6	Shaft	MS	1

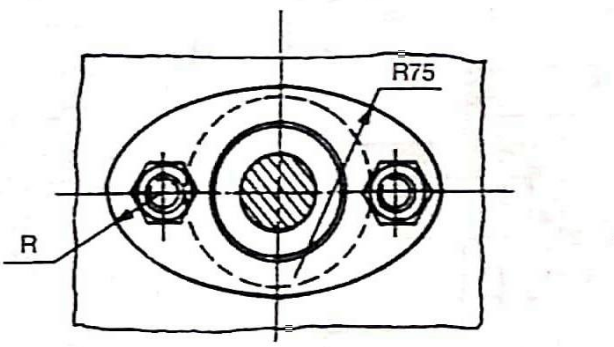


Fig 9.12 Stuffing box

Software Package Used:

SOLIDWORKS 2020

Hardware Specifications of System:

Processor:________________________,

RAM:_________________,

Hard Disk: _________________

Modules used:

Features used:

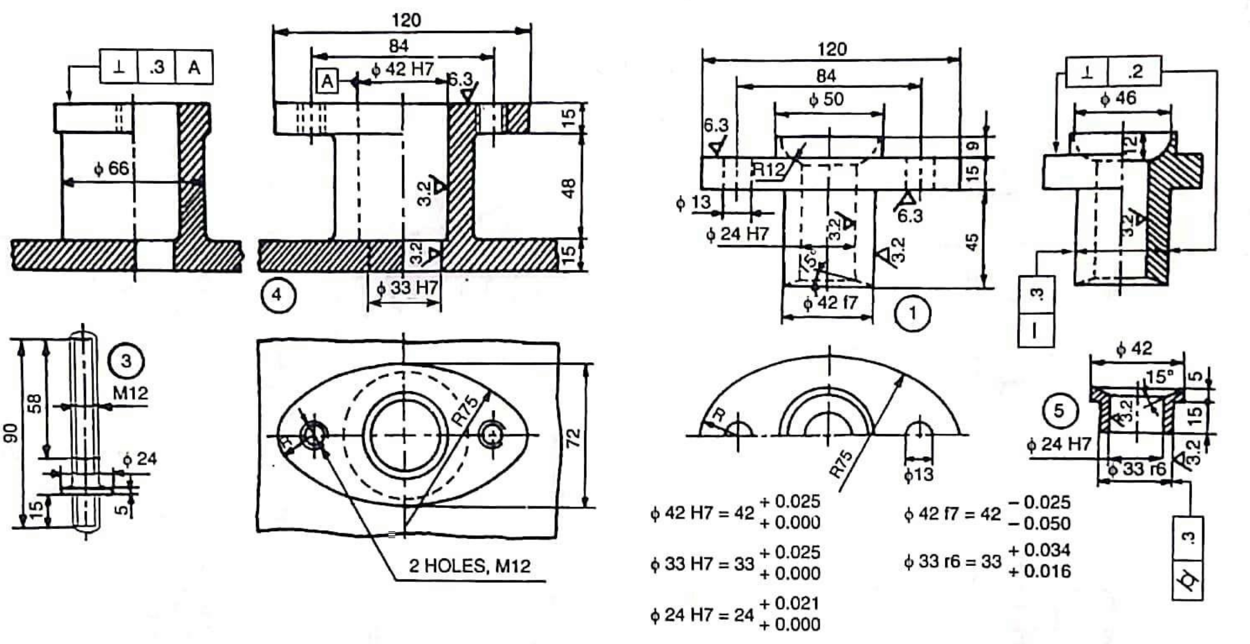
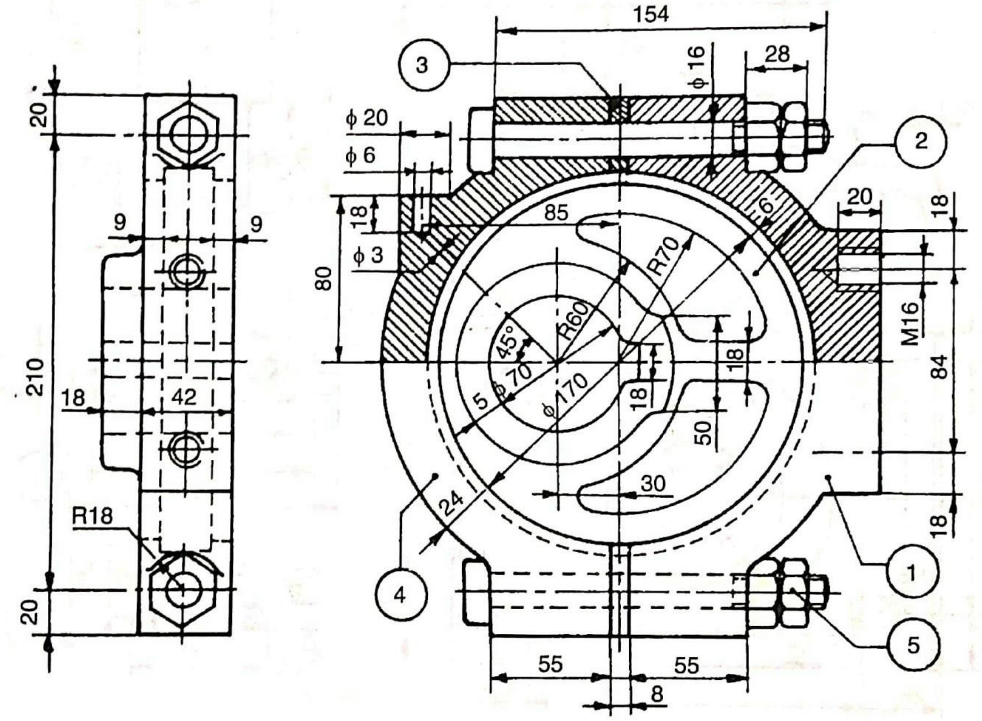


Fig. 9.13 Details of stuffing box

2. Eccentric

Aim:

Create a 3D model of each part from the assembly given below. Create the production drawing of each part along with tolerance (Dimensional, Geometric & Surface Finish) information. Create a Process chart for at least one component.



Parts List

Part No.	Qty.	Name	Matl.
1	1	Strap	CI
2	1	Sheave	CI
3	2	Shim	Brass
4	1	Strap	CI
5	2	Bolt with nuts	MS

Fig. 9.10 Eccentric

Software Package Used:
SOLIDWORKS 2020

Hardware Specifications of System:

Processor:________________________,
RAM:_________________,
Hard Disk: _________________

Modules used:

Features used:

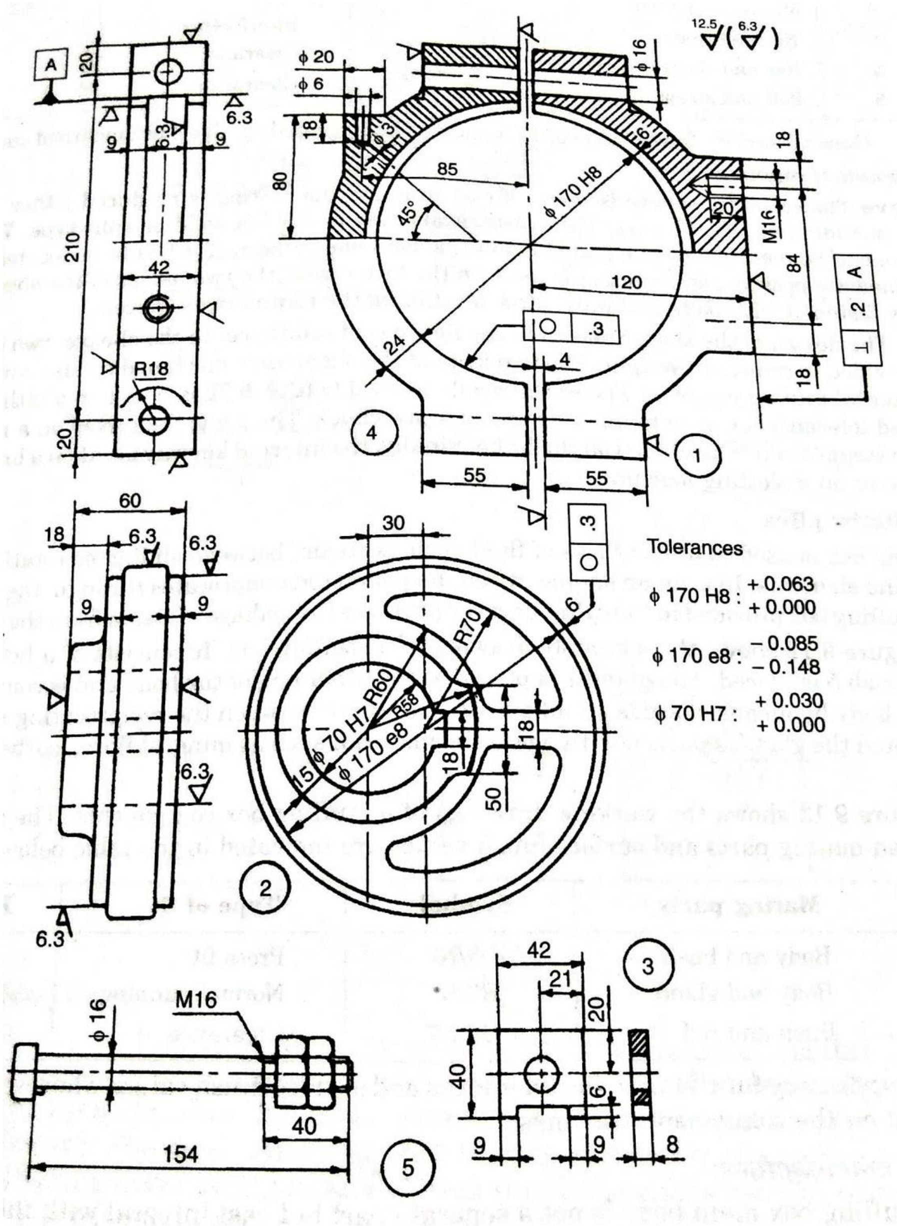


Fig. 9.11 Details of eccentric

Experiment No 8

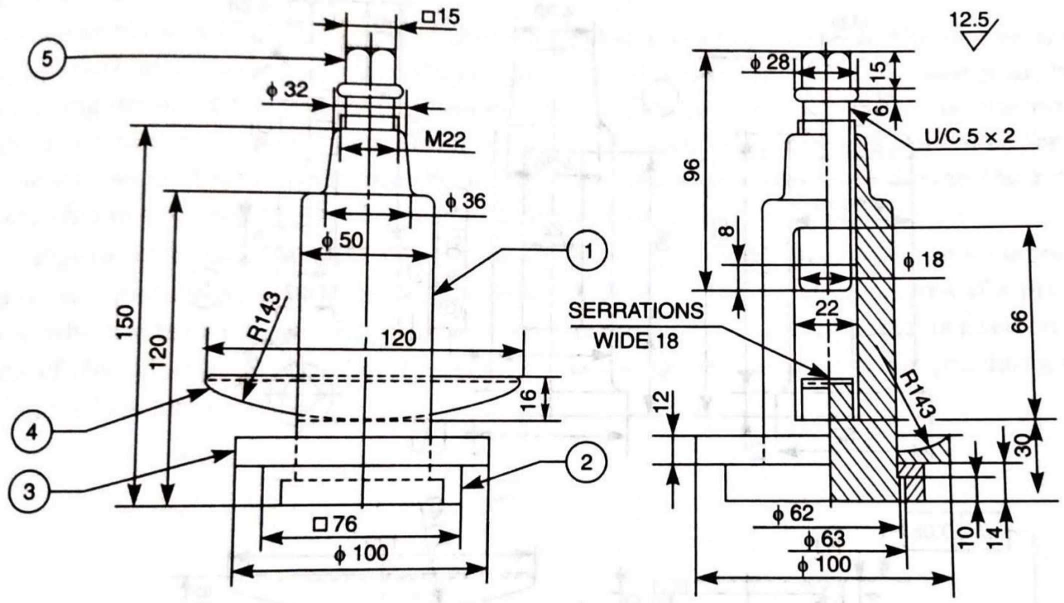
Preparation of Process sheet incorporating Tool work orientation diagrams.

3. Single tool post

Aim:

Create a 3D model of each part from the assembly given below. Create the production drawing of each part along with tolerance (Dimensional, Geometric & Surface Finish) information. Create a Process chart for at least one component.

Production Drawings of Assemblies **229**



Parts List

Part No.	Name	Matl.	Qty.
1	Pillar	MCS	1
2	Block	MCS	1
3	Ring	MS	1
4	Wedge	MCS	1
5	Screw	TS	1

Fig. 9.30 Single tool post

Software Package Used:

SOLIDWORKS 2020

Hardware Specifications of System:

Processor:_____________,RAM:______________________________,

Hard Disk: ________________

Modules used:

Features used:

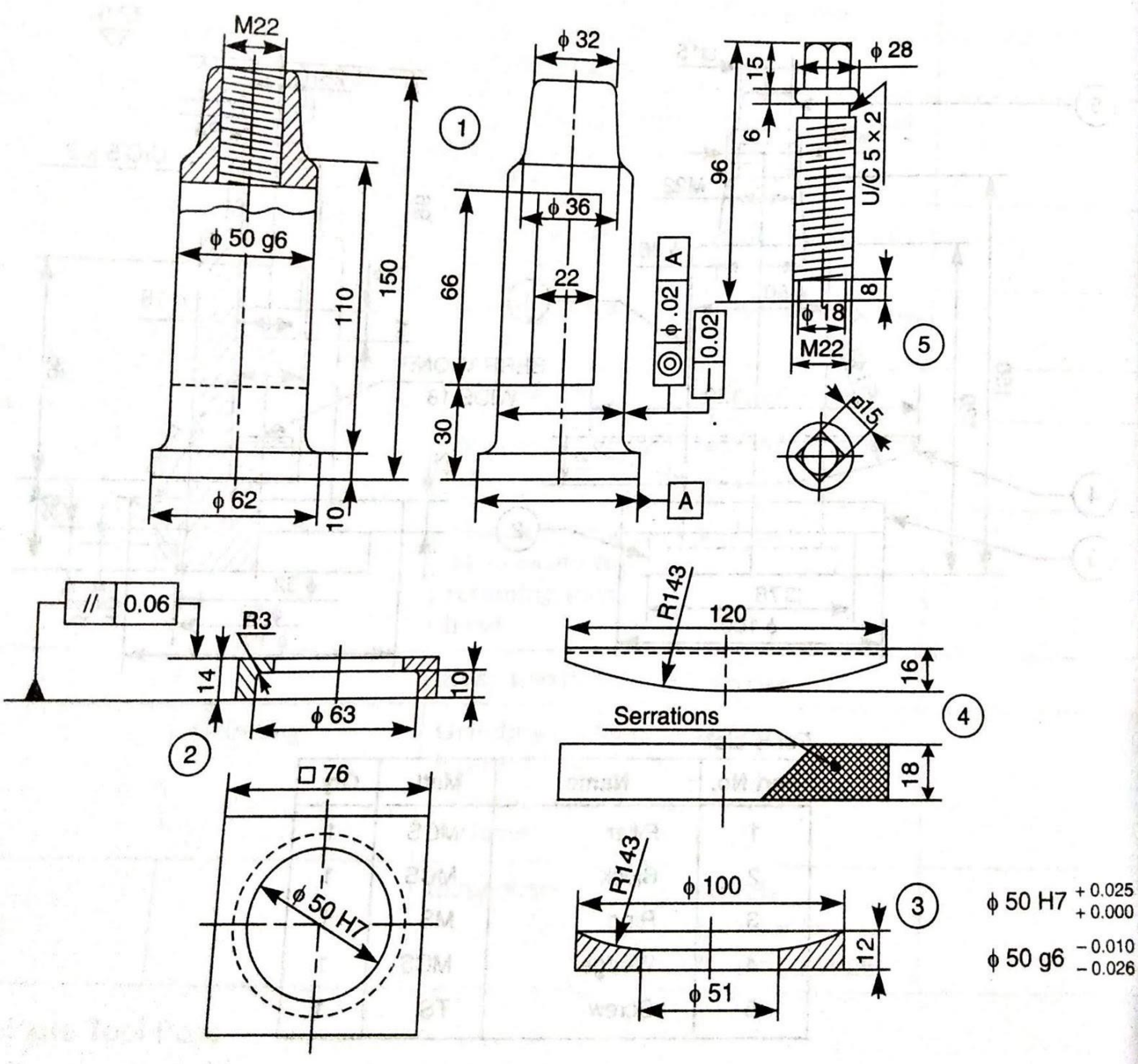
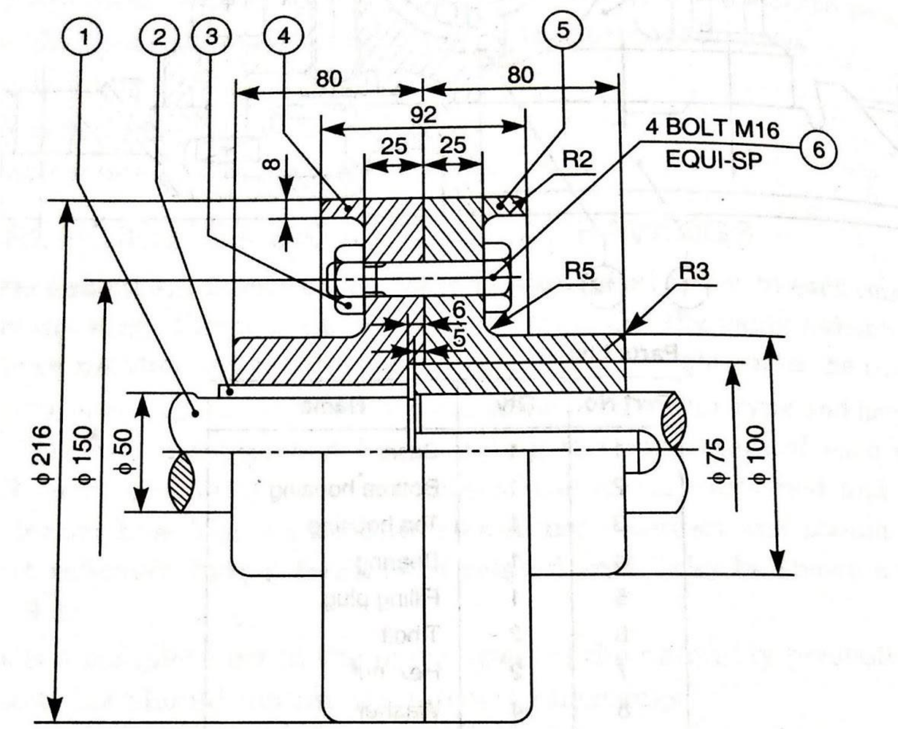


Fig. 9.31 Details of single tool post

4. Protected flange coupling

Aim:

Create a 3D model of each part from the assembly given below. Create the production drawing of each part along with tolerance (Dimensional, Geometric & Surface Finish) information. Create a Process chart for at least one component.



Parts List

Part No.	Qty.	Name	Matl.
1	2	Shaft	MS
2	2	Sunk key	MS
3	4	Nut	MS
4	1	Flange	CI
5	1	Flange	CI
6	4	Bolt	MS

Fig. 9.2 Protected flange coupling

Software Package Used:

SOLIDWORKS 2020

Hardware Specifications of System:

Processor:________________________,

RAM:_________________,

Hard Disk: ________________

Modules used:

Features used:

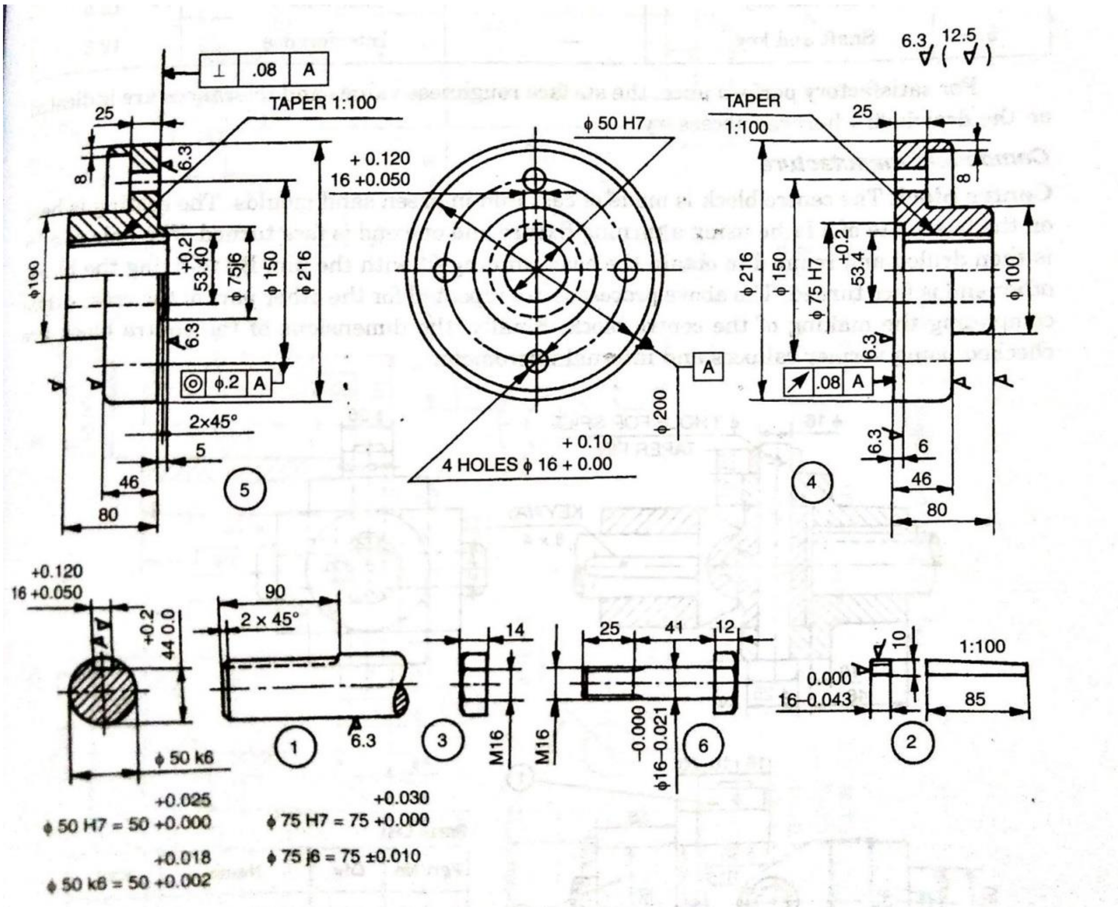


Fig. 9.3 Details of protected flange coupling

CAM MANUAL

INTRODUCTION ABOUT CNC MACHINE

Total 8 tools fixed in the turret 4 internal operation & 4 for external operation) & each tool is for the following operation:

1-Drill Bit 12mm Diameter

2-Turning

3-Boring 8mm Diameter

4-External Threading

5-Internal Threading 16mm

6-Grooving

7-Drill 6mm Diameter

8- No Tool, need to Fix the Tool in This

Tool materials:

Tool tip-carbide

Groove tool, drill bit- HSS

- Maximum length to be fixed in CNC machine is 70mm & maximum diameter is 32mm

Spindle speeds for each operation is as follows:

Turning-1200rpm

Threading-500rpm

Drilling-1000rpm

Boring-1200rpm

Grooving-1000rpm

Facing-1000rpm

Steps to be followed for writing the new program:

monitor-edit-prgm-local disk(d) EOB -prgm-mcet-EOB -EOB -prgm-edit-new-enter prgm no EOB-reset (save)-monitor-path-cls-middle-for zoom (preview pageup)

Steps to be followed for tool change:

Monitor-jog-edit-MD1-change the tool no-start-EOB

Format in tool change:

G54G18G00 MO6T0202
M30
%
Where,

M06 represents tool change,

0202 represents the tool number to be changed. (02-tool number, 02-tool offset)

Steps need to be followed for tool offset

Monitor-jog-touch the workpiece(+, X-, zt, Z-)-edit-coordinate-, go to tool number we are using-cut diameter-12-monitor-jog (x+, zt) touch the workpiece - edit coordinate-go to tool number-cut length-0-jog(z+) home-start.

For running the program:

Auto-SBK-start-press the start button until the red light comes.

SBK-single block i.e program will run step by step (few step machining takes place)

Deselect SBK and press AUTO for continuous operation

Backup

Monitor-File- local disk(D)-EOB- Program -EOB-MCET-Copy (F3)- File-Data traveler-EOB f4 (Paste)

Recovery

Insert pendrive-monitor-system-manage-select setup-super user-"00000" (password)-EOB - Para recover-EOB-again change to guest mode- EOB. New text document

File

Save as

All files

Loading a program

Home-start- monitor- file-edit-local disk(D)- EOB-program-EOB-MCET-EOB-EOB-program edit-program section-offset.

Backup File Open Format in System

New text document-file-open-file name "should be :CNC" File location should be "All files".

The following are the codes used in writing the CNC program:

G70 - Finishing cycle

G71 - Multiple rough turning cycle.

G72 - Multiple rough facing cycle.

G74 - Drilling cycle

G75 - Grooving cycle.

G76 - Threading cycle.

G90 - Step turning

G94 - Face cycle

G21- Metric input (mm)

G20 - Inches

G28 - Go to reference point.

G98- feed rate mm/min (or) Feed rate per time.

G99-feed rate mm/rev

M03-Spindle clockwise.

M05-Spindle Stop.

U-Finishing allowance in X

W - Finishing allowance in Z

P - Program starting end block (or) no of the 1st block of the shape.

Q - Program ending end block (or) no of the last block of the shape.

FORMAT NEED TO BE FOLLOWED IN WRITING THE CNC PROGRAMS:

- **For Drilling (G74)**

G74 R1

G74 X__Z__Q__F__

Where,

R1-Tool Retract

X-Center Point of WorkPiece

Z-Tool Depth of Hole

Q-Each Pecking Depth

F-Feed Rate

To drill the hole in a single pass (without pecking), set Q equal to the depth of the hole.

- **For Grooving Cycle(G75)**

GOO X max Z start

G75 R1

G75 X.min Z end Q__P__F__

Where,
Q-Tool Shift in mm
P-each depth of cut in mm
F-Feed rate

- **For Threading Cycles (G76)**

G76 P030060 Q0.05 R0.02
G76 X min Z end QO.1 P0.919 F1.5

Where,

P03- Finishing Process

00-Pull Cut (Or) Tool Relief Angle

60-Thread Angle

Q0.05- Each Depth of Thread

R0.02- Finishing Allowance

Q0.1-First Thread Depth Cut

P0.919-Thread Height F-Pitch

- **Turning Cycle (G71)**

G71 D(d)__R__
G71 P(s)__Q(e)__U(u)__W__F__
N........

Where,

U(d)- depth of cut, radius value

R- Retract amount, radius value

P- no. of the first block of the shape

Q- no. of the last block of the shape

U(u)- finishing allowance in X, diameter value

W-finishing allowance in Z

f-feed rate

Programs saved in CNC machine are for the following operations

MCET 0001/Program 1 - Multiple turning
MCET 0002/Program 2- Multiple turning
MCET 0003/Program 3- Facing
MCET 0004/Program 4-Threading external, grooving
MCET 0005/Program 5-Internal boring/turning
MCET 0006/Program 6-Internal threading
MCET 0007/Program 7-Drilling

Experiment No 9

To perform simple turning, step turning, chamfering & fillet operations on a given shaft using CNC lathe.

AIM:
To machine the given aluminium shaft into the required shape using simple turning, step turning, chamfering & fillet operations on a given shaft using CNC lathe.

Apparatus
CNC Lathe Machine

Specification
CNC slant Bed Lathe
Total 8 tools fixed in the turret 4 internal operation & 4 for external operation

Procedure

- **Program Selection**
  Program-edit(F1)-Program selection (F2)-program 1-EOB

- **Tool change**
  Monitor - jog - edit - MDI - give tool number i.e. M06T0202 - start - EOB

- **Tool offset**

➢ Monitor - jog - Move the tool towards the surface of the workpiece, for that, press Z- and X+ (doubleclick step mode for small (0.1mm) increments) until the tool touches the surface of the workpiece.
After it touches go for edit - coord - go to tool number 2 - give cut diameter values as **25.4** - EOB - home start.

➢ Monitor - jog Move the tool towards the face of the workpiece, for that, press Z- and X+ (doubleclick step mode for small (0.1mm) increments) until the tool touches the face of the workpiece.
After it touches go for edit - coord - go to tool number 2 - give cut length values as **0** - EOB - home start.

- **Running the Program**
  Monitor - Auto - SBK - Start.
  Press the start button until the red light flashes on the button.
  Deselect SBK - start (for continuous operation)

Program For Turning operations

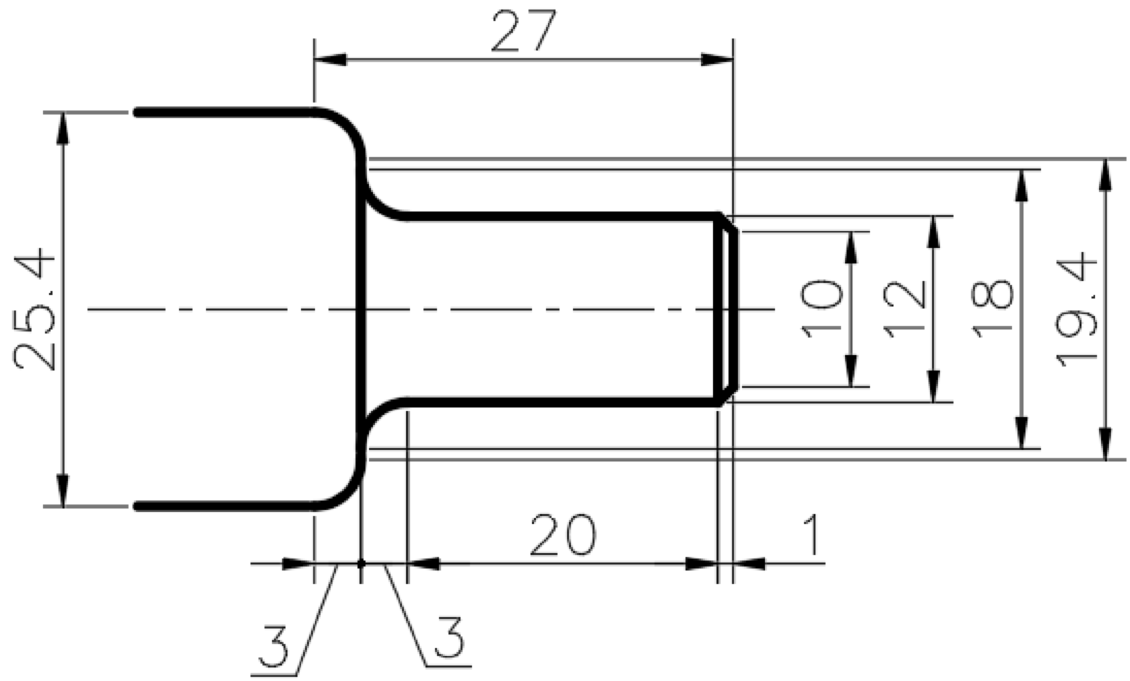
Diameter of the workpiece need to be fixed to perform this Program is 25.4 mm

```
G21 G98
G28 U0W0
M06 T0202
M03 S1200
G00 X25.4 Z-5
G71 U0.2 R0.5
G71 P1Q2 U0.1W0.1 F100
N1 G00X10
G01 Z0
G01 X12 Z-1
G01 X12 Z-21
G02 X18 Z-24 R3
G01 X19.4 Z-24
G03 Z25.4 Z-27 R3
G01 X25.4 Z-27
G70 P1Q1
G28 U0W0
M05
M30
```

For taking offset, we need to set the coordinates as follows:

Coordinates

S.No	Tool No	Cut diameter	Cut length
1	2	25.4	0


27
25.4
10
12
18
19.4
20
1
3
3

Experiment No 10

To perform the facing operation on a given shaft using CNC lathe

AIM:
To machine the given aluminium shaft into the required shape using the facing operation on a given shaft using CNC lathe.

Apparatus
CNC Lathe Machine

Specification
CNC slant Bed Lathe
Total 8 tools fixed in the turret 4 internal operation & 4 for external operation

Procedure

- **Program Selection**
  Program-edit(F1)-Program selection (F2)-program 3-EOB

- **Tool change**
  Monitor - jog - edit - MDI - give tool number i.e. M06T0202 - start - EOB

- **Tool offset**

➢ Monitor - jog - Move the tool towards the surface of the workpiece, for that, press Z- and X+ (doubleclick step mode for small (0.1mm) increments) until the tool touches the surface of the workpiece.
After it touches go for edit - coord - go to tool number 2 - give cut diameter values as **32** - EOB - home start.

➢ Monitor - jog Move the tool towards the face of the workpiece, for that, press Z- and X+ (doubleclick step mode for small (0.1mm) increments) until the tool touches the face of the workpiece.
After it touches go for edit - coord - go to tool number 2 - give cut length values as **0** - EOB - home start.

- **Running the Program**
  Monitor - Auto - SBK - Start.
  Press the start button until the red light flashes on the button.
  Deselect SBK - start (for continuous operation)

Program For Facing

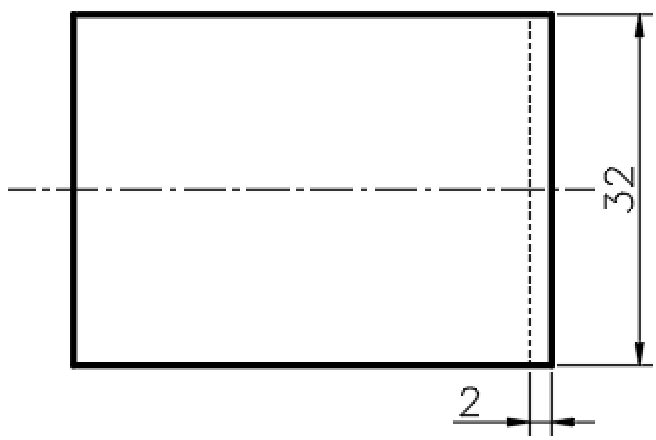
Diameter of the workpiece need to be fixed to perform this Program is 32 mm

```
G21 G98
G28 U0W0
M06 T0202
M03 S1200
G00 X32 Z5
G94 X-1 Z0 F40
Z-0.1
Z-0.2
Z-0.3
Z-0.4
Z-0.5
Z-0.6
Z-0.7
Z-0.8
Z-0.9
Z-1.0
Z-1.1
Z-1.2
Z-1.3
Z-1.4
Z-1.5
Z-1.6
Z-1.7
Z-1.8
Z-1.9
Z-2.0
G28 U0 W0
M05
M30
```

For taking offset, we need to set the coordinates as follows:

Coordinates

S.No	Tool No	Cut diameter	Cut length
1	2	32	0



Experiment No 11

To perform external threading and grooving operations on a given shaft using CNC lathe

AIM:
To machine the given aluminium shaft into the required shape using external threading and grooving operations on a given shaft using CNC lathe.

Apparatus
CNC Lathe Machine

Specification
CNC slant Bed Lathe
Total 8 tools fixed in the turret 4 internal operation & 4 for external operation

Procedure

- **Program Selection**
  Program-edit(F1)-Program selection (F2)-program 4-EOB

- **Tool change for Threading**
  Monitor - jog - edit - MDI - give tool number i.e. M06T0404 - start - EOB (threading tool)

- **Tool offset for Threading Tool**

➢ Monitor - jog - Move the tool towards the surface of the workpiece, for that, press Z- and X+ (doubleclick step mode for small (0.1mm) increments) until the tool touches the surface of the workpiece.
After it touches go for edit - coord - go to tool number 4 - give cut diameter values as **12** - EOB - home start.

➢ Monitor - jog Move the tool towards the face of the workpiece, for that, press Z- and X+ (doubleclick step mode for small (0.1mm) increments) until the tool touches the face of the workpiece.
After it touches go for edit - coord - go to tool number 4 - give cut length values as **0** - EOB - home start.

- **Tool change for Grooving**
  Monitor - jog - edit - MDI - give tool number i.e. M06T0606 - start - EOB (grooving tool)

- **Tool offset for Grooving Tool**

➢ Monitor - jog - Move the tool towards the surface of the workpiece, for that, press Z- and X+ (doubleclick step mode for small (0.1mm) increments) until the tool touches the surface of the workpiece.
After it touches go for edit - coord - go to tool number 6 - give cut diameter values as **12** - EOB - home start.

➢ Monitor - jog Move the tool towards the face of the workpiece, for that, press Z- and X+ (doubleclick step mode for small (0.1mm) increments) until the tool touches the face of the workpiece.
After it touches go for edit - coord - go to tool number 6 - give cut length values as **0** - EOB - home start.

- **Running the Program**
  Monitor - Auto - SBK - Start.
  Press the start button until the red light flashes on the button.
  Deselect SBK - start (for continuous operation)

Program For External Threading and Grooving

Workpiece obtained after performing Multiple Turning operation (diameter 25.4) Need to be fixed to perform this program.

G21 G98

G28 U0W0

M06 T0404

M03 S500

G00 X12 Z2

G76 P030060 Q0.05 R0.02

G76 X10.161 Z-12 Q0.1 P0.919 F1.5

G28 U0W0

M05

M06 T0606

M03 S1000

G00 X12.2 Z-15

G75 R1

G75 X8 Z-17 Q0.5 P0.01 F30

G28 U0W0

M05

M30

Where

T0404 : Threading tool

S500 : Constant speed for threading

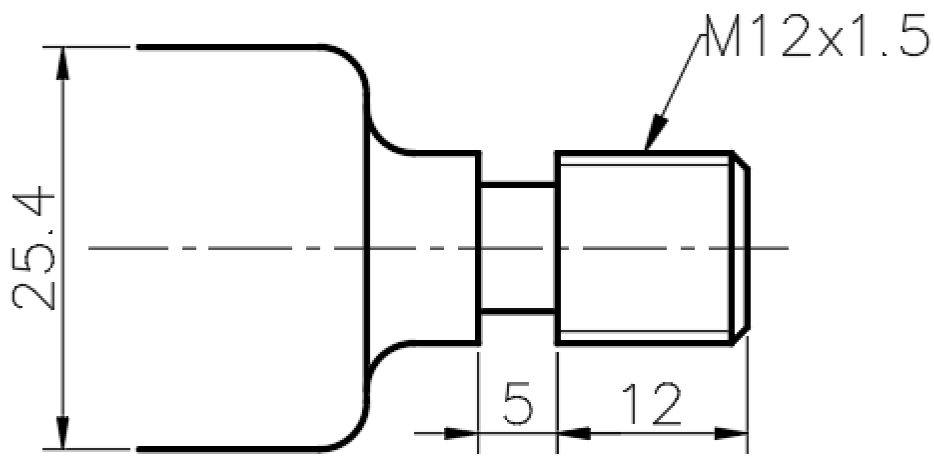
Q0.1 : First thread depth

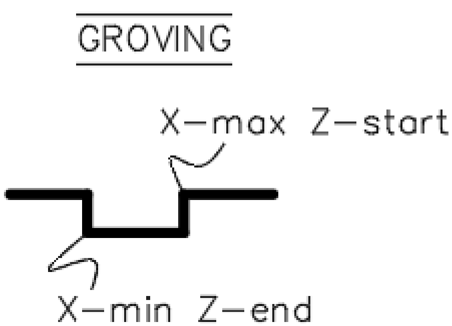
T0606 : Grooving Tool

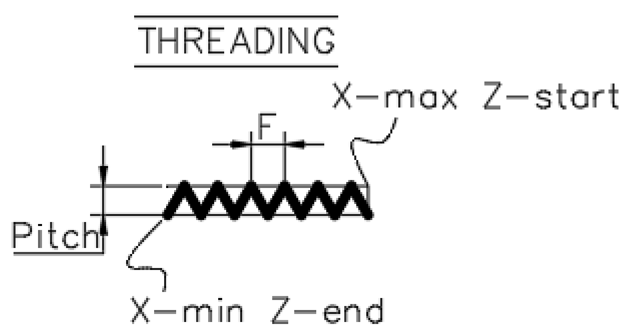
For taking offset, we need to set the coordinates as follows:

Coordinates

S.No	Tool No	Cut diameter	Cut length
1	4	12	0
2	6	12	0


M12x1.5
25.4
5
12


GROVING
X-max Z-start
X-min Z-end


THREADING
X-max Z-start
F
Pitch
X-min Z-end

Experiment No 12

To perform drilling operation on a given shaft using CNC lathe

AIM:
To machine the given aluminium shaft into the required shape drilling operation on a given shaft using CNC lathe.

Apparatus
CNC Lathe Machine

Specification
CNC slant Bed Lathe
Total 8 tools fixed in the turret 4 internal operation & 4 for external operation

Procedure

- **Program Selection**
  Program-edit(F1)-Program selection (F2)-program 7-EOB

- **Tool change**
  Monitor - jog - edit - MDI - give tool number i.e. M06T0707 - start - EOB

- **Tool offset**

➢ Monitor - jog - Move the tool towards the surface of the workpiece, for that, press Z- and X+ (doubleclick step mode for small (0.1mm) increments) until the tool touches the surface of the workpiece.
After it touches go for edit - coord - go to tool number 7 - give cut diameter values as **38** (32+6) - EOB - home start. (32 is the diameter of the workpiece and 6 drill bit diameter).

➢ Monitor - jog Move the tool towards the face of the workpiece, for that, press Z- and X+ (doubleclick step mode for small (0.1mm) increments) until the tool touches the face of the workpiece.
After it touches go for edit - coord - go to tool number 7 - give cut length values as **0** - EOB - home start.

- **Tool change**
  Monitor - jog - edit - MDI - give tool number i.e. M06T0101 - start - EOB

- **Tool offset**

➢ Monitor - jog - Move the tool towards the surface of the workpiece, for that, press Z- and X+ (doubleclick step mode for small (0.1mm) increments) until the tool touches the surface of the workpiece.
After it touches go for edit - coord - go to tool number 7 - give cut diameter values as **44** (32+12) - EOB - home start. (32 is the diameter of the workpiece and 12 drill bit diameter).

➢ Monitor - jog Move the tool towards the face of the workpiece, for that, press Z- and X+ (doubleclick step mode for small (0.1mm) increments) until the tool touches the face of the workpiece.
After it touches go for edit - coord - go to tool number 1 - give cut length values as **0** - EOB - home start.

- **Running the Program**
  Monitor - Auto - SBK - Start.
  Press the start button until the red light flashes on the button.
  Deselect SBK - start (for continuous operation)

Program For Drilling

Diameter of the workpiece need to be fixed to perform this Program is 32 mm

G21 G98

G28 U0W0

M06 T0707 (Ø 6)

M03 S1000

G00 X0 Z5

G74 R1

G74 X0 Z-5 Q0.1 F40

G28 U0 W0

M05

M06 T0101 (Ø 12)

M03 S1000

G00 X0 Z5

G74 X0 Z-25 Q0.1 F40

G28 U0W0

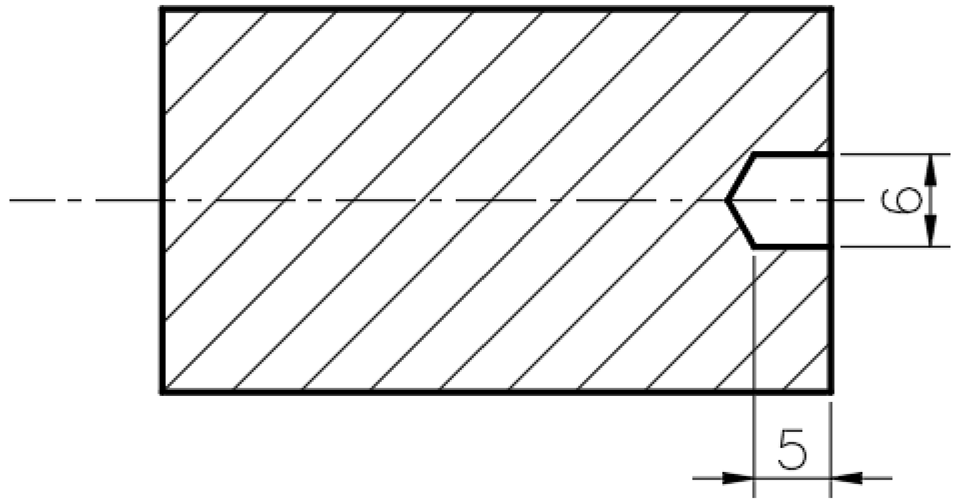
M05

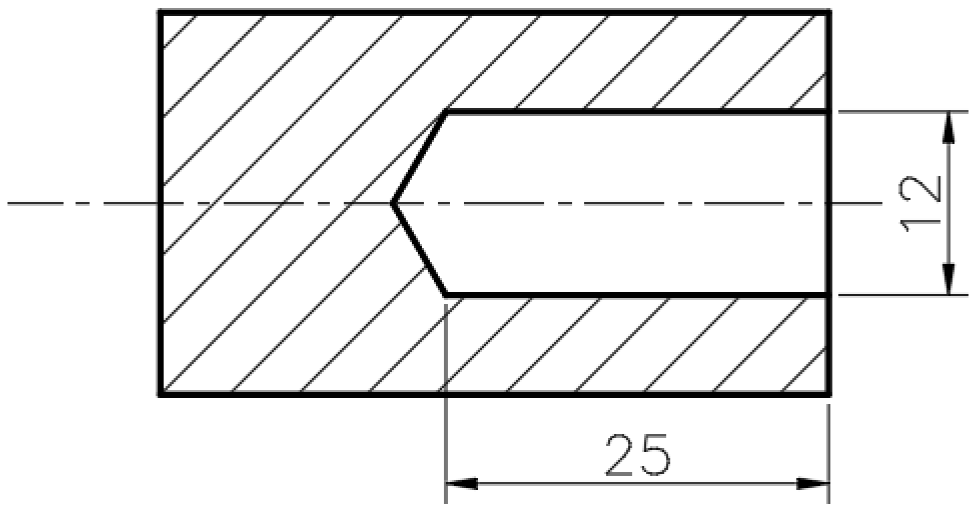
M30

For taking offset, we need to set the coordinates as follows:

Coordinates

S.No	Tool No	Cut diameter	Cut length
1	7	32+6 = 38	0
2	1	32+12 = 44	0


6
5


12
25

Experiment No 13

To perform boring operation on a given shaft using CNC lathe

AIM:
To machine the given aluminium shaft into the required shape using boring operation on a given shaft using CNC lathe.

Apparatus
CNC Lathe Machine

Specification
CNC slant Bed Lathe
Total 8 tools fixed in the turret 4 internal operation & 4 for external operation

Procedure

- **Program Selection**
  Program-edit(F1)-Program selection (F2)-program 5-EOB

- **Tool change**
  Monitor - jog - edit - MDI - give tool number i.e. M06T0303 - start - EOB

- **Tool offset**

➢ Monitor - jog - Move the tool towards the workpiece until the tool moves in and touches internal diameter, for that, press Z- and X+ (doubleclick step mode for small (0.1mm) increments) until the tool touches the internal diameter of the workpiece.
After it touches go for edit - coord - go to tool number 3 - give cut diameter values as **-12** (diameter of the hole already made using drilling).

❖ **Don't do a home start in this operation.**
❖ **No need to add workpiece diameter in offset**

➢ Monitor - jog Move the tool towards the face of the workpiece, for that, press Z- and X+ (doubleclick step mode for small (0.1mm) increments) until the tool touches the face of the workpiece.
After it touches go for edit - coord - go to tool number 3 - give cut length values as **0** - EOB - home start.

- **Running the Program**
  Monitor - Auto - SBK - Start.
  Press the start button until the red light flashes on the button.
  Deselect SBK - start (for continuous operation)

Program For Internal Boring

Workpiece obtained after performing drilling operation. Need to be fixed to perform this program.

G21 G98

G28 U0W0

M06 T0303

M03 S1200

G00 X11.5 Z5

G71 U0.2 R0.2

G71 P1 Q2 U-0.1 W0.1 F100

N1 G00 X28

G01 Z0

G01 X26 Z-1

G01 X26 Z-13

G03 X22 Z-15 R2

G02 X18 Z-17 R2

N2 G01 X12 Z-20

G70 P1 Q2

G28 U0 W0

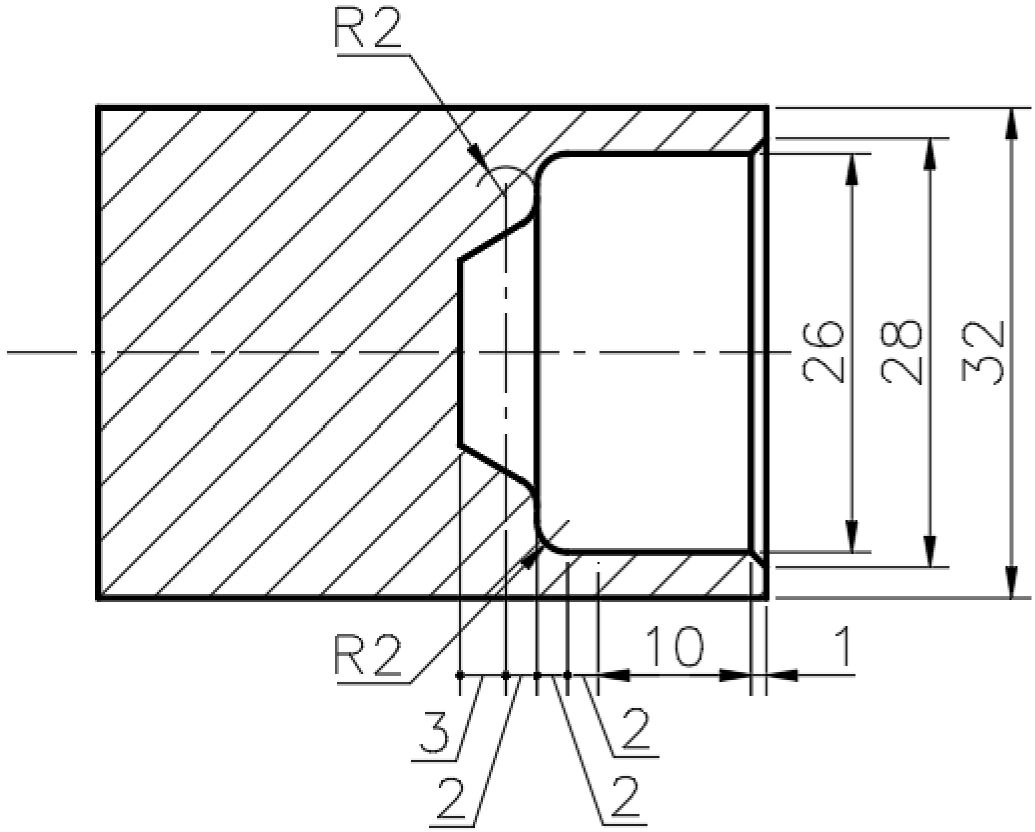
M05

M30

For taking offset, we need to set the coordinates as follows:

Coordinates

S.No	Tool No	Cut diameter	Cut length
1	3	12	0


R2
26
28
32
R2
10
1
3
2
2
2

Experiment No 14

To perform internal threading operation on a given shaft using CNC lathe

AIM:
To machine the given aluminium shaft into the required shape using internal threading operation on a given shaft using CNC lathe.

Apparatus
CNC Lathe Machine

Specification
CNC slant Bed Lathe
Total 8 tools fixed in the turret 4 internal operation & 4 for external operation

Procedure

- **Program Selection**
  Program-edit(F1)-Program selection (F2)-program 6-EOB

- **Tool change**
  Monitor - jog - edit - MDI - give tool number i.e. M06T0505 - start - EOB

- **Tool offset**

➢ Monitor - jog - Move the tool towards the workpiece until the tool touches the boring diameter (internal diameter), for that, press Z- and X+ (doubleclick step mode for small (0.1mm) increments) until the tool touches the internal diameter of the workpiece.
After it touches go for edit - coord - go to tool number 5 - give cut diameter values as **-26** (Don't do a home start).

➢ Monitor - jog Move the tool towards the face of the workpiece, for that, press Z- and X+ (doubleclick step mode for small (0.1mm) increments) until the tool touches the face of the workpiece.
After it touches go for edit - coord - go to tool number 5- give cut length values as **0** - EOB - home start.

- **Running the Program**
  Monitor - Auto - SBK - Start.
  Press the start button until the red light flashes on the button.
  Deselect SBK - start (for continuous operation)

Program For Internal Threading

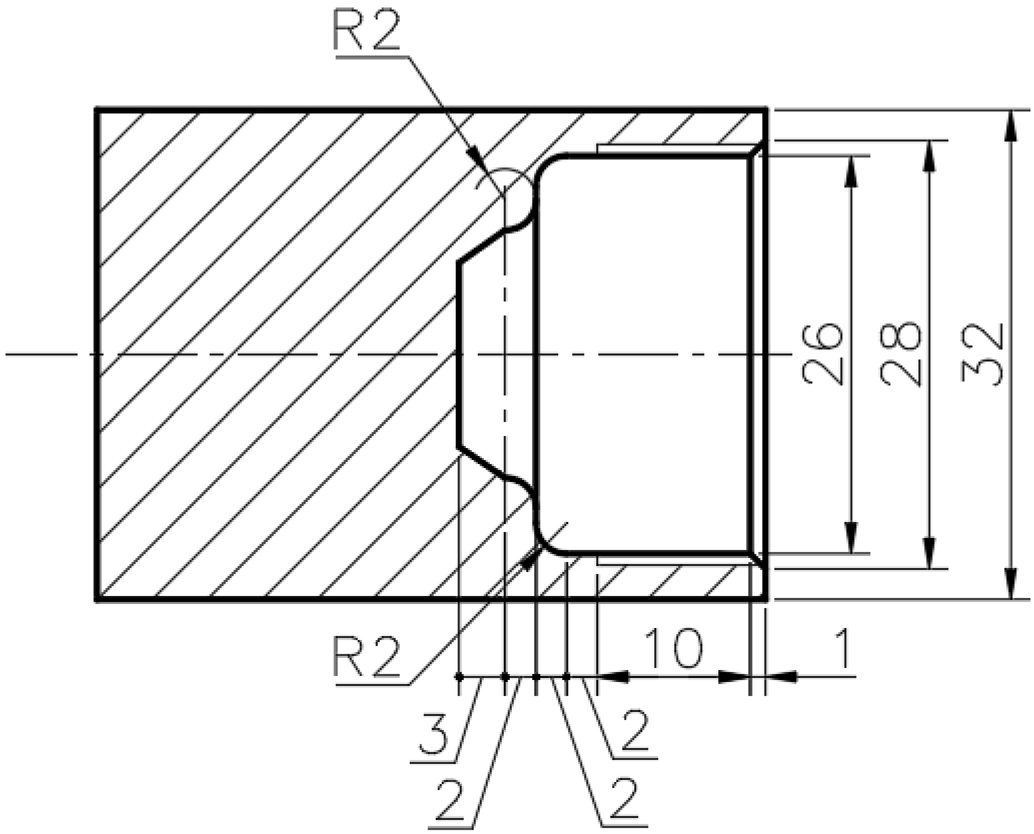
Workpiece obtained after performing a boring operation. Need to be fixed to perform this program.

```
G21 G98
G28 U0W0
M06 T0505
M03 S500
G00 X26 Z5
G76 P030060 Q0.05 R0.02
G76 X27.839 Z-10 Q0.1 P0.919 F1.5
G28 U0 W0
M05
M30
```

For taking offset, we need to set the coordinates as follows:

Coordinates

S.No	Tool No	Cut diameter	Cut length
1	5	26	0


R2
26
28
32
R2
10
1
3
2
2
2

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