AIRCRAFT ENGINEERING DRAWING USING CAD

LABORATORYMANUAL

B. TECH (IIYEAR–ISEM)

(2021-22)

Prepared by

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MALLA REDDY COLLEGE OF ENGINEERING & TECHNOLOGY

(AutonomousInstitution–UGC,Govt. of India)

Recognizedunder2(f)and12(B) ofUGCACT 1956

AffiliatedtoJNTUH,Hyderabad, ApprovedbyAICTE-AccreditedbyNBA&NAAC–'A'Grade-ISO 9001:2015 Certified)Maisammaguda, Dhulapally(PostVia.Hakimpet),Secunderabad–500100, TelanganaState,India

DEPARTMENT OF AERONAUTICAL ENGINEERING

VISION

Department of Aeronautical Engineering aims to be indispensable source in Aeronautical Engineering which has a zeal to provide the value driven platform for the students to acquire knowledge and empower themselves to shoulder higher responsibility in building a strong nation.

MISSION

a) The primary mission of the department is to promote engineering education and research.

(b) To strive consistently to provide quality education, keeping in pace with time and technology.

(c) Department passions to integrate the intellectual, spiritual, ethical and social development of the students for shaping them into dynamic engineers.

PROGRAMME EDUCATIONAL OBJECTIVES (PEO'S)

PEO1: PROFESSIONALISM & CITIZENSHIP To create and sustain a community of learning in which students acquire knowledge and learn to apply it professionally with due consideration for ethical, ecological and economic issues.
PEO2: TECHNICAL ACCOMPLISHMENTS To provide knowledge based services to satisfy the needs of society and the industry by providing hands on experience in various technologies in core field.
PEO3: INVENTION, INNOVATION AND CREATIVITY To make the students to design, experiment, analyze, interpret in the core field with the help of other multi disciplinary concepts wherever applicable.
PEO4: PROFESSIONAL DEVELOPMENT To educate the students to disseminate research findings with good soft skills and become a successful entrepreneur.
PEO5: HUMAN RESOURCE DEVELOPMENT To graduate the students in building national capabilities in technology, education and research.

PROGRAM SPECIFIC OBJECTIVES (PSO'S)

1. To mould students to become a professional with all necessary skills, personality and sound knowledge in basic and advance technological areas.
2. To promote understanding of concepts and develop ability in design manufacture and maintenance of aircraft, aerospace vehicles and associated equipment and develop application capability of the concepts sciences to engineering design and processes.
3. Understanding the current scenario in the field of aeronautics and acquire ability to apply knowledge of engineering, science and mathematics to design and conduct experiments in the field of Aeronautical Engineering.
4. To develop leadership skills in our students necessary to shape the social, intellectual, business and technical worlds.

PROGRAM OBJECTIVES (PO'S)

Engineering Graduates will be able to:

1. **Engineering knowledge**: Apply the knowledge of mathematics, science, engineering fundamentals, and an engineering specialization to the solution of complex engineering problems.
2. **Problem analysis**: Identify, formulate, review research literature, and analyze complex engineering problems reaching substantiated conclusions using first principles of mathematics, natural sciences, and engineering sciences.
3. **Design / development of solutions**: Design solutions for complex engineering problems and design system components or processes that meet the specified needs with appropriate consideration for the public health and safety, and the cultural, societal, and environmental considerations.
4. **Conduct investigations of complex problems**: Use research-based knowledge and research methods including design of experiments, analysis and interpretation of data, and synthesis of the information to provide valid conclusions.
5. **Modern tool usage**: Create, select, and apply appropriate techniques, resources, and modern engineering and IT tools including prediction and modeling to complex engineering activities with an understanding of the limitations.
6. **The engineer and society**: Apply reasoning informed by the contextual knowledge to assess societal, health, safety, legal and cultural issues and the consequent responsibilities relevant to the professional engineering practice.
7. **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.
8. **Ethics**: Apply ethical principles and commit to professional ethics and responsibilities and norms of the engineering practice.
9. **Individual and team work**: Function effectively as an individual, and as a member or leader in diverse teams, and in multidisciplinary settings.
10. **Communication**: Communicate effectively on complex engineering activities with the engineering community and with society at large, such as, being able to comprehend and write effective reports and design documentation, make effective presentations, and give and receive clear instructions.
11. **Project management and finance**: Demonstrate knowledge and understanding of the engineering and management principles and apply these to one's own work, as a member and leader in a team, to manage projects and in multi disciplinary environments.
12. **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.

MALLA REDDY COLLEGE OF ENGINEERING & TECHNOLOGY

II Year B. Tech, ANE-I Sem

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0	3	1.5

(R20A2182) AIRCRAFT ENGINEERING DRAWING LAB USING CAD

OBJECTIVES:

- To expose them to existing national standards related to technical drawings.
- It gives all the external and internal details of the machine component from which it can be manufactured. The machining symbols, tolerances, bill of material, etc. are specified on the drawing.
- The knowledge of machine drawing helps in designing the various parts of machine elements. The course content is designed in such a way that the balancing of part drawings (machine components) and assembly drawings of aircraft can be known.

Unit 1 Machine Drawing Conventions:
Need for drawing conventions – introduction to IS conventions – Conventional representation of materials, common machine elements

Unit 2 Limits and tolerances:
Limit System – Tolerances – Fits - Tolerances of Form and Position – Standards followed in Industry

Unit 3 Assembly Drawings:
Drawings of assembled views for the part drawings of the following using conventions and easy drawing proportions.
a) Engine parts – stuffing boxes, Knuckle joint, Eccentric.
b) Wing, Landing gear, horizontal stabilizer.

NOTE: First angle projection to be adopted. The student should be able to provide working drawings of actual parts

Unit 4

1. INTRODUCTION to CAD and AutoCAD – BASICS
2. 2D – FIGURES for practice USING AutoCAD (Orthographic Projection)
3. ISOMETRIC DRAWING for practice USING AutoCAD

Unit 5

1. Introduction to CREO – 3.0
2. INTRODUCTION TO CREO 3.0
3. Modeling of 3-D FIGURES USING CREO
    a. Modeling of Knuckle Joint
    b. Modeling of stuffing box

Unit 1
Machine Drawing Conventions

CONVENTIONAL REPRESENTATION:

Certain drawing conventions are used to represent materials in section and machine elements in engineering drawings.

MATERIALS:

As a variety of materials are used for machine components in engineering applications, it is preferable to have different conventions of section lining to differentiate between various materials. The recommended conventions in use are shown in Fig.1.

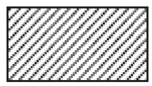
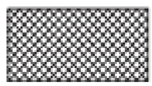
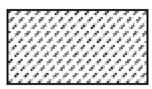
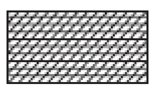
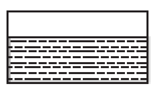
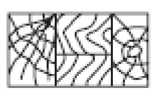
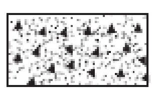
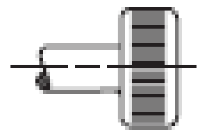
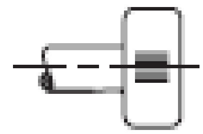
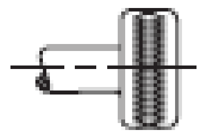
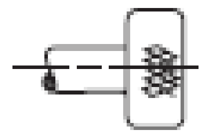
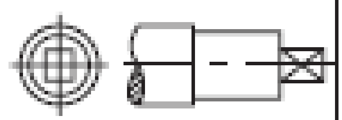
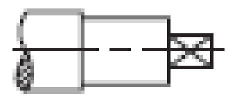
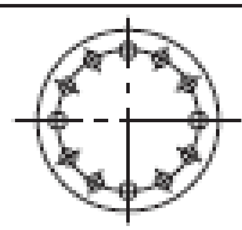
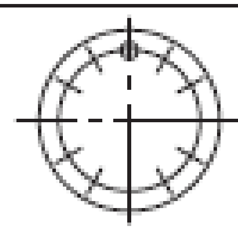
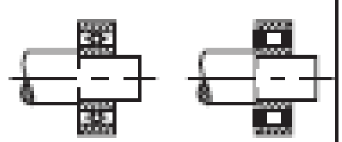
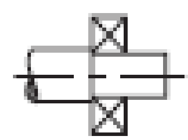
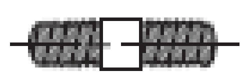
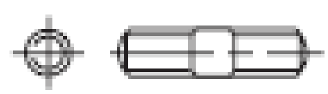
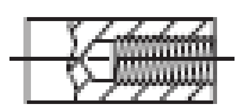
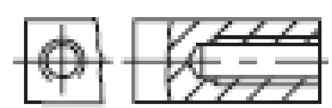
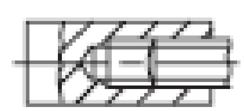
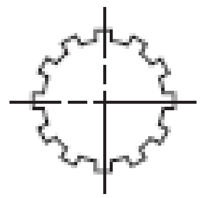
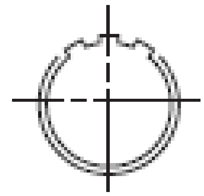
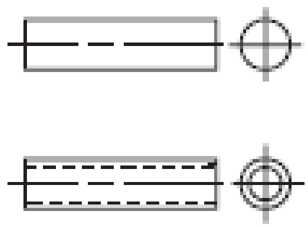
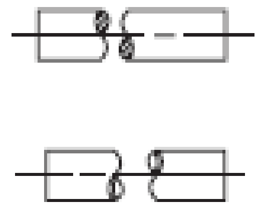
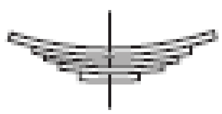
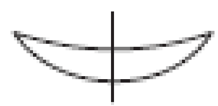
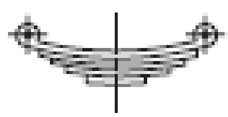
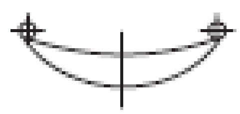
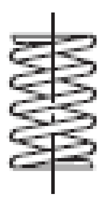
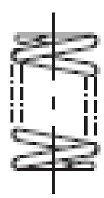
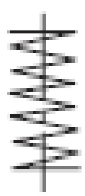
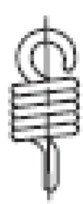
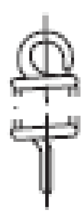
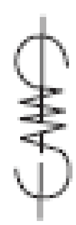
Type	Convention	Material
Metals		Steel, Cast Iron, Copper and its Alloys, Aluminium and its Alloys, etc.
		Lead, Zinc, Tin, White-metal, etc.
Glass		Glass
Packing and Insulating material		Porcelain, Stoneware, Marble, Slate, etc.
		Asbestos, Fibre, Felt, Synthetic resin products, Paper, Cork, Linoleum, Rubber, Leather, Wax, Insulating and Filling materials, etc.
Liquids		Water, Oil, Petrol, Kerosene, etc.
Wood		Wood, Plywood, etc.
Concrete		A mixture of Cement, Sand and Gravel

Fig. 1: Conventional representation of Materials

MACHINE COMPONENTS:

When the drawing of a component in its true projection involves a lot of time, its convention may be used to represent the actual component. Figure 2 shows typical examples of conventional representation of various machine components used in engineering drawing.

Title	Subject	Convention
Straight knurling		
Diamond knurling		
Square on shaft		
Holes on circular pitch		
Bearings		
External screw threads (Detail)		
Internal screw threads (Detail)		
Screw threads (Assembly)		

Title	Subject	Convention	
Splined shafts			
Interrupted views			
Semi-elliptic leaf spring			
Semi-elliptic leaf spring with eyes			
	Subject	Convention	Diagrammatic Representation
Cylindrical compression spring			
Cylindrical tension spring			

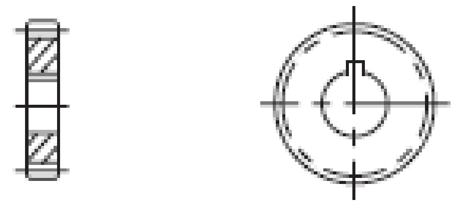
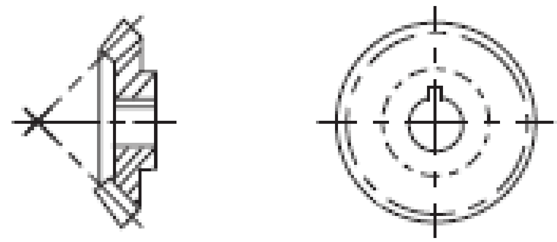
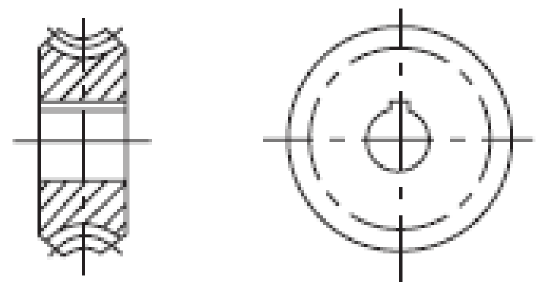
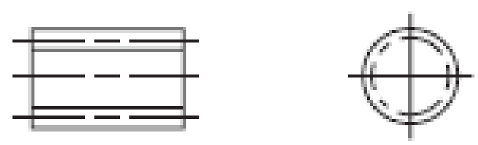
Title	Convention
Spur gear	
Bevel gear	
Worm wheel	
Worm	

Fig. 2: Conventional representation of machine components

UNIT 2
LIMITS AND TOLERANCES

LIMIT SYSTEM

Following are some of the terms used in the limit system:

TOLERANCE

The permissible variation of a size is called tolerance. It is the difference between the maximum and minimum permissible limits of the given size. If the variation is provided on one side of the basic size, it is termed as unilateral tolerance. Similarly, if the variation is provided on both sides of the basic size, it is known as bilateral tolerance.

LIMITS

The two extreme permissible sizes between which the actual size is contained are called limits. The maximum size is called the upper limit and the minimum size is called the lower limit.

DEVIATION

It is the algebraic difference between a size (actual, maximum, etc.) and the corresponding basic size.

ACTUAL DEVIATION:

It is the algebraic difference between the actual size and the corresponding basic size.

UPPER DEVIATION:

It is the algebraic difference between the maximum limit of the size and the corresponding basic size.

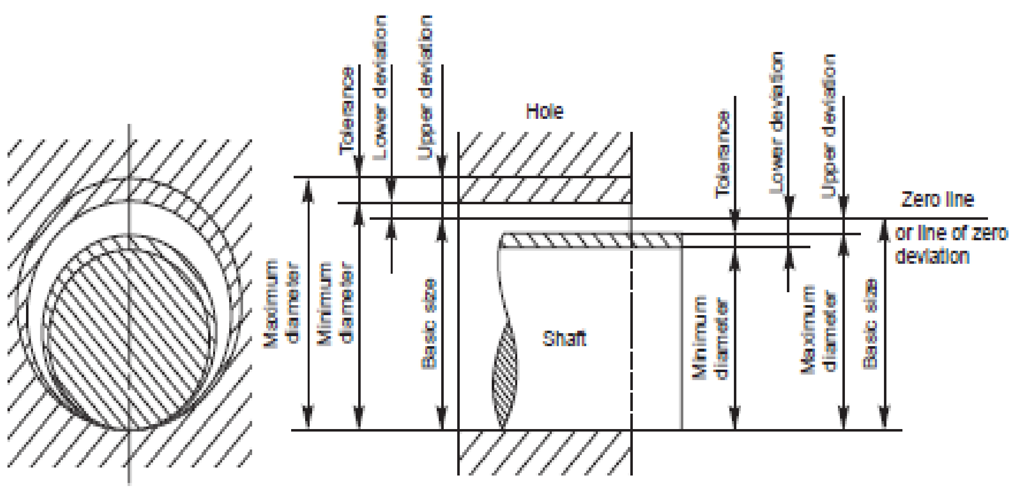


Fig. 3: Diagram illustrating basic size deviations and tolerances

LOWER DEVIATION:

It is the algebraic difference between the minimum limit of the size and the corresponding basic size.

ALLOWANCE

It is the dimensional difference between the maximum material limits of the mating parts, intentionally provided to obtain the desired class of fit. If the allowance is positive, it will result in minimum clearance between the mating parts and if the allowance is negative, it will result in maximum interference.

BASIC SIZE

It is determined solely from design calculations. If the strength and stiffness requirements need a 50mm diameter shaft, then 50mm is the basic shaft size. If it has to fit into a hole, then 50 mm is the basic size of the hole. Figure 3 illustrates the basic size, deviations and tolerances.
Here, the two limit dimensions of the shaft are deviating in the negative direction with respect to the basic size and those of the hole in the positive direction. The line corresponding to the basic size is called the zero line or line of zero deviation.

DESIGN SIZE

It is that size, from which the limits of size are derived by the application of tolerances. If there is no allowance, the design size is the same as the basic size. If an allowance of 0.05 mm for clearance is applied, say to a shaft of 50 mm diameter, then its design size is (50 – 0.05) = 49.95 mm. A tolerance is then applied to this dimension.

ACTUAL SIZE

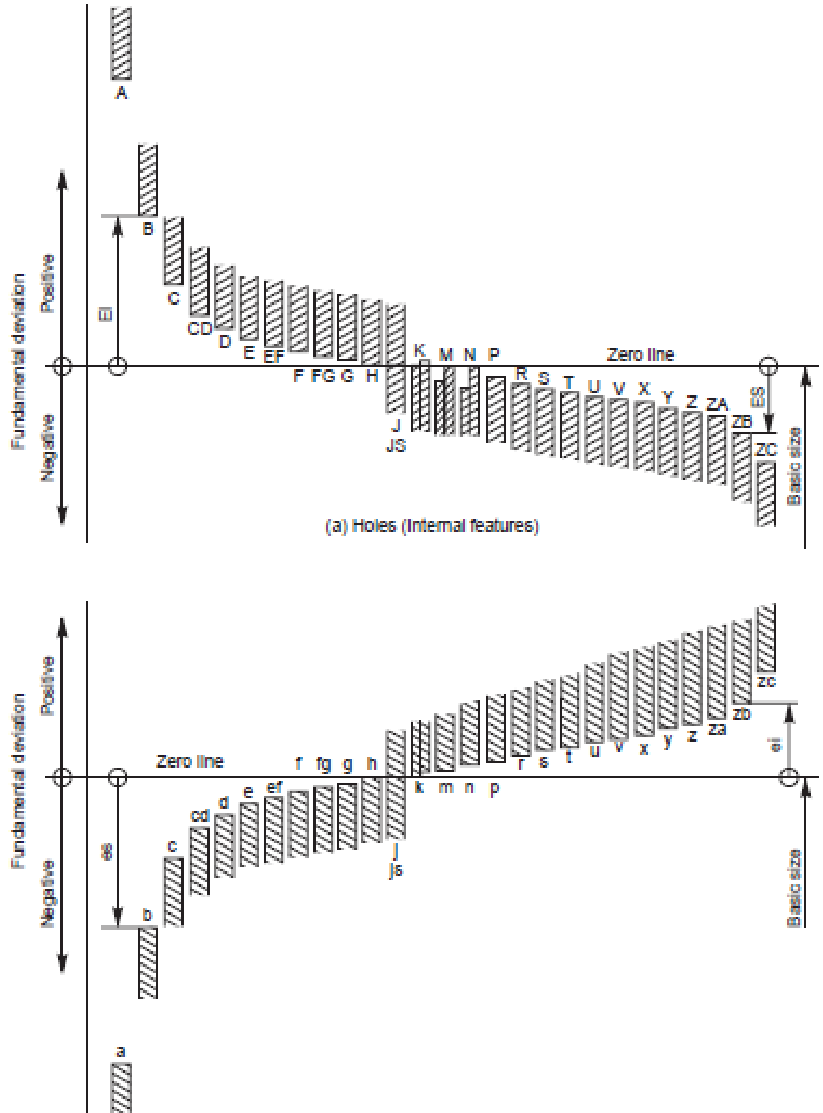
It is the size obtained after manufacture.

TOLERANCES:

Great care and judgement must be exercised in deciding the tolerances which may be applied on various dimensions of a component. If tolerances are to be minimum, that is, if the accuracy requirements are severe, the cost of production increases. In fact, the actual specified tolerances dictate the method of manufacture. Hence, maximum possible tolerances must be recommended wherever possible.
Figure 4 shows the tolerances (in microns or in micrometers) that may be obtained by various manufacturing processes and the corresponding grade number.

FUNDAMENTAL TOLERANCES:

Tolerance is denoted by two symbols, a letter symbol and a number symbol, called the grade. Fig. 4 shows the graphical illustration of tolerance sizes or fundamental deviations for letter symbols and Table 1 lists the fundamental tolerances of various grades. It may be seen from Fig. 4 that the letter symbols range from **A** to **ZC** for holes and from **a** to **zc** for shafts. The letters **I, L, O, Q, W** and **i, l, o, q, w** have not been used. It is also evident that these letter symbols represent the degree of closeness of the tolerance zone (positive or negative) to the basic size.
Similarly, it can be seen from Table 1 that the basic sizes from l mm to 500 mm have been subdivided into 13 steps or ranges. For each nominal step, there are 18 grades of tolerances, designated as IT 01, IT 0 to IT 1 to IT 16, known as “Fundamental tolerances”.
The fundamental tolerance is a function of the nominal size and its unit is given by the empirical relation, standard tolerance unit, $i = 0.45 \times \sqrt[3]{D} + 0.001\,D$
where *i* is in microns and D is the geometrical mean of the limiting values of the basic steps mentioned above, in millimeters. This relation is valid for grades 5 to 16 and nominal sizes from 3 to 500 mm. For grades below 5 and for sizes above 500 mm, there are other empirical relations for which it is advised to refer IS: 1919–1963. Table 1A gives the relation between different grades of tolerances and standard tolerance unit i.



(a) Holes (Internal features)

(b) Shafts (external features)

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	$7i$	$10i$	$16i$	$25i$	$40i$	$64i$	$100i$	$160i$	$250i$	$400i$	$640i$	$1000i$

Table 1: Relative magnitude of IT tolerances for grades 5 to 16 in terms of tolerance unit *i* for sizes upto 500 mm

FUNDAMENTAL DEVIATIONS:

The symbols used (Fig. 15.3) for the fundamental deviations for the shaft and hole are as follows :

	Hole	*Shaft*
Upper deviation (E′ cart superior)	ES	es
Lower deviation (E′ cart inferior)	EI	ei

Table 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

Fundamental deviations for shafts of types **a** to **k** of sizes upto 500mm

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											

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
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											

Fundamental deviations for shafts of types **m** to **zc** of sizes upto 500 mm

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

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*													
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

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
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																

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						

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*
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						

For each letter symbol from **a** to **zc** for shafts and **A** to **ZC** for holes; the magnitude and size of one of the two deviations may be obtained from the above tables and the other deviation is calculated from the following relationship

Shafts, ei = es – IT

Holes, EI = ES – IT

FITS:

The relation between 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.

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. 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.

TRANSITION FIT:

This fit may result in either interference or a clearance, depending upon the actual values of the tolerance of individual parts. The shaft in Fig. 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).

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. The shaft in Fig. is larger than the hole, so it requires a press fit, which has an effect similar to welding of two parts. The value of minimum interference is 30.25 – 30.30 = – 0.05 mm and maximum interference is 30.15 – 30.40 = – 0.25 mm.

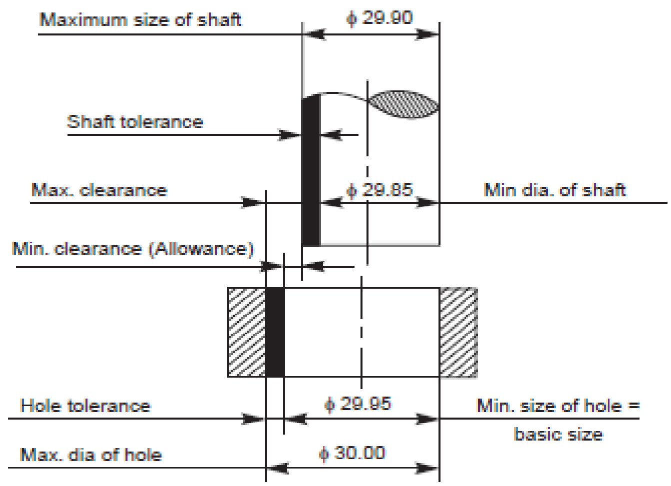


Fig. 15.10 Clearance fit

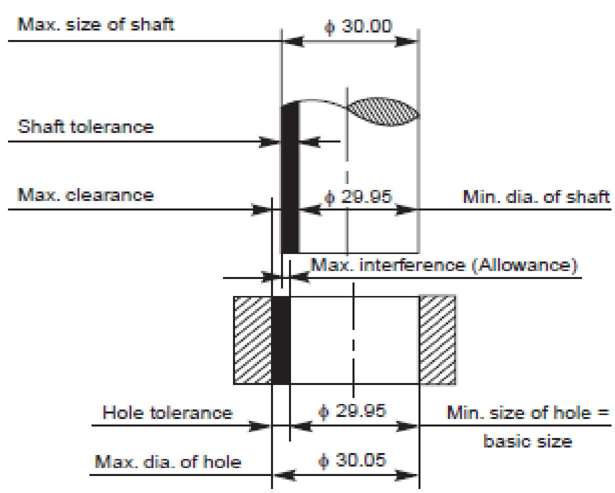


Fig. 15.11 Transition fit

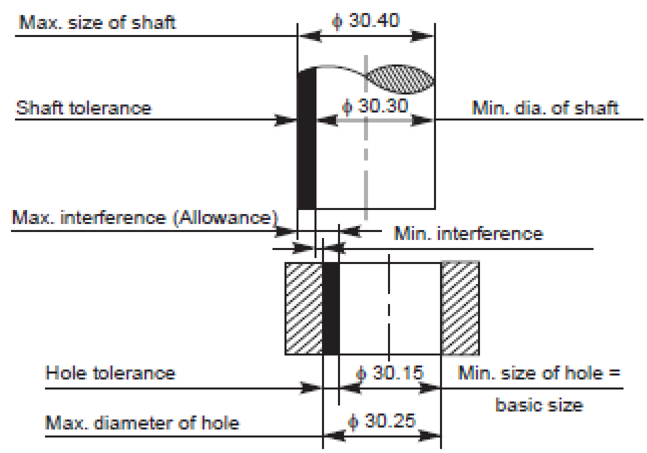
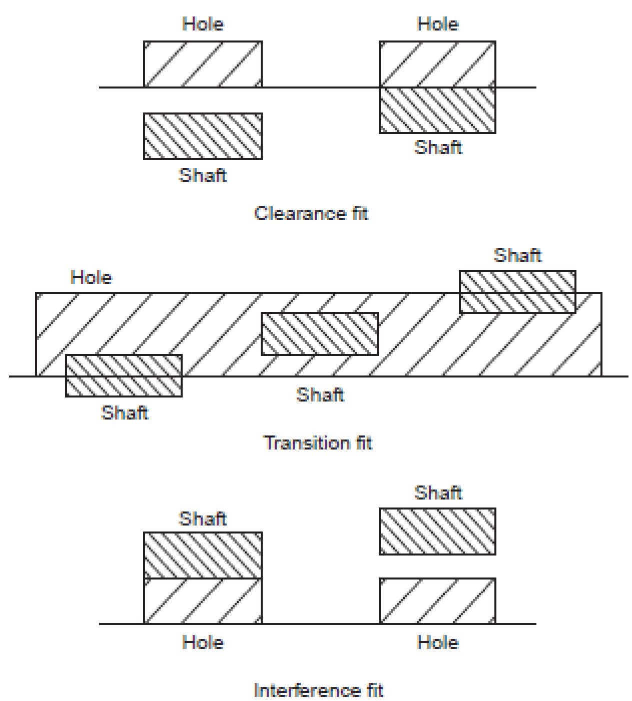


Fig. 15.12 Interference fit

SCHEMATIC REPRESENTATION OF FITS:



HOLE BASIS AND SHAFT BASIS SYSTEMS:

In working out limit dimensions for the three classes of fits; two systems are in use, *viz.*, the hole basis system and shaft basis system.

HOLE BASIS SYSTEM

In this system, the size of the shaft is obtained by subtracting the allowance from the basic size of the hole. This gives the design size of the sha ances are then applied to each part separately. In this system, the lower deviation of the hole is zero. The letter symbol for this situation is '*H*'. The hole basis system is preferred in most cases, since standard tools like drills, reamers, broaches, etc., are used for making a hole.

SHAFT BASIS SYSTEM

In this system, the size of the hole is obtained by adding the allowance to the basic size of the shaft. This gives the design size for the hole. Tolerances are then applied to each part. In this system, the upper deviation of the shaft is zero. The letter symbol for this situation is '*h*'. The shaft basis system is preferred by (*i*) industries using semi-finished shafting as raw materials, *e.g.*, textile industries, where spindles of same size are used as cold-finished shafting and (*ii*) when several parts having different fits but one nominal size is required on a single shaft.

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

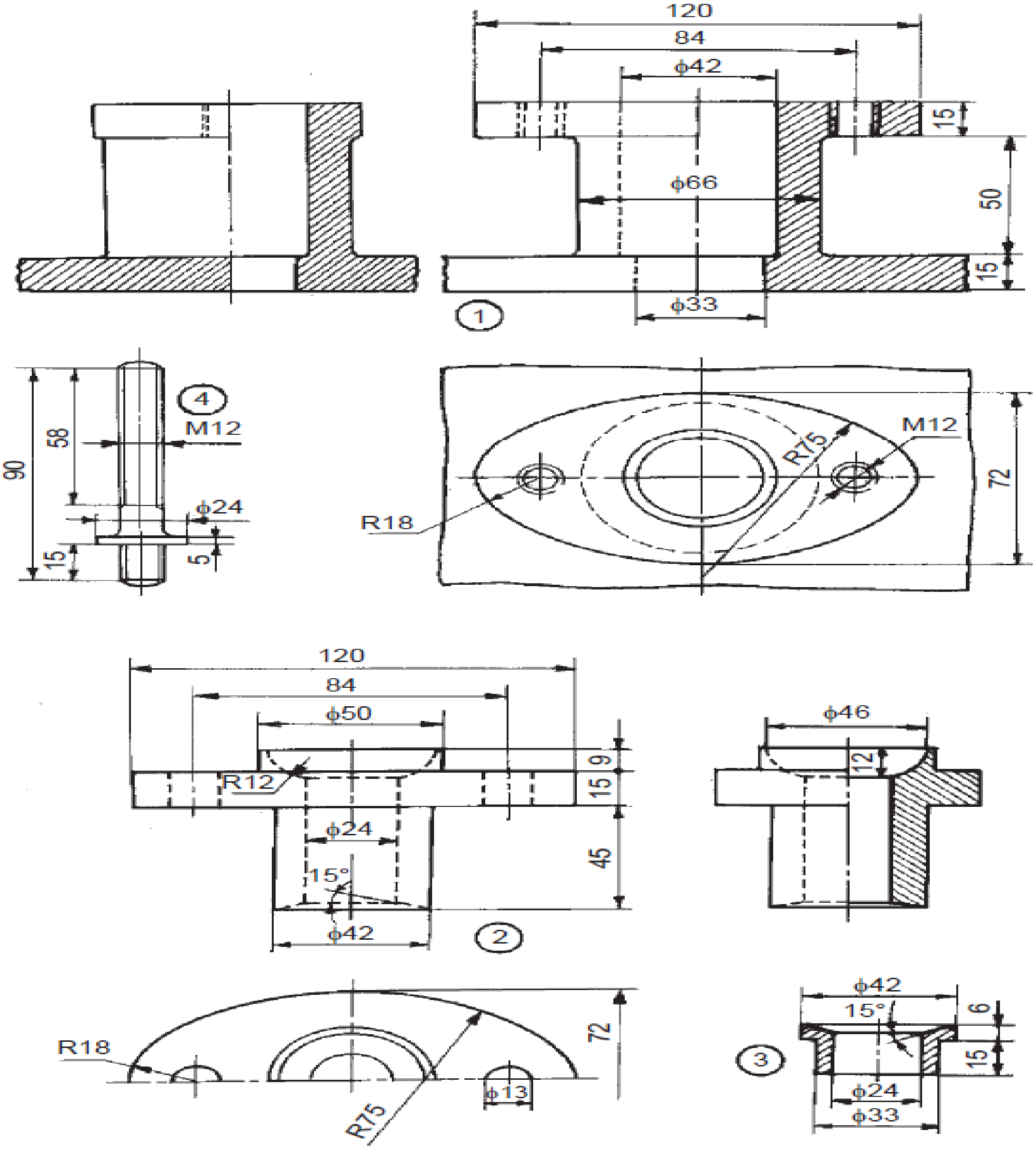
Types of fits with symbols and applications:

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.

UNIT 3
ASSEMBLY DRAWINGS

ENGINE PARTS

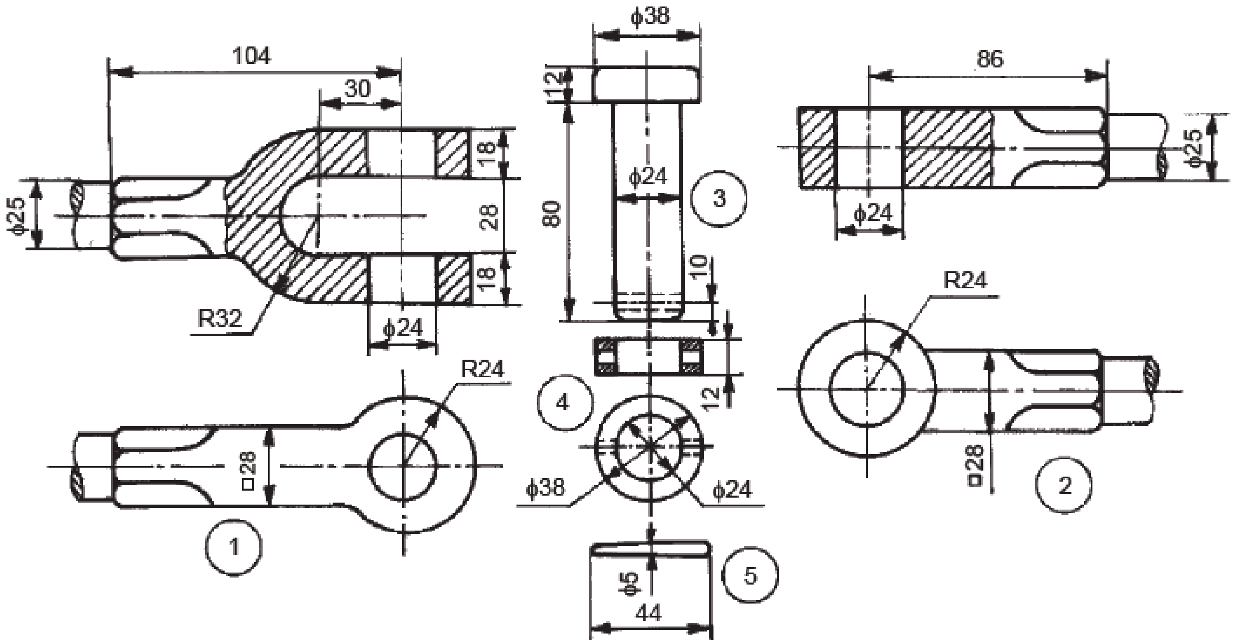
Stuffing Box



Parts list

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

Knuckle Joint:



Parts list

Sl. No.	Name	Matl.	Qty.
1	Fork end	Forged steel	1
2	Eye end	Forged steel	1
3	Pin	Mild steel	1
4	Collar	Mild steel	1
5	Taper pin	Mild steel	1

UNIT 4

INTRODUCTION to CAD and AutoCAD

4.1 INTRODUCTION

Computer Aided Drafting is a process of preparing a drawing of an object on the screen of a computer. There are various types of drawings in different fields of engineering and sciences. In the fields of mechanical or aeronautical engineering, the drawings of machine components and the layouts of them are prepared. In the field of civil engineering, plans and layouts of the buildings are prepared. In the field of electrical engineering, the layouts of power distribution system are prepared. In all fields of engineering use of computer is made for drawing and drafting.

The use of CAD process provides enhanced graphics capabilities which allows any designer to

- Conceptualize his ideas
- Modify the design very easily
- Perform animation
- Make design calculations
- Use colors, fonts and other aesthetic features.

REASONS FOR IMPLEMENTING A CAD SYSTEM

1. **Increases the productivity of the designer**: CAD improves the productivity of the designer to visualize the product and its component, parts and reduces the time required in synthesizing, analyzing and documenting the design
2. **Improves the quality of the design**: CAD system improves the quality of the design. CAD system permits a more detailed engineering analysis and a larger number of design alternatives can be investigated. The design errors are also reduced because of the greater accuracy provided by the system
3. **Improves communication:** It improves the communication in design. The use of a CAD system provides better engineering drawings, more standardization in the drawing, and better documentation of the design, few drawing errors and legibility.
4. **Create data base for manufacturing:** In the process of creating the documentation for these products, much of the required data base to manufacture the products is also created.
5. **Improves the efficiency of the design:** It improves the efficiency of the design process and the wastage at the design stage can be reduced.

APPLICATION OF CAD

There are various processes which can be performed by use of computer in the drafting process.

1. **Automated Drafting**: This involves the creation of hard copy engineering drawings directly from CAD data base. Drafting also includes features like automatic dimensioning, generation of cross – hatched areas, scaling of the drawing and the capability to develop sectional views

and enlarged views in detail. It has ability to perform transformations of images and prepare 3D drawings like isometric views, perspective views etc.,

2. **Geometric Modeling**: Concerned with the computer compatible mathematical description of the geometry of an object. The mathematical description allows the image of an object to be displayed and manipulated on a graphics terminal through signals from the CPU of the CAD system. The software that provides geometric modeling capabilities must be designed for efficient use both by computer and the human designer.

BENEFITS OF CAD

The implementation of the CAD system provides variety of benefits to the industries in design and production as given below:

1. Improved productivity in drafting
2. Shorter preparation time for drawing
3. Reduced man power requirement
4. Customer modifications in drawing are easier
5. More efficient operation in drafting
6. Low wastage in drafting
7. Minimized transcription errors in drawing
8. Improved accuracy of drawing
9. Assistance in preparation of documentation
10. Better designs can be evolved
11. Revisions are possible
12. Colors can be used to customize the product
13. Production of orthographic projections with dimensions and tolerances
14. Hatching of all sections with different filling patterns
15. Preparation of assembly or sub assembly drawings
16. Preparation of part list
17. Machining and tolerance symbols at the required surfaces
18. Hydraulic and pneumatic circuit diagrams with symbols
19. Printing can be done to any scale

LIMITATIONS OF CAD

1. 32 – bit word computer is necessary because of large amount of computer memory and time
2. The size of the software package is large
3. Skill and judgment are required to prepare the drawing
4. Huge investment

AutoCAD - INTERFACE

The topmost part of the window is called title block. In the new version it has been enhanced to hold some additional options apart from Filename & Software Version details. The large button on the top left gives File Menu. There is some file handling commands near it. On the top right there are usual windows options to minimize, maximize or close the window. In the middle there are options for help.

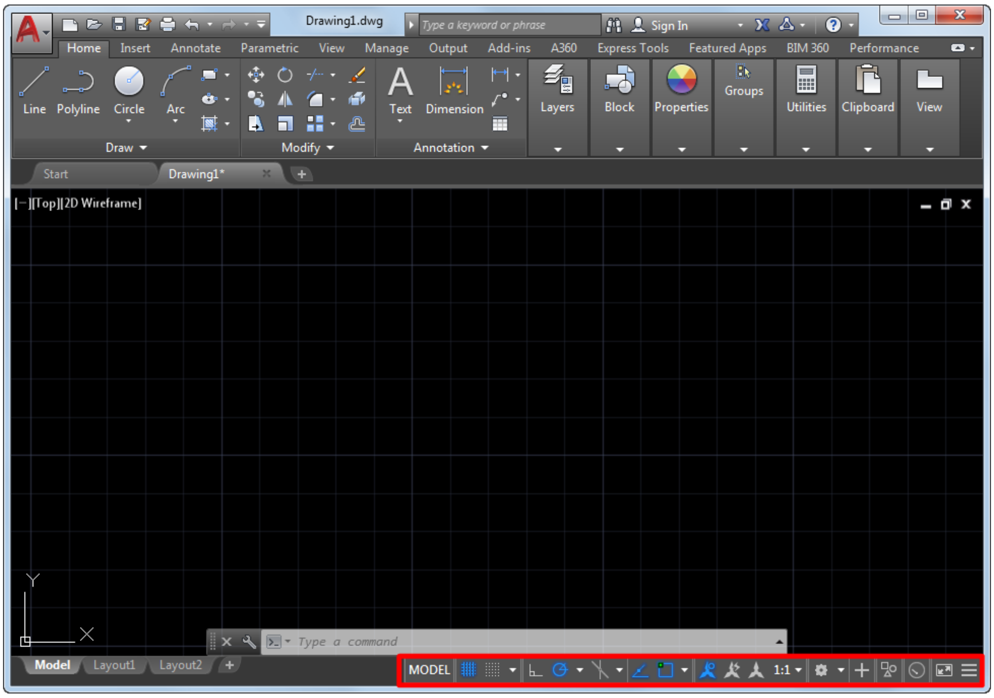
Note: For information on a particular command just rest the cursor on the button for a while. Or press F1 when command is active.

The bottommost line is called status bar. It holds tabs for model space (which is a space for working out the drawings) and layout space (Which is for setting the drawing on paper). Then there are some drawing aids (like snap, grid, object snap, ortho etc…) which help in creation of the drawing.

Just above the status bar is the command line which is where all the proceedings of the commands appear. It is suggested that you always keep observing command-line to follow instructions given to avoid any problems.

The black screen in the middle is model space which is where the result of our work is displayed. At the left bottom of model space is UCS - User Coordinate System. UCS stays at the origin when the origin is visible on screen; else it stays at the lower left corner.

Then there is the plotter which is controlled by mouse & which does all the work in AutoCAD. There are cross hair pointing along the x, y & z directions & at the middle there is a box known as pick-box. Its function is to select objects on which it overlaps. In drawing mode this is replaced by another such box known as Aperture. Aperture's function is to highlight snap points in drawing mode.



DEVICES

Keyboard & Mouse are the most widely used devices for operation AutoCAD.

Keyboard is the fastest way of giving a command. A few things to note:

1. Space bar works just like enter in most of the cases.
2. Every command has a shortcut which can be typed to activate that command.
3. ESC button can be used to come out of any command.
4. Pressing space bar when no command is active selects the most recently used command.
5. Previous commands can be browsed using arrow keys.
6. The basic windows functions like cut (ctrl+x), copy (ctrl+c) etc can be used here as well.

Mouse is the fastest way to select objects & points in the model space and can also be used to activate commands. A few notable points:

1. Left click is used for selection of point or object as the case may be.
2. Clicking in the empty space activates a window for selecting objects. Left to right is blue & selects objects completely inside. While Right to Left is green this selects objects on boundary as well.
3. Clicking & dragging results in an area selection mode with irregular boundary.
4. Right click is sometimes used as enter. When no command is active it gives a quick menu for previous commands & other navigational options.
5. The Scroll Button when rotated zoom the window. Whereas when pressed, it pans the window. Double clicking this button zooms to extents.
6. Shift + Scroll Button is used for 3D orbit, i.e. it rotated the model space in 3D.
7. Ctrl + Scroll Button is used for scrolling in 2 directions.
8. Shift + Right Click displays a menu with all object snap points from which temporary single object snap can be chosen. (Useful for selecting mid of selected points).

NAVIGATION

Moving around the model space is easy when the mouse is fully functional. However in cases when there are problems with the mouse following points will be helpful.

COORDINATE SYSTEMS

Absolute: In this method all the points are measured from the fixed origin. The coordinates can be entered directly separated by commas like {x,y,z} (without space).

Relative: In this the next point is measured relative to the previous point taking it as origin. For using this method we need to type {@x,y,z}. Just typing @ will reselect the previous point. Negative values can be used to get to the opposite side.

Polar: In this method the points are represented as radial distance & angle from positive x axis. Mostly it is used in relative format. So we need to enter it in the format - {@r<theta}.

CAD SOFTWARES

The software is an interpreter or translator which allows the user to perform specific type of application or job related to CAD. The following software's are available for drafting

1. AUTOCAD
2. CRO
3. CATIA
4. SOLID WORKS
5. NX UNIGRAPHICS
6. FUSION 360
7. INVENTOR
8. SOLID EDGE

The above software’s are used depending upon their application.

AutoCAD

Auto CAD package is suitable for accurate and perfect drawings of engineering designs. The drawing of machine parts, isometric views and assembly drawings are possible in AutoCAD. The package is suitable for 2D and 3D drawings.

4.2. AutoCAD – BASICS

4.1 STARTING WITH ACAD

CAD uses four basic elements for preparation of any drawing:

1. Line
2. Curves
3. Text
4. Filling point.

Computer Aided Drafting is done by the operator by placing the mouse pointer by placing the mouse pointer at the desired location and then executing the command to draw the graphic elements using different methods.

Advanced computer aided drafting packages utilize four areas on the screen.

1. Drawing Area
2. Command Area
3. Menu Area
4. Tool boxes.

4.2 DRAWING ENVIRONMENT

ACAD provides two drawing environments for creating and laying out the drawing.

i. Model Space
ii. Paper Space.

ACAD allows creating drawing, called a model, in full scale in an area known as model space without regard to the final layout or size when the drawing is plotted on the paper.

In the space opened for the first time, it is possible to create floating viewports to contain different views of the model. In the paper space, floating viewports are treated as objects which can be moved and resized in order to create a suitable layout.

a. Limits: This sets and controls the drawing boundaries.

At the command prompt, enter **limits**

ON/OFF/<LOWER LEFT CORNER> <current>: Specify a point, enter on or off, or Press Enter.

LTSCALE

This sets the line type scale factor. Use LTSCALE to change the relative length of the dash – dot line types per drawing unit
At the Command prompt, enter **ltscale**

New scale factor <current>: Enter a positive real value or press enter
Changing the line type scale factor causes the drawing to regenerate.

b. Units: The format for display co–ordinates and measurement can be selected according to the requirement.
Several measurement styles are available in ACAD. The main methods are engineering and architectural, having specific base unit assigned to them.

i. Decimal: select to enter and display measurements in decimal notation
ii. Engineering: Display measurements in feet and decimal inches.
iii. Architectural: Display measurements in feet, inches and fractional inches
iv. Fractional: Display measurements in mixed numbers notation
v. Scientific: Display measurements in scientific notation.

The precision that is specified controls the number of decimal places or fractional size to which we want linear measurements displayed.

c. Measure: This places point objects or blocks at measured intervals on an object. At the command Prompt, enter **measure**

Select object to measure: Use an object selection method <segment length> / Block: Specify a distance.

d. Angles: Select the format in which we want to enter and display angles.

i. Decimal Degrees: Display partial degrees as decimals
ii. Deg/Min/Sec: Display partial degrees as minutes and seconds.
iii. Grades: Display Angles as grades
iv. Radians: Display angles as radians.
v. Surveyor: Displays angles in surveyor units.

e. Angle measure: Select the direction of the zero angle for the entry of angles:
i. East: Select to specify the compass direction east as the zero angle.
ii. North: Select to specify the compass direction north as the zero angle.
iii. West: Select to specify the compass direction west as the zero angle.
iv. South: Select to specify the compass direction south as the zero angle.
v. Other: Select to specify a direction different from the points of the compass as the zero angle.

f. Area: Enter the approximate width and length which is planned to draw in full scale units. This limits the area of the drawing covered by grid dots when the grid is turned on. It also adjusts several default settings, such as text height, line type scaling and snap distance to convenient values. It is possible to adjust these settings.

g. Title block: Select the description of an ACAD drawing file of a title block to insert as a symbol in the new drawing. It can add or remove drawing files of title blocks from the list with the Add or Remove buttons

h. Layout: Paper space is often used to create complex multiple view drawings. There are three types of paper spaces:
1. Work on the drawing while viewing the layout.
2. Work on the drawing without the layout visible
3. Work on the layout of the drawing.

The following procedure is used for this purpose

1. From the File menu or from the standard tool bar, choose New
2. In the startup dialog box, choose Use a wizard, and select Advanced wizard
3. Choose OK
4. In the Advanced Setup Dialog box, select Title Block.
5. Select Title Block Description and Title Block file Name from the lists and then choose Add.
6. In the Select Title Block File dialog box, Select a title block, then choose open
7. In the Advanced Setup dialog box, a sample of that title is displayed.
8. Choose Done.

i. Co-Ordinate System: The co- ordinate system can be modified in the AutoCAD. There are two types of co- Ordinate systems used.
The WCS (World co- ordinate system) is a universal system in which its origin is at the fixed position.

The UCS (User co- ordinate system) is a system in which User can fix his origin at any point.

1. UCS : This manages user co- ordinate systems

At the command prompt enter **ucs**

Origin / z axis/ 3 point/ object/ view/ X/Y/Z / Prev/ Restore/Save/ Del/?/< world>: enter an option or press enter

2. WCS: This manages world co- ordinate system

j. **Layers** (LA): These are like transparent sheets of paper.

	Display	***Create***	***Modify***
Off	No	Yes	Yes
Freeze	No	No	No
Lock	Yes	Yes	No

Note: Current layer cannot be frozen.

2.3 ELEMENTS OF DRAWING

2.3.1 BASIC GEOMETRIC COMMANDS

Line: A line is specified by giving its two end points or first point and the distance of line along with its angle of inclination. A line can be drawn by using two commands.

Command: **line**

Specify first point: Specify a point (1)

Specify next point or [Undo]: Specify a point (2)

The second point can be indicated by **@d<a**

Where **d** is the distance of line and **a** is the angle of inclination in degrees.

Pline: This is a poly line which allows continuous segment of the line and it is drawn similar to the line command. The polyline allows to change the thickness of the line according to the requirement.

From the Draw tool bar choose the Polyline flyout.

Draw pull down menu: **Polyline**

At the command prompt, enter **pline**

Curves: Following are the various types of curves used in the drawings:

i. Circle

ii. Ellipse

iii. Arc

iv. Regular or any other type.

i. Circle: The circle can be drawn by using two types of commands

a. Circle

b. Donut

a. Circle: This command draws the circle by using four methods:

Center point and radius

Two point circle

Three point circle

Tangent circle

At the command prompt, enter **circle**

Specify center point for circle or [3P (Three Points)/2P (Two Points)/Ttr]: Specify a point or enter an option

b. Donut: This draws filled circles and rings.

Donuts are constructed of a closed polyline composed of wide arc segments.

At the command prompt, enter **donut**

Specify inside diameter of donut <current>: Specify a distance or press ENTER

If you specify an inside diameter of 0, the donut is a filled circle.
Specify outside diameter of donut <current>: Specify a distance or press ENTER
Specify center of donut or <exit>: Specify a point (1) or press ENTER to end the command

ii. Ellipse: Creates an ellipse or an elliptic arc. It is a curve having major and minor axis with a center.

The ellipse can be prepared by four methods.

Axis endpoint
Arc
Centre
Iso circle

Axis end point: Defines the first axis by two specified endpoints. The angle of the first axis determines the angle of the ellipse. The first axis can define either the major or the minor axis of the ellipse.

Arc: Creates an elliptical arc. The angle of the first axis determines the angle of the elliptical arc. The first axis can define either the major or the minor axis of the elliptical arc.

Center: Creates the ellipse by a specified center point.

Isocircle: Creates an isometric circle in the current isometric drawing plane.

At the command prompt, enter **ellipse.**

iii. Arc: The arc is a curve specified by center and radius as well as the start angle and end angle. There are seven method used for drawing an arc.

1. Three point method
2. Start point-center point –end point
3. Start point-center point-length of chord
4. Start point-end point –angle of inclusion
5. Start point-end point-direction
6. Start point-center point-angle of inclusion
7. Start point-end point-radius

These methods can be used by executing the arc command

- **Arc:** creates an arc.

At the command prompt, enter arc

Center/<start point>: specify a point, enter c, or press enter

- **Polyarc:** the second method of the drawing the arc is poly arc by use of pline command. This command allows drawing of filled arc of any width it also allows for drawing of a regular or irregular curve.

Rectangle: A rectangle can be drawn by LINE command or by Rectangle command. The **PLINE** command also allows for drawing of hollow or filled rectangle. A **SOLID** command is also used for drawing of filled rectangles.

1. **Rectangles:** draws a rectangular polyline

At the command prompt, enter **rectangle**

First corner: specify point (1)

Other corner: specify point (2)

2. **Solid:** creates solid –filled polygons .solids are filled only when fill system

Variable is set to on view is set to plan.

At the command prompt, enter **solid**

First corner: specify point (1)

Other corner: specify point (2)

The first two points define one edge of the polygon.

Third point: specify a point (3) diagonally opposite the second

Forth point: specify a point (4) or press enter

Polygon: Creates an equilateral closed polyline .A polygon is a polyline object. AUTOCAD draws polyline with zero width and no tangent information.

At the command prompt enter **polygon**

Number of sides <**current**>: enter a value between **3 and 1024** or press enter

Edge/<center of polygon>: **specify a point (1) or enter**.

Point: Creates a point object .points can act as nodes to which you can snap objects. You can specify a full 3D location for a point.

At the command prompt, enter **point**

Point: specify a point

Array: This creates multiple copies of objects in pattern.
Each object in an array can be manipulated independently.
At the command prompt enter, **array** Rectangular or polar array<current>: enter an option or press enter specify a point

2.3.2 EDITING COMMANDS

1. Erasing Of Object: The object can be removed or erased by use of erase command ERASE. This removes object from drawing.

At the command prompt, enter **erase**

Select objects: use an object selection method.

2. Coloring Of Object:

The object can be drawn with any variety of color which ranges from 0 to 256.

The setting of color can be done by color command

Hatch: This fills an area with a pattern. HATCH fills the specified hatch boundary with non-associative hatch.
A non –associative hatch is not updated when its boundaries are modified. A hatch Boundary consists of an object or objects that completely enclose an area at the command prompt, enter hatch Pattern (? Or name/ U, style) <current>: enter a predefined pattern name, enter u, enter? Or press enter.

3. Scaling Of Drawing: zoom command displays the object at a specified scale factor. The value entered is relative to the limits of the drawing. for example, entering 2 doubles the apparent display size of any objects from what it would be if it were zoomed to the limits of the drawing
If you enter a value followed by xp, auto CAD specifies the scale relative to paper scale units. For example, entering 0.5xp displays model space at half the scale of paper space unit's. The following illustration shows a number of viewports arranged in paper space. The view in each view port is scaled relative to paper space. The first view is scaled 1=1

relative to paper space (1xp), the second is scaled 0.5=1 relative to paper space (0.5xp), and so on.

4. Trim: Trims objects at a cutting object defined by other objects. Objects that can be trimmed include arcs, circles, elliptical arcs, lines, open 2D and 3D polylines, rays and splines

At the command prompt, enter trim
Select cutting edges: Select objects: use object selection method
<Select object to trim>/project/edge/undo: select an object, enter an option, or press enter

5. Break: This erases an object or splits the object in to two parts
From the modify toolbar select break flyout
At the command prompt, enter **break**
Select objects: use an object selection method
First point of the mirror line: specify a point (1) on an object
Enter second point: specify the second break point (2) or enter F

6. Area: This allows calculation of the area and perimeter of objects or of defined areas From the object properties toolbar, choose the inquiry flyout, then At the command prompt, enter **area**

<First point>/object/add/subtract: specify a point or enter option

7. Fillet: Rounds and fillets the edges of the object At the command prompt enter **fillet**

Polyline / Radius / Trim / <Select first object>: use an object selection method or enter an option Select first object

Select second object: use an object selection method Enter radius <current>: specify a distance or press Chain / Radius <Select edge>: Select edges or enter **c** or **r** their intersection

8. Explode: This breaks a compound object into its component objects

At the command prompt enter **explode**

Select objects: use an object selection method.

9. Union: This measures the distance and angle between two points.

At the command prompt, enter **union**

Select object: Use an object selection method

10. Dist: This measures the distance and the angle between two points.

At the command prompt area enter **dist**

First point: Specify a point (1)

Second point: Specify a point (2)

Distance = calculated distance

Angle in XY plane = angle from XY plane = angle

Delta X = change in X

Delta Y = change in Y

Delta Z = change in Z.

11. Regeneration of Drawing: ACAD provides a facility of regenerating a drawing to clear the cross points or marks on the screen.

- REDRAW
- REGEN
- REGENALL
- REGENAUTO

12. Tolerance: This creates geometric tolerances. Geometric tolerances define the maximum allowable variations of form or profile, orientation, location and run out from the exact geometry in a drawing. They specify the required accuracy for proper function and fit the objects drawn in AutoCAD

13. Sketch: This creates a series of free hand line segments.

From the miscellaneous toolbar, choose

At the command prompt enter **sketch**

Follow the prompting

14. TEXT: The text in software is indicated by font's .the fonts define the shapes of the text characters that make up each character set. In AUTOCAD, you can use true type fonts in addition to AUTOCAD's own compiled shape (SHX) fonts.

A font is indicated by various parameters like

i. Style :these are four types: normal, bold, italic, underline

ii. Size: this is the size of characters

iii. Color: there are facilities to color the characters selecting layer.

iv. Type: different types of fonts may be used:

Mono text: COMPUTER AIDED DESIGN

Romans: COMPUTER AIDED DESIGN

Romand: **COMPUTER AIDED DESIGN**

Dtext: This displays text on the screen as it is entered .AutoCAD can create text with a variety of character patterns, or fonts. These fonts can be stretched, compressed, oblique, mirrored, or aligned in a vertical column by applying a style to the font .text can be rotated, justified, and made any size.

At the command prompt, enter text

Justify/style/<start point>: specify a point or enter an option

Text: This creates a single line of text. AutoCAD can create text with a variety of character patterns, or fonts. These fonts can be stretched, compressed, oblique, mirrored, or aligned in a vertical column by applying a style to the font.

At the command prompt, enter **text**

Justify/style/<start point>: specify a point or enter an option

QTEXT: This controls the display and plotting of text and attribute of objects.

At the command prompt, enter **text**

ON/OFF <current>: enter on or off, or press enter

4.3.3 DISPLAY CONTROL COMMANDS

PAN: This moves the drawing display in the current viewport. At the command prompt, enter **Pan**

Displacement: Specify a point (1)

The point which specify indicates the amount to move the drawing or the location of the drawing to be moved.

Second point: Press or specify a point (2)

If pressed, ACAD moves the drawing by the amount which is specified in the Displacement

Prompt. If we specify a point, ACAD moves the location of the drawing to that point.

ZOOM: This increases or decreases the apparent size of objects in the current view port

At the command prompt, enter zoom

All/center/dynamic/extents/left/previous/vmax/window/<scale(x/xp)>: enter an option or value, specify a point, or press enter

2.3.4 TRANSFORMATIONS

These are the modifications in the drawn objects.

There are different types of transformations used

1. **Move:** This allows to move or displace objects a specified distance in a specified direction

At the command prompt, enter **move**

Select objects: use an object selection method

Base point or displacement: specify a base point (1)

Second point of displacement: specify a point (2) or press enter

2. **Copy:** This is used for producing a duplicate copy of the drawing.

At the command prompt, enter **copy**

Select objects: use an object selection method

<Base point or displacement >/multiple: specify a base point (1)

For a single copy or enter m for multiple copies

3. **Rotate:** It moves objects about a base point

   At the command prompt, enter **rotate**

   Select objects: use an object selection method

<Rotate angle >/reference: specify an angle or enter r

4. **Stretch:** This moves or stretches objects .AutoCAD stretches lines, arcs, elliptical arcs, splines, rays and polyline segments that cross the selection window.

   At the command prompt, enter **stretch**
   Select objects: use the CPOLYGON or cross object selection method (1,2)
   Base point or displacement: specify a point (3) or press
   Second point of displacement: specify a point ($) or press

5. **EXTEND:** This extends an object to meet another object. Objects that can be extended include arcs, elliptical arcs, lines, open 2D, and 3Dpolylines and rays.

   At command prompt, enter **extend**
   Select boundary edges
   (projmode=UCS, edge mode=no extend)
   Select objects: use an object selection method

6. **SCALE:** This enlarges or reduces selected objects equally in X and Y directions

   At the command prompt, enter **scale**
   Select objects: use an object selection method
   Base point: specify a point (1)
   <Scale factor>/reference: specify a scale or enter r

7. **TRACE:** This creates solid lines.

   From the miscellaneous tool bar choose
   At the command prompt, enter **trace**
   Trace width<current>: specify a distance, enter a value ,or press enter
   From point: specify point (1)
   To point: specify a point (2)
   To point: specify a point (3) or press to end the command

8. **EXTRUDE:** This creates unique solid primitives by extruding existing two-dimensional objects extrudes also creates solids by extruding two-dimensional objects along a specified path .we can extrude multiple objects with extrude

   At the command prompt enter, **extrude**
   Select objects: use an object selection method
   Path/<height of extrusion>: specify a distance or enter p

9. **MIRROR:** This is used to producing mirror image of the object

   At the command prompt enter, **mirror**
   Select objects: use an object selection method
   First point of the mirror line: specify a point (1)
   Second point: specify a point (2)

10. **OFFSET**: This creates concentric circles, parallel lines and parallel curves, offset creates a creates a new object at a specified distance from an existing object or through a specified point

   At the command prompt enter, **offset**
   Offset distance: specify a distance, enter t or press enter

2.5 3D FUNCTIONS

1. **Box:** This creates a three dimensional solid box.

At the command prompt enter **box**

Center/<corner of the box><0,0,0> :

Specify a point (1), enter c, or press enter

Corner of a box

Specifying a point or pressing defines the first corner of the box.

Cube/length /<other corner>: specify a point (2) or enter an option **center**

Creates the box by a specified center point

2. **Cone:** This creates a 3D solid cone. A cone is solid primitive with a circular or elliptical based tapering symmetrically to a point perpendicular to its base.

   At the command prompt enter **cone**

   Elliptical /<center point> <0,0,0> : specify a point , enter e or press enter

3. **Cylinder:** This creates a 3D solid cylinder. A cylinder is solid primitive with a circular or elliptical based to a point perpendicular to its base without a taper.

   At the command prompt enter **cylinder**

   Elliptical /<center point> <0,0,0> : specify a point , enter e or press enter

4. **Sphere:** This creates a 3D solid sphere. A sphere is positioned so that its central axis is parallel to the Z-axis of the current UCS. Latitudinal lines are parallel to the XY plane.

   At the command prompt enter **sphere**

   Center of the sphere <0,0,0> : specify a point , enter e or press enter

5. **Wedge:** This creates a three dimensional solid with a sloped face tapering along X axis.

   At the command prompt enter **wedge**

   Center <corner of the wedge> <0,0,0> : specify a point , enter e or press enter

   Follow the prompting

6. **Elev:** This sets an elevation and extrusion thickness of new objects. The current elevation is the Z value that is used whenever a 3D point is expected but only X and y values are supplied.

   At the command prompt enter **elev** Follow the prompting

7. **Shade:** This displays a flat shaded image of the drawing in the current view port. SHADE removes hidden lines and displays a shaded picture of the drawing.

   From the render toolbar, choose

   At the command prompt, enter **shade**

8. **Region:** This creates a region object from a selection set of existing objects. Regions are 2Dimensional areas you create from closed shapes.

9. **Reinit:** This reinitializes the input/output ports, digitizer, display and program parameters file.

10. **Replay:** This displays a GIF, TGA or TIFF image.

    From the tools menu, choose image, then view.

11. **Revolve :**This creates a solid by revolving a two – dimensional object about an axis. From the solids toolbar, choose

    At the command prompt, enter **revolve**

12. **Shape :** This inserts a shape. Before inserting a shape, you must load the file containing the desired shape.
13. **Rotate 3d:** This moves objects about a three dimensional axis
    From the modify toolbar, choose the rotate flyout then
    Follow the prompting

14. **Section:** This uses the intersection of a plane and solids to create a region.

    AutoCAD creates regions on the current layer and inserts them at the location of the cross – section. Selecting several solids creates separate regions for each solid.

15. **Slice:** This slices a set of solids with a plane.

16. **Shell :** This accesses operating system commands.

17. **Revolve:** This creates a solid by revolving a two dimensional object about an axis.

18. **Render:** This creates a realistically shaded image of a three dimensional wireframe or solid model. RENDER produces an image using information from a scene, the current selection set, or the current view.

3. 2D DRAWINGS

FIGURE 1

Aim: - Draw the plan and elevation of the given drawing using autocad-2019.

Commands used: - Construction line, circle, fillet, Tangent trim offset.

Procedure:-

1) Invoke auto cad -2019 from the menu or from shortcut icon on the desktop.
2) Set limits for the working window
   limits <enter> (0,0)<enter>(150, 150) <enter>
   xl <enter> select horizontal line and again.

xl <enter> select vertical line. The origin is considered as O1.

3) Draw a circle of diameter 28 mm from the origin (O1)
4) Now take an offset command with a distance of 60mm and draw a vertical line on right side. The origin is considered as O2. From origin O2 draw a circle of diameter 28mm.
5) Draw the circle of radius 27mm from the originsO1 and O2.
6) With an offset command, draw a horizontal line with a distance of 40mm in the upward direction from the origins O1 and O2 lines. Draw a vertical line on the right side with an offset distance of 26mm from the origin O1 line. This gives us an origin O from the origin O3; draw a circle of dia 20mm.
7) With an offset command, draw a horizontal line with a distance of 82mm in the upward direction from the origin O1 andO2 line. We get origin O4 from origin O4draw a circle of 28mm dia and 27mm radius
8) Use the fillet command to get an arc on the large circle of origin O1 and O4 with a 30mm radius and on the large circle of origin O1 andO2 with a radius 10mm.
10) Using OSNAP function, →settings→ tangent draw a line on O4 and O2.

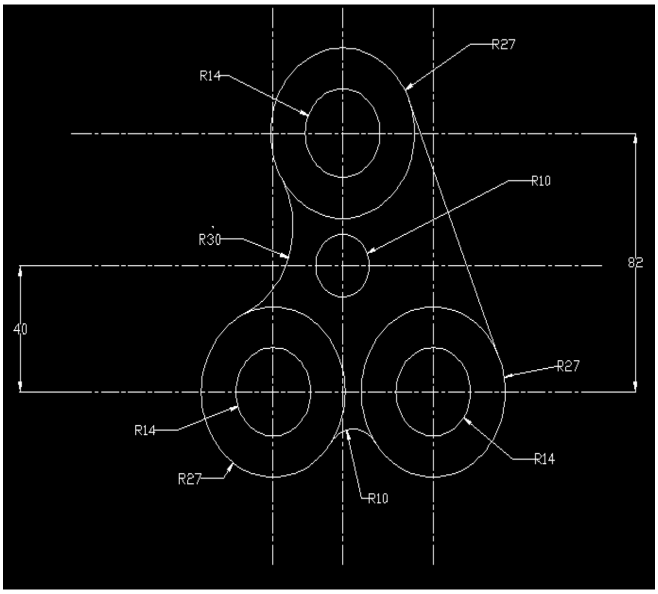


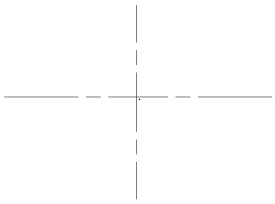
FIGURE 2

Aim: to draw the following figure using ACAD

PROCEDURE

STEP 1: Draw axis lines in the respective format with their intersection point at **(0,0)**

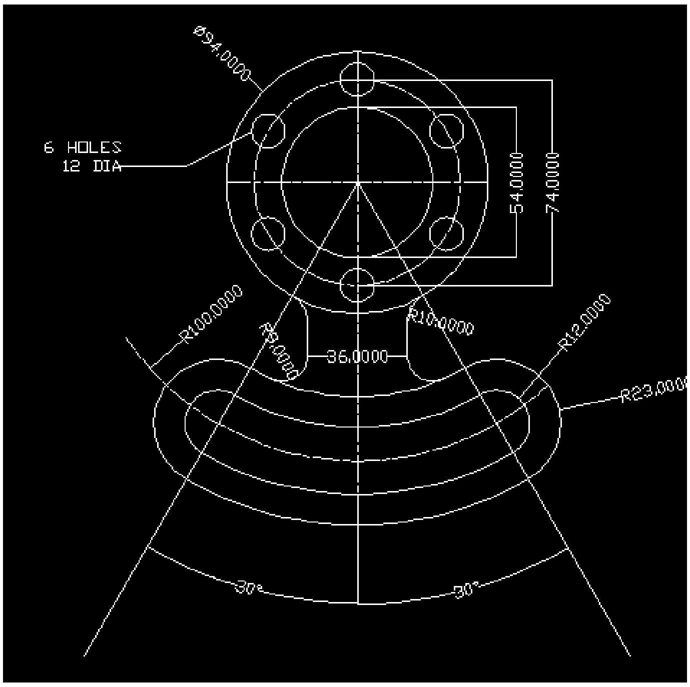
- Go to **LAYER PROPERTIES** tool bar and select LineType
- Load line type as **ISO LONG DASH SHORT DASH** in the line type area.
- Select line type ISO LONG DASH SHORT DASH in the line type area.

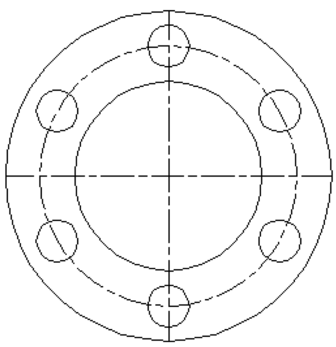


STEP 2 a: Draw circles of given dimensions using circle command with their centre as the intersection of the axis lines.

- 3 circles of diameters 94, 74 and 54 are to be drawn
- The circle with 74 diameter is of **ISO LONG DASH SHORT DASH** format

STEP 2 b: Using **POLAR ARRAY** draw the 6 holes on the circle of diameter 74 each of 12 dia as shown in the figure below





STEP 3: Draw two construction lines at an angle of 30° to the vertical axis line

STEP 4: With A as center an radius 100 draw an arc between the above lines

STEP 5: Offset the arc on the either side by the distances as mentioned in the figure.

STEP 6: Complete the figure by using fillet command.

STEP 7: Give dimensions to the completed figure.

Command: _qsave

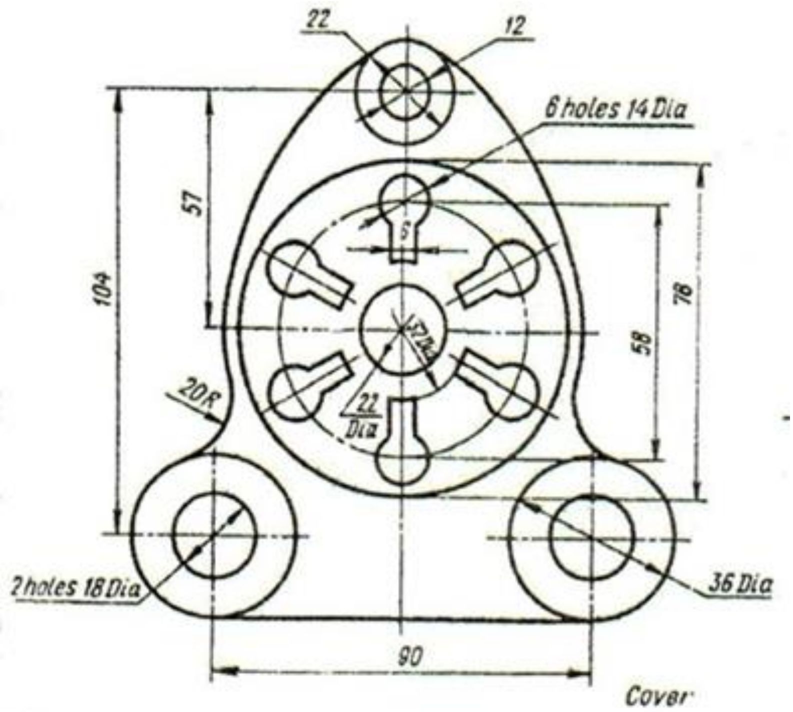
<u>PRECAUTIONS:</u>

Put **ORTHO ON** where ever necessary.

Use the required modify tool bar commands like **TRIM, ERASE, COPY, MIRROR** ETC.,

FIGURE 3

Aim: to draw the following figure using ACAD



PROCEDURE

Set the limits of the drawing screen

STEP 1: Draw axis lines in the respective format with their intersection point at **(0,0)**

STEP 2: Draw circles of given dimensions using circle command with their centre as the intersection of the axis lines.

STEP 3: Using **POLAR ARRAY** draw the 6 key holes on the circle of diameter 58 of given dimensions

STEP 4: For the outer cover use **CIRCLE** command and the in command prompt area type **TAN TAN RADIUS**. This gives the idea of drawing the outer cover

STEP 7: Give dimensions to the completed figure.

Command: _qsave

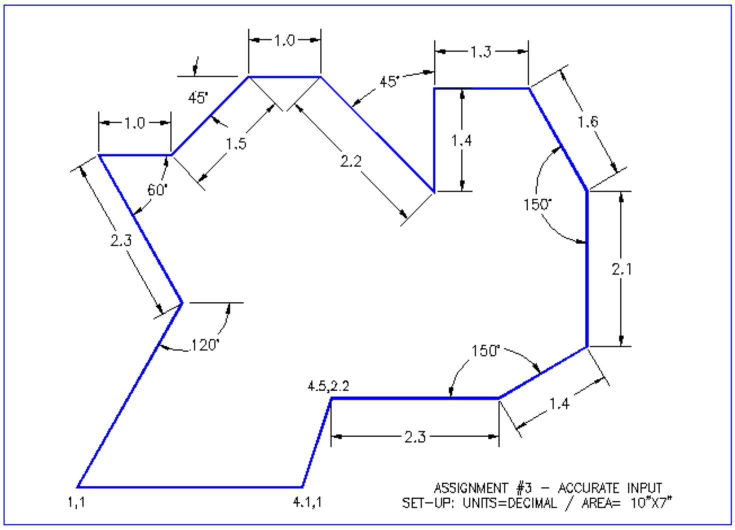
PRECAUTIONS:

Put **ORTHO ON** where ever necessary.

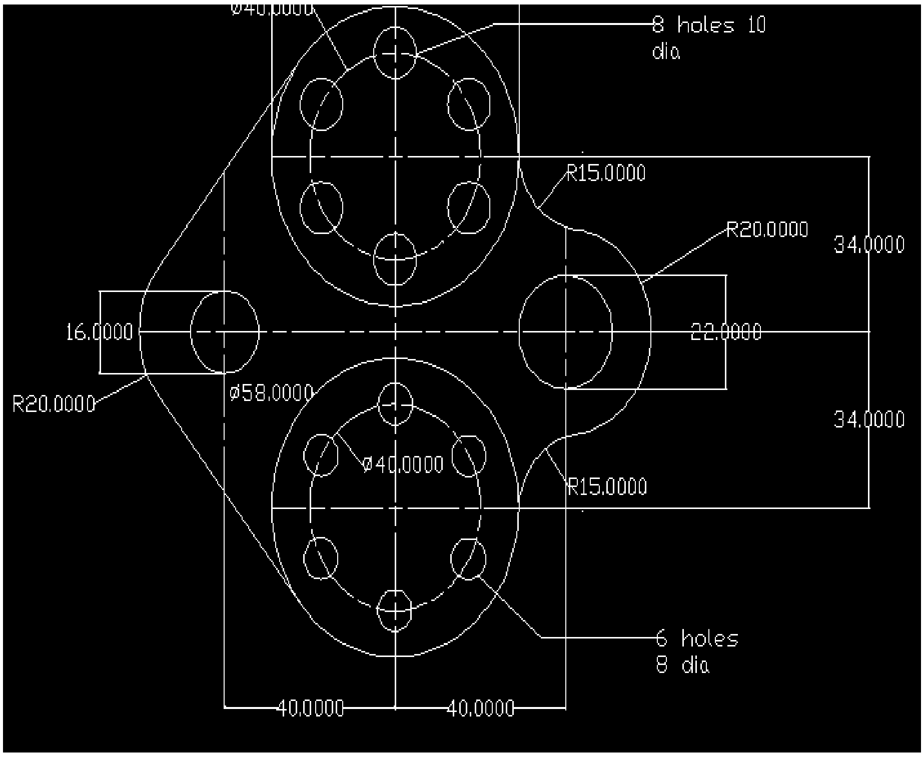
Use the required modify tool bar commands like **TRIM, ERASE, COPY, MIRROR**

ETC.,

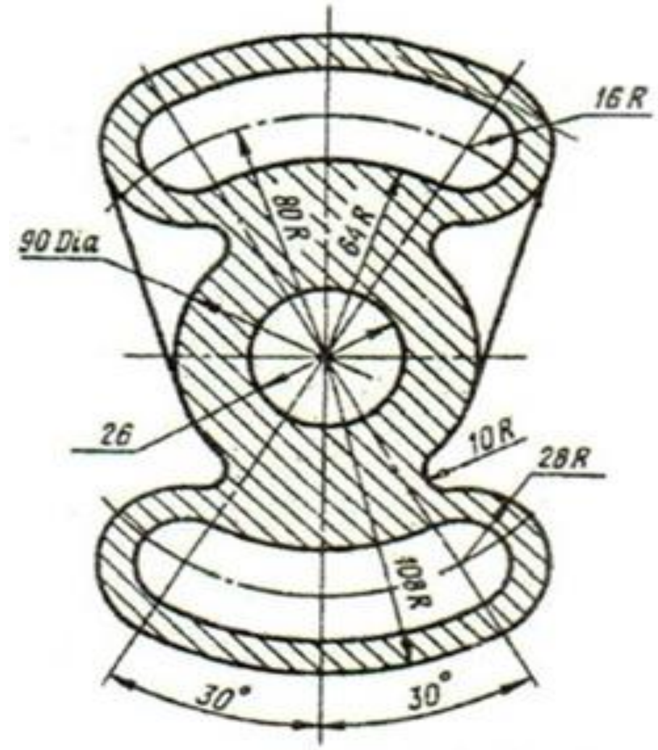
PRACTICE FIGURE 1



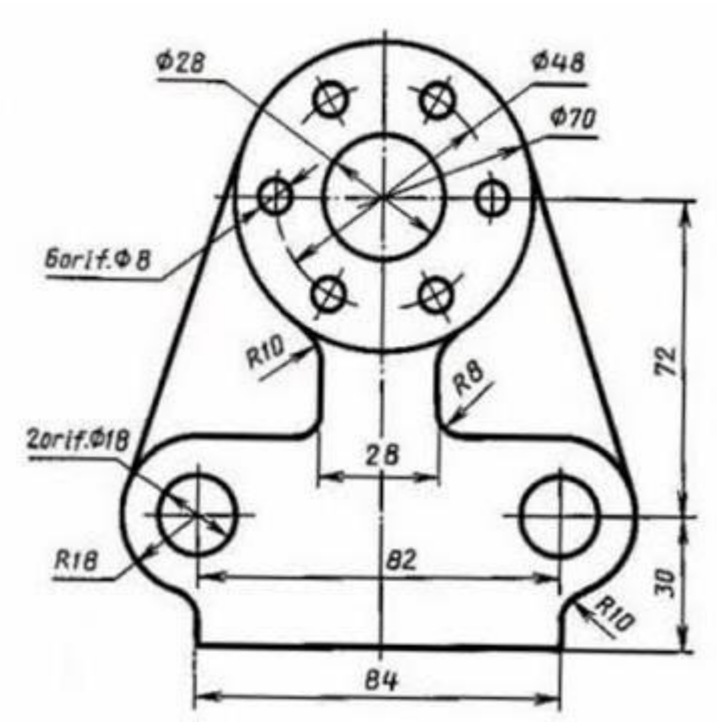
PRACTICE FIGURE 2



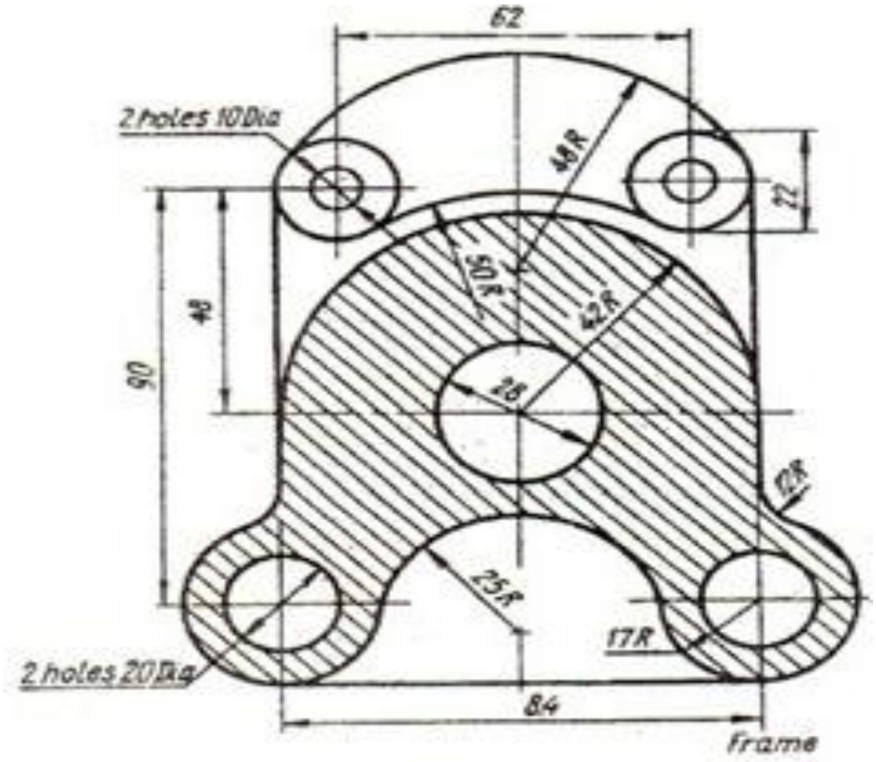
PRACTICE FIGURE 3



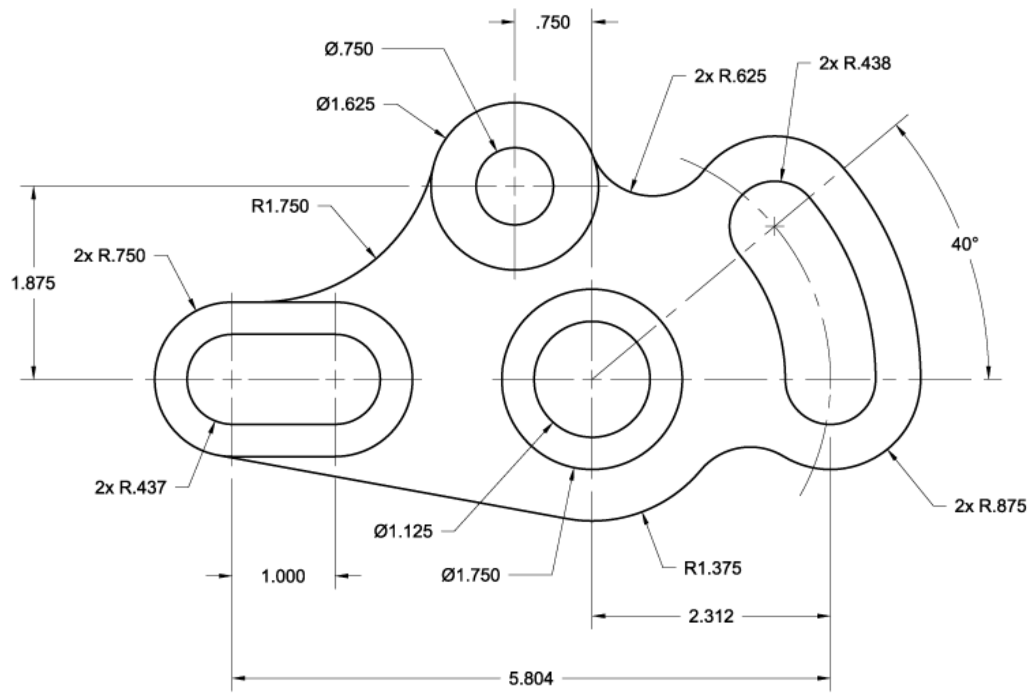
PRACTICE FIGURE 4



PRACTICE FIGURE 5



PRACTICE FIGURE 6



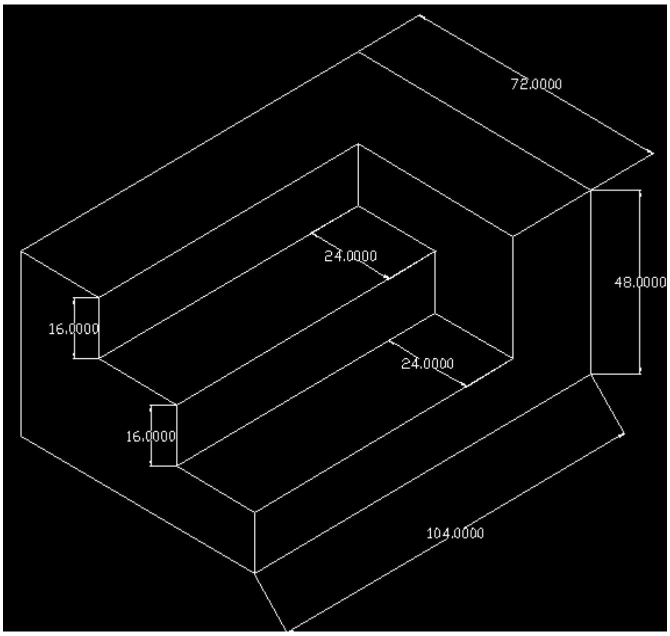
4. ISOMETRIC DRAWINGS

For all isometric figures right click **GRID** in drafting tool bar <setting> change grid snap to **ISOMETRIC SNAP**. And check **ORTHO ON**

F5 – TOGGLE KEY BETWEEN ISOPLANE TOP, ISOPLANE LEFT AND ISOPLANE RIGHT

FIGURE 1

Aim: to draw the following figure using ACAD



COMMANDS USED

Line, Dimensions, Drafting commands

PROCEDURE

<Ortho on> <Isoplane Top> <Osnap on>

Command: _line Specify first point:

Specify next point or [Undo]: **104**

Specify next point or [Undo]:

Command: _qsave

Command: _dimaligned

Specify first extension line origin or <select object>:

Specify second extension line origin:

Command: _dimlinear

Specify first extension line origin or <select object>:

Specify second extension line origin:

Specify dimension line location or [Mtext/Text/Angle/Horizontal/Vertical/Rotated]:

Dimension text = 48.0000

Command: _dimedit

Enter type of dimension editing [Home/New/Rotate/Oblique] <Home>: **_o**

Select objects: 1 found

Enter obliquing angle (press ENTER for none): **30** **Command: _qsave**

FIGURE 2

Aim: to draw the following figure using ACAD

COMMANDS USED

Line, Drafting commands, Dimension aligned, Dimension linear, Dimension oblique, Layers

Command: _line

Specify first point: <Isoplane Left>

Specify next point or [Undo]: 12

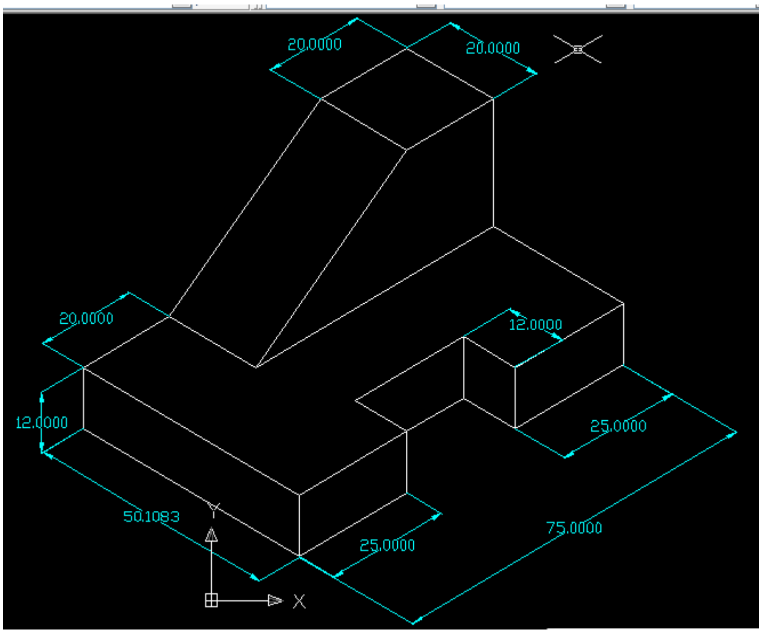
Specify next point or [Undo]: <Isoplane Top> 25

Command: _qsave

Command: _dimlinear

Specify first extension line origin or <select object>:

Specify second extension line origin:



Specify dimension line location or [Mtext/Text/Angle/Horizontal/Vertical/Rotated]:

Dimension text = 12.0000

Command: _dimaligned

Specify first extension line origin or <select object>:

Specify second extension line origin:

Specify dimension line location or [Mtext/Text/Angle]:

Dimension text = 25.0000

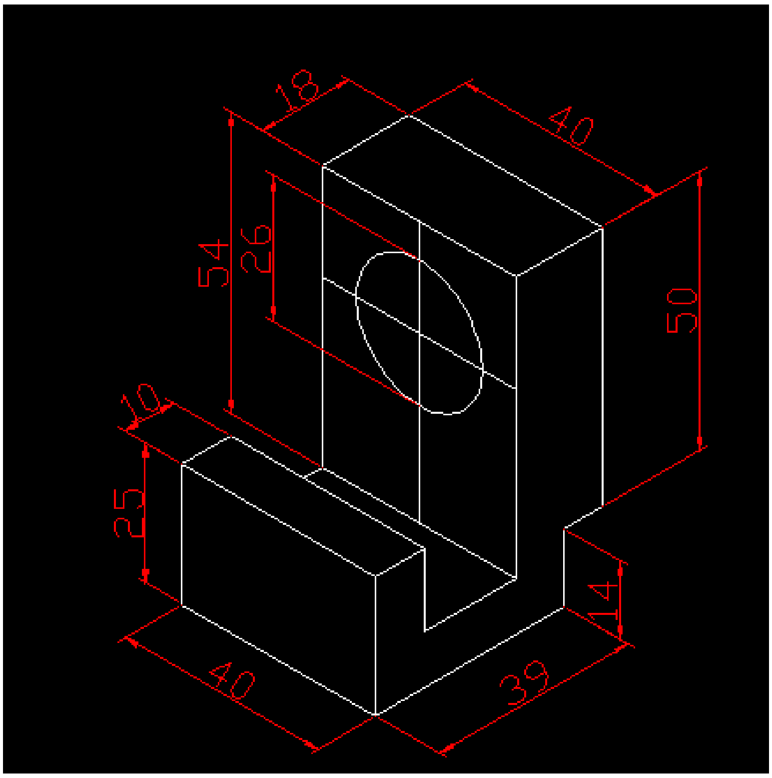
Command: _dimedit

Enter type of dimension editing [Home/New/Rotate/Oblique] <Home>: **_o**

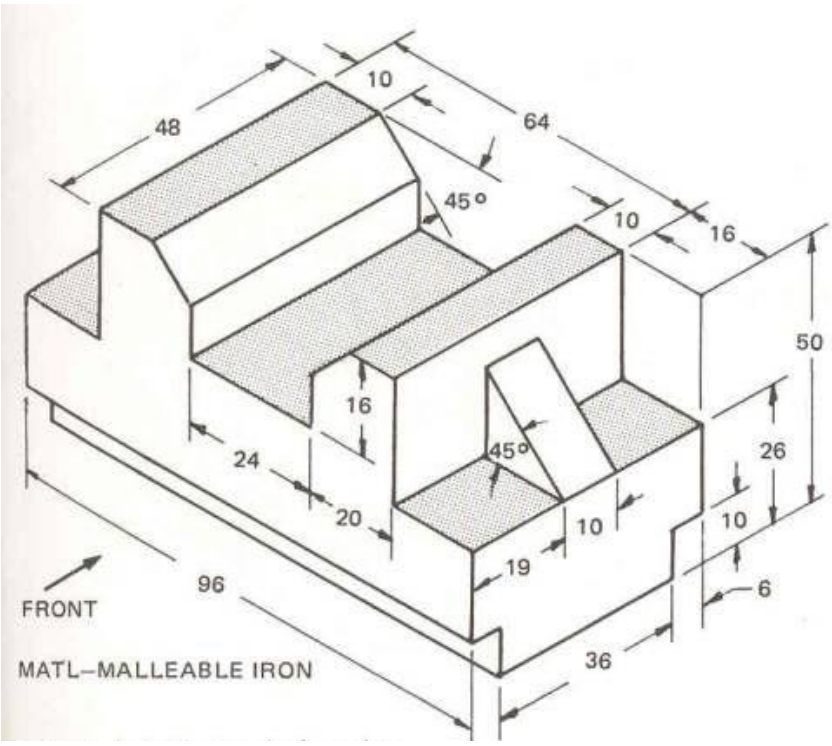
Select objects: 1 found

Enter obliquing angle (press ENTER for none): **30 or -30** **Command: _qsave**

PRACTICE FIGURE 1

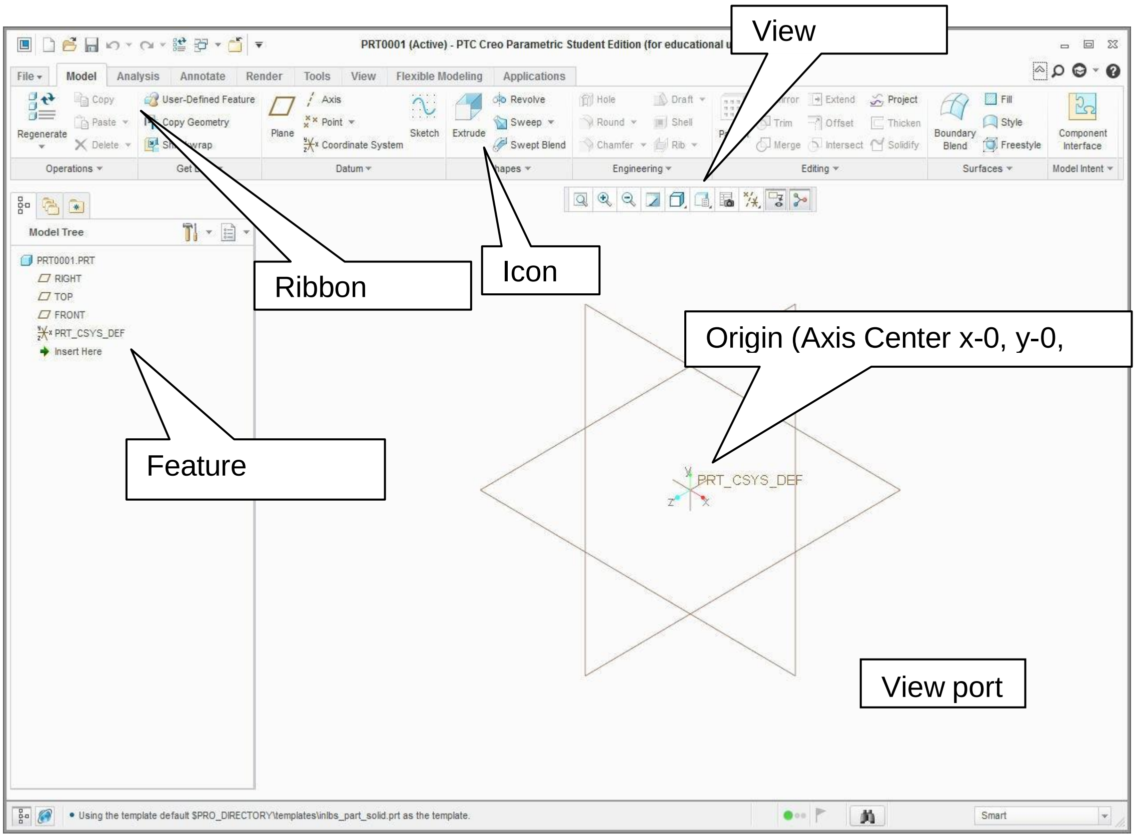


PRACTICE FIGURE 2



INTRODUCTION TO CREO 3.0

CREOParametric 3.0 Interface



Mouse Buttons

Left Button - Most commonly used for **selecting** objects on the screen or sketching.

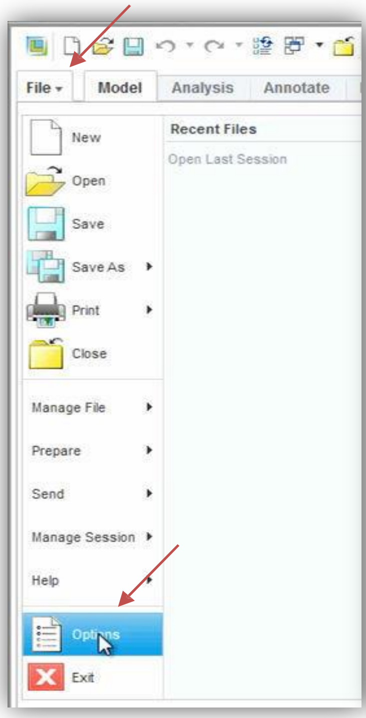
Right Button – Used for activating pop-up **menu** items, typically used when editing. (*Note: you must hold the down button for 2 seconds*)

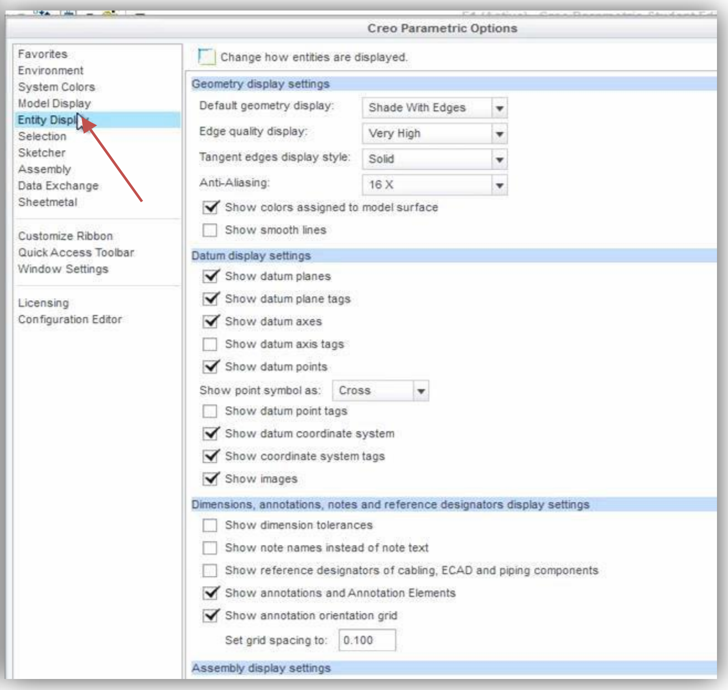
Center Button – (option) Used for model **rotation**, **dimensioning**, **zoom** when holding Ctrl key, and **pan** when holding Shift key. It also **cancels** commands and line chains.

Center Scroll Wheel – (option) same as Center Button when depressed, only it activates **Zoom** feature when scrolling wheel

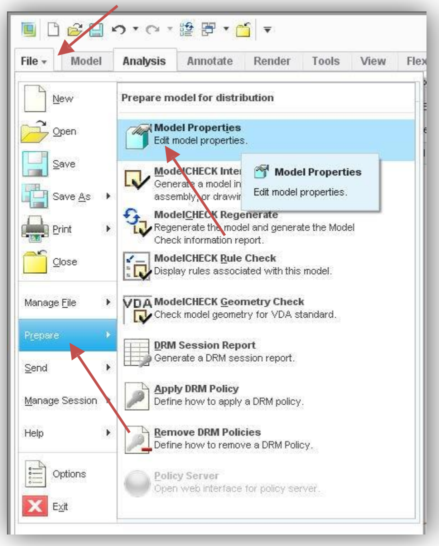
Options & Properties, menus, the heart of Creo

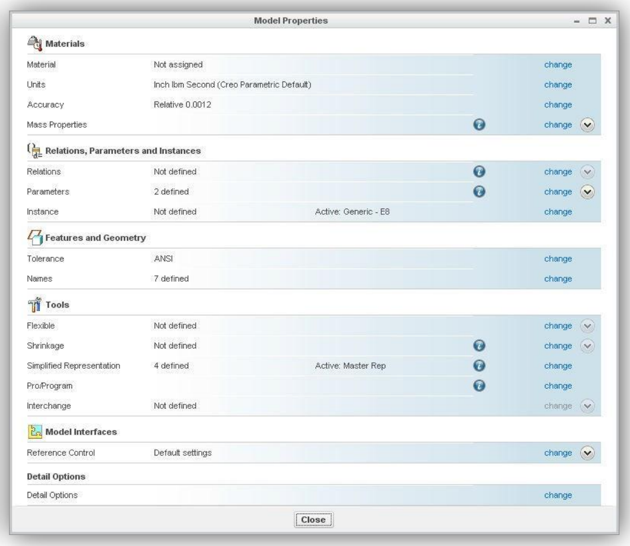
Selecting the File—Options --pull down- *(located at the top left side of the screen)* **opens the active documents Options.**



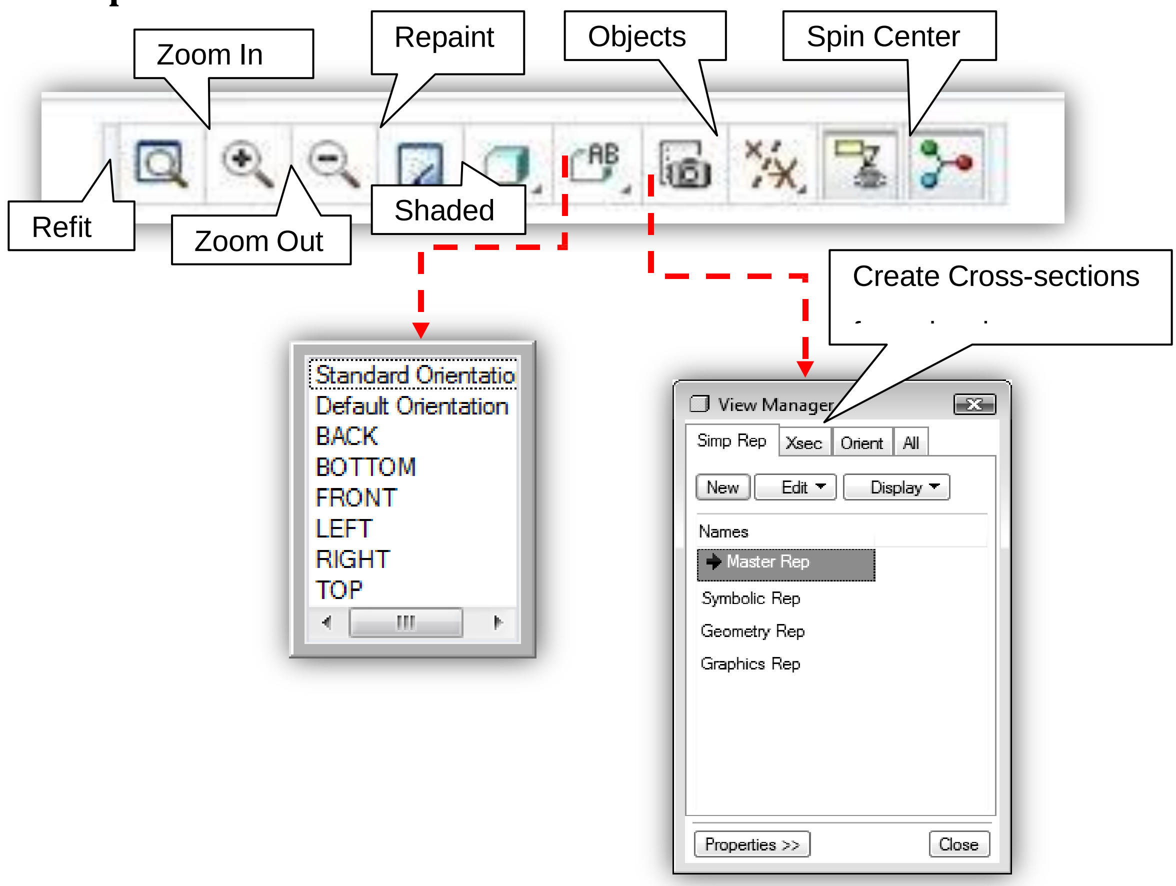


Model Properties

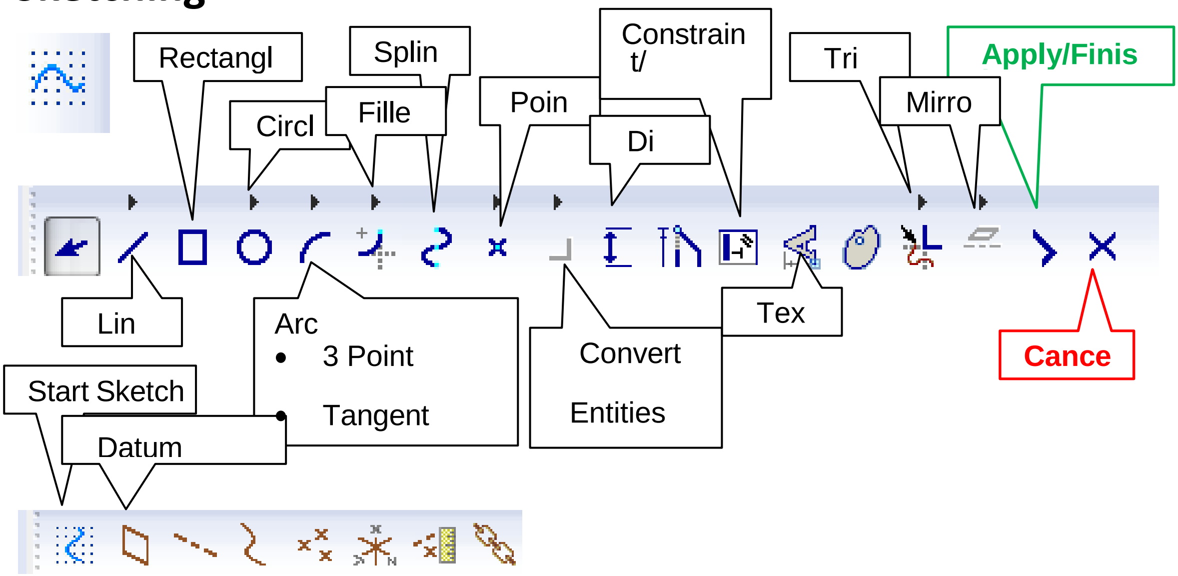




View options



Sketching

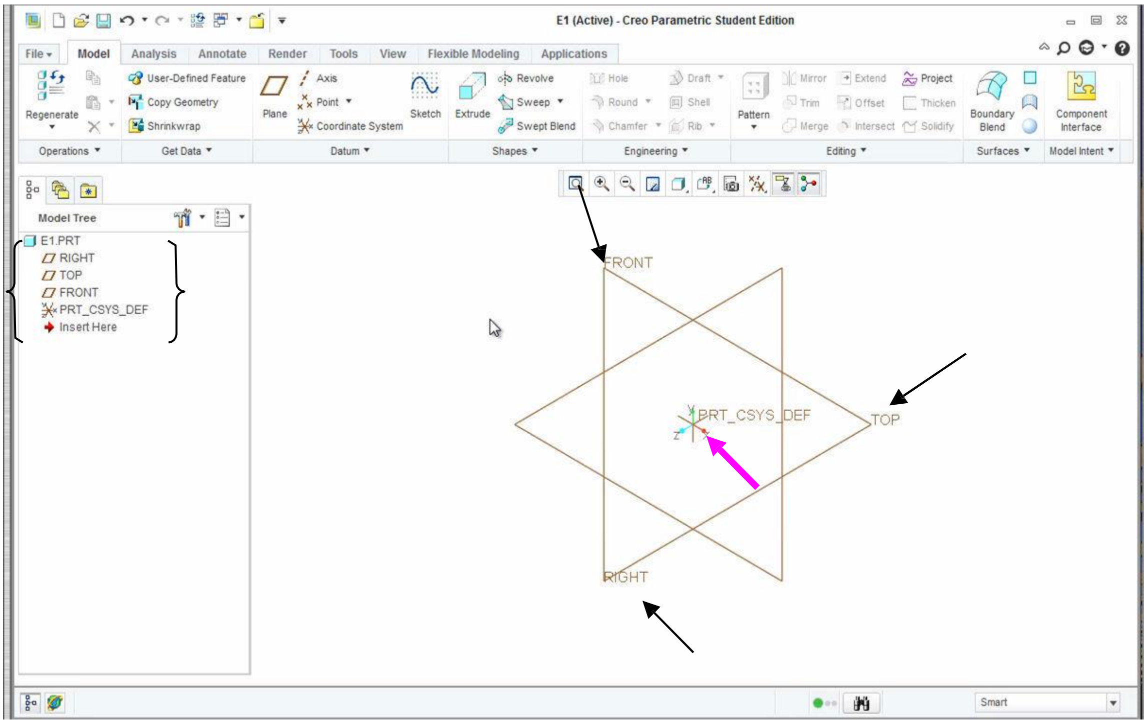


NOTE: If you do not see all of these icons on your interface you can customize the toolbars to bring them up.

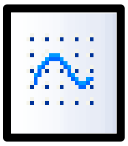
Where do you start a sketch?

Sketches can be created on any Datum Plane or Planar Face or Surface. Pro/E provides you with three datum planes centralized at the Origin (your zero mark in space)

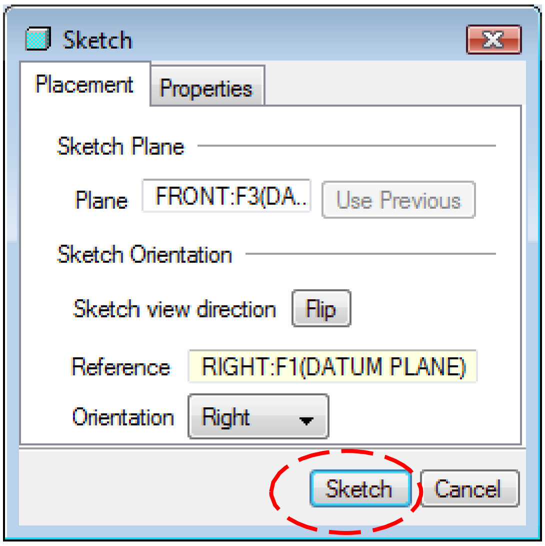
NOTE: Planes can also be created and will be discussed in more detail in the future. Also after completing a sketch always select the Apply/Finish check mark on the sketch toolbar, this will activate the extrude or revolve feature tools.



To start a sketch Pre-select the plane or face you desire to sketch on and then select the Sketch Icon.



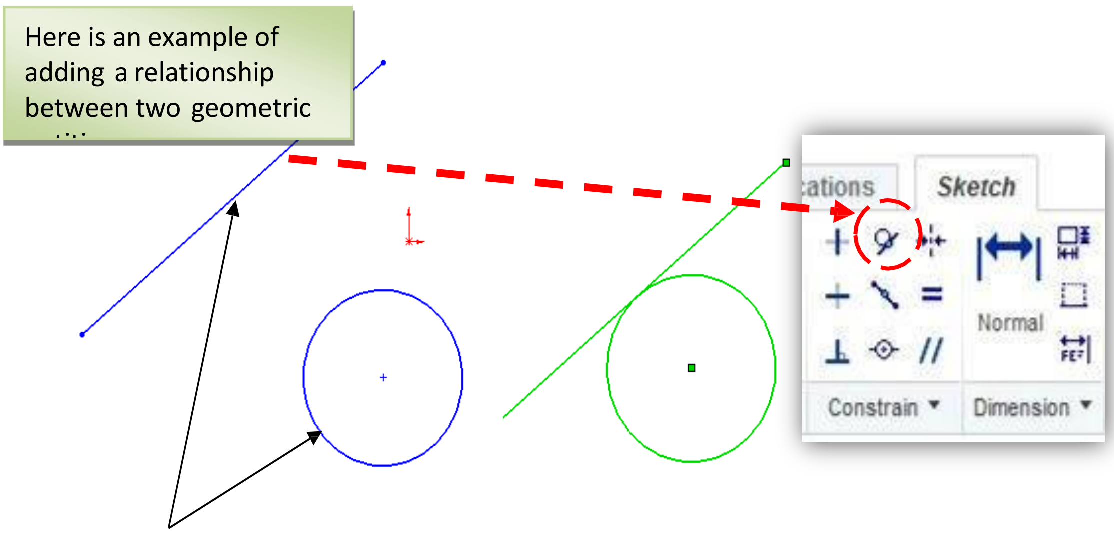
Sketch Options



Controlling your geometry

Pro/E uses two methods for constraining geometric entities.

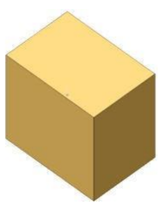
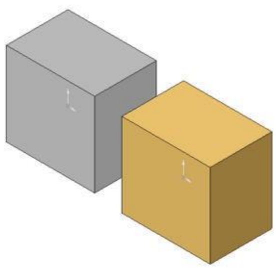
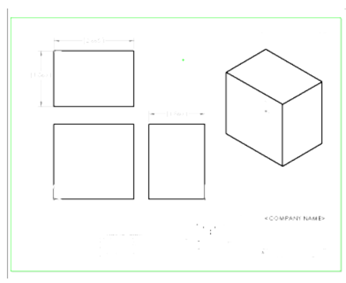
Constraints and **Dimensions:**
Constraints can be referred to as common elements of geometry such as Tangency, Parallelism, and Concentricity. These elements can be added to geometric entities automatically or manually during the design process.



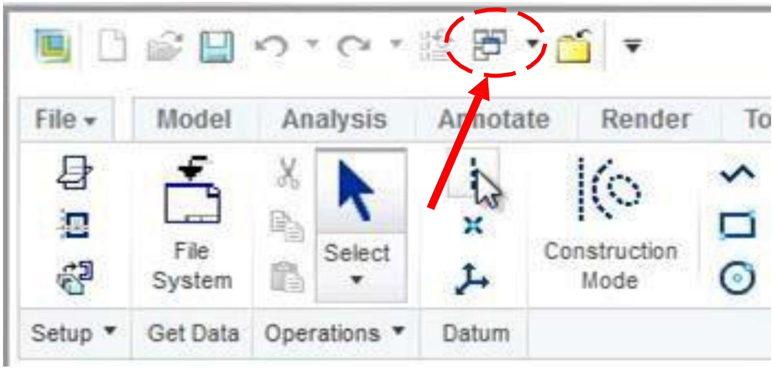
Cautious sketching can save time

There are 3 primary file types in Creo, which iŶclude…

1. ***Part*** (.prt) Single part or volume.
2. ***Assembly*** (.asm) Multiple parts in one file assembled.
3. ***Drawing*** (.drw) The 2D layout containing views, dimensions, and annotations.

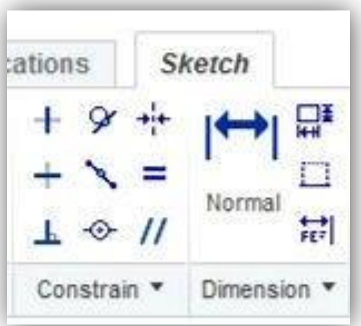




Switching between documents (Activating a document)



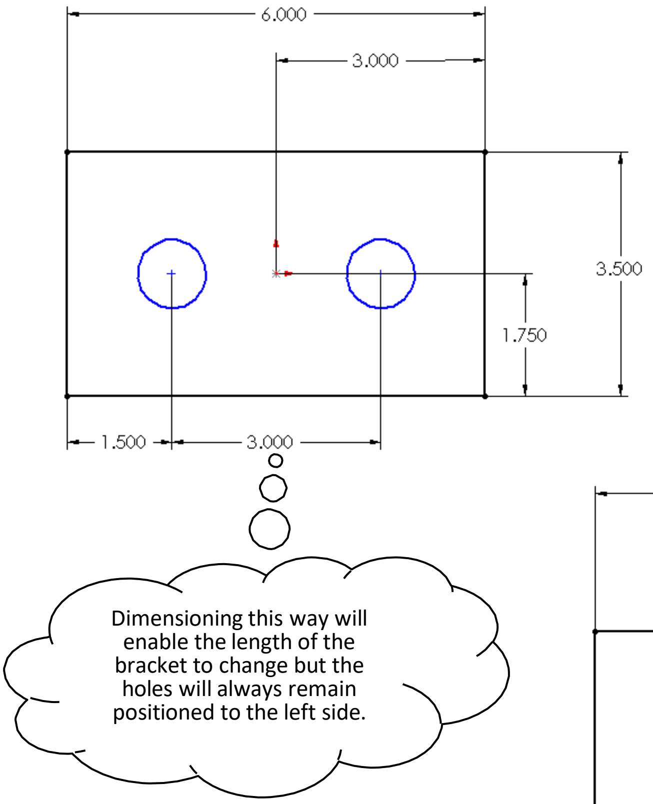
Select the Window pull-down menu and you will see the available documents. Click on the document you wish to work on from the list to update it.

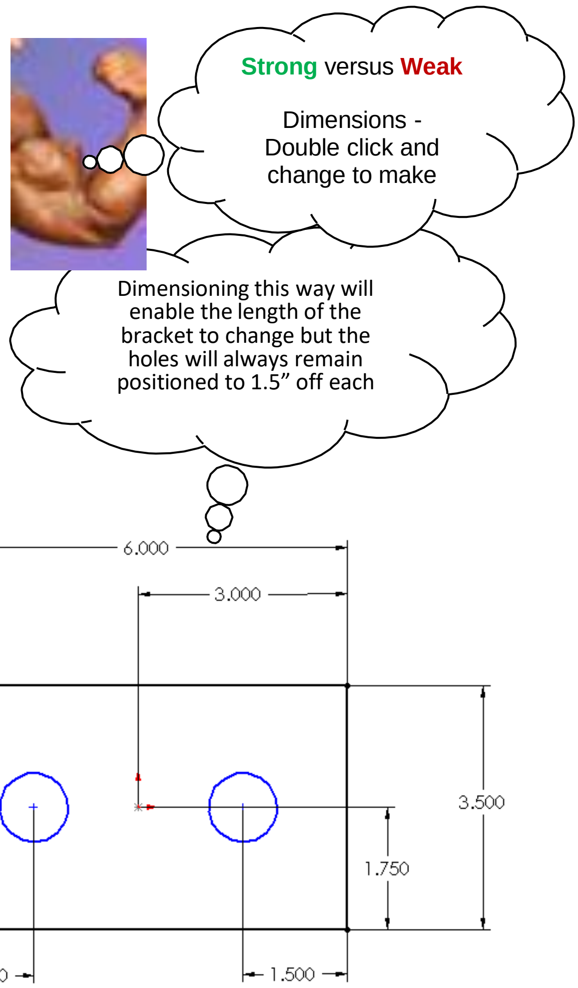
Sketch Constraints (Relations)

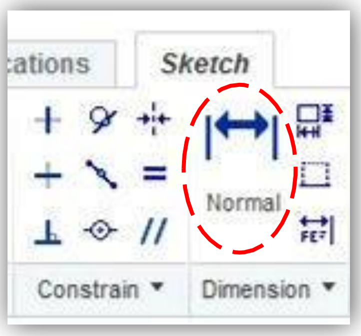


Constraint	Geometric entities to select	Resulting Constraint
Horizontal or Vertical	One or more lines or two or more points.	The lines become horizontal or vertical (as defined by the current sketch space). Points are aligned horizontally or vertically.
Collinear	Two or more lines.	The items lie on the same infinite line.
Perpendicular	Two lines.	The two items are perpendicular to each other.
Parallel	Two or more lines. A line and a plane (or a planar face) in a 3D sketch.	The items are parallel to each other. The line is parallel to the selected plane.
Tangent	An arc, ellipse, or spline, and a line or arc.	The two items remain tangent.
Concentric	Two or more arcs, or a point and an arc.	The arcs share the same centerpoint.
Midpoint	Two lines or a point and a line.	The point remains at the midpoint of the line.
Coincident	A point and a line, arc, or ellipse.	The point lies on the line, arc, or ellipse.
Equal	Two or more lines or two or more arcs.	The line lengths or radii remain equal.
Symmetric	A centerline and two points, lines, arcs, or ellipses.	The items remain equidistant from the centerline, on a line perpendicular to the centerline.

Controlling your geometry with dimensions…

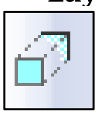




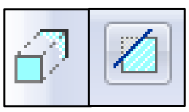


Solid Modeling Basics

1. Layer Cake method



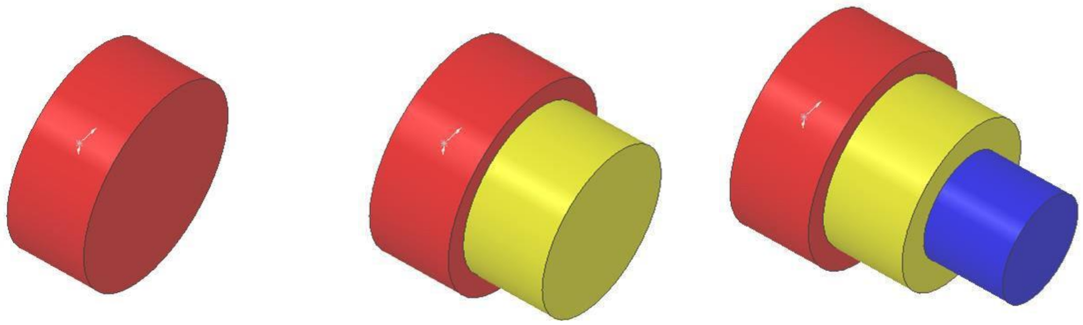
Extruded Boss/Base (Creates/Adds material)



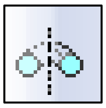
Extruded Cut (Removes material)

2. Ingredients:

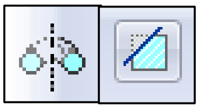
- Profile



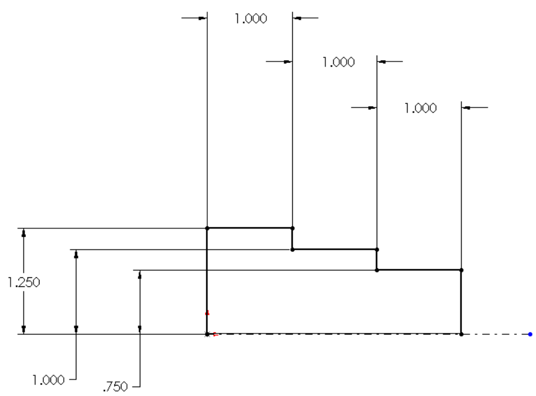
3. Revolve method

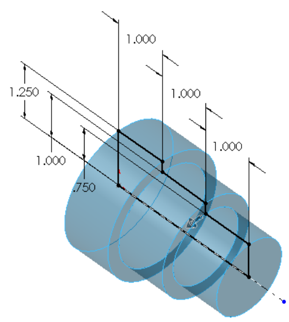


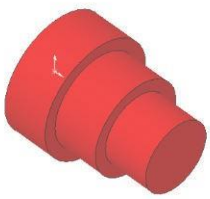
Revolve Boss/Base (Creates/Adds material)



Revolve Cut (Removes material)



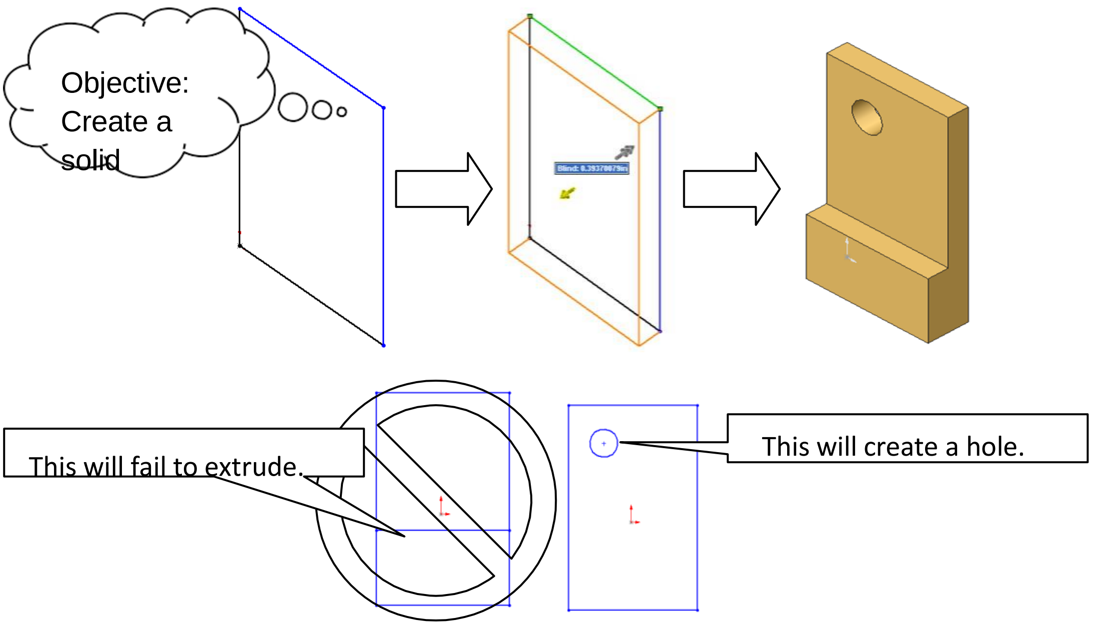


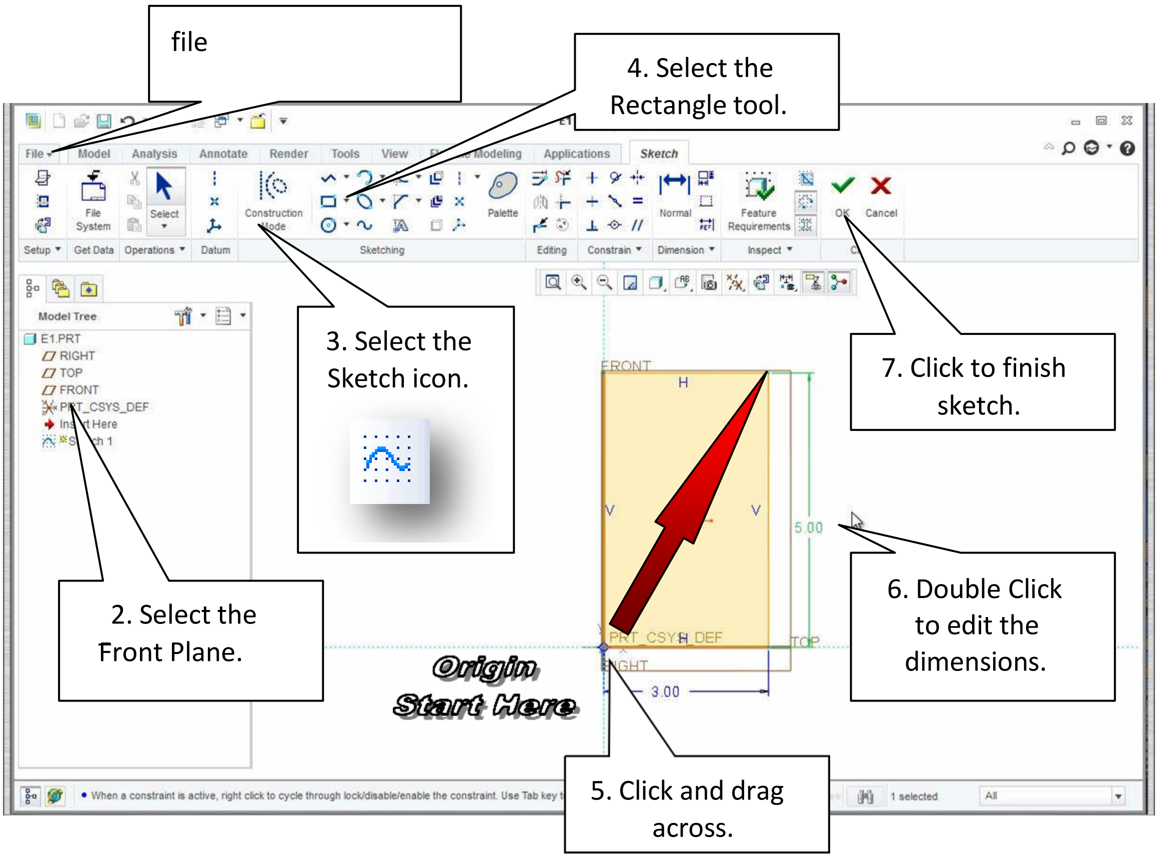


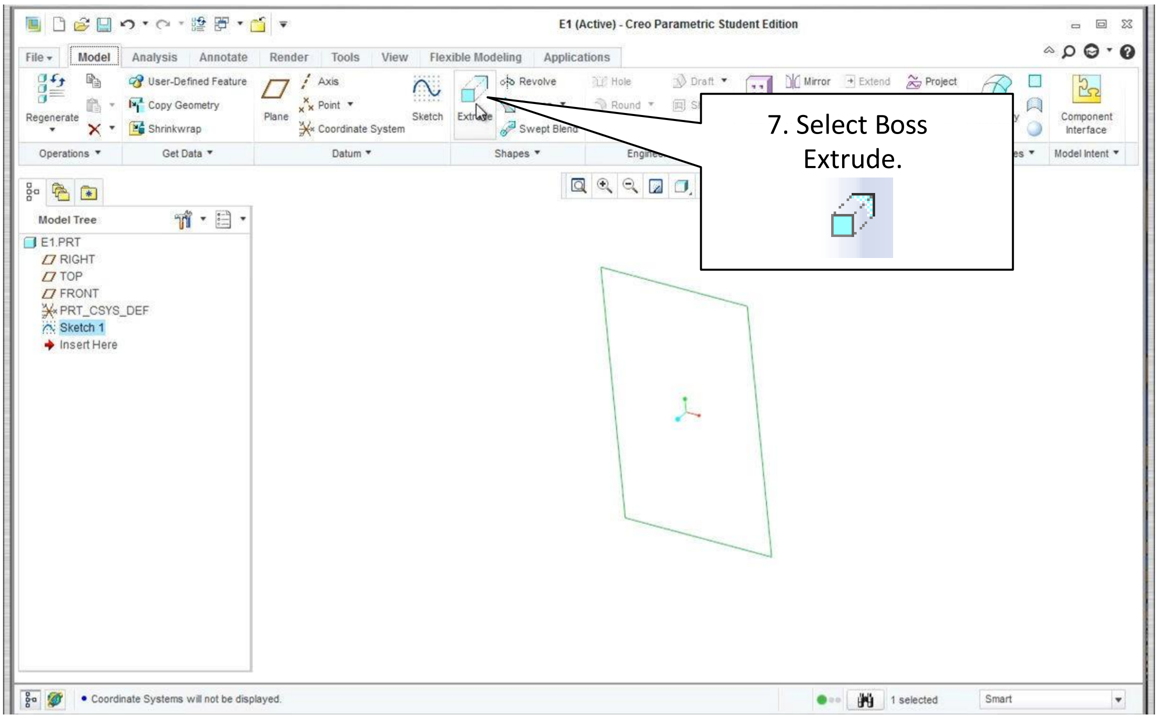
EXERCISE 1

Introduction to basic part modeling

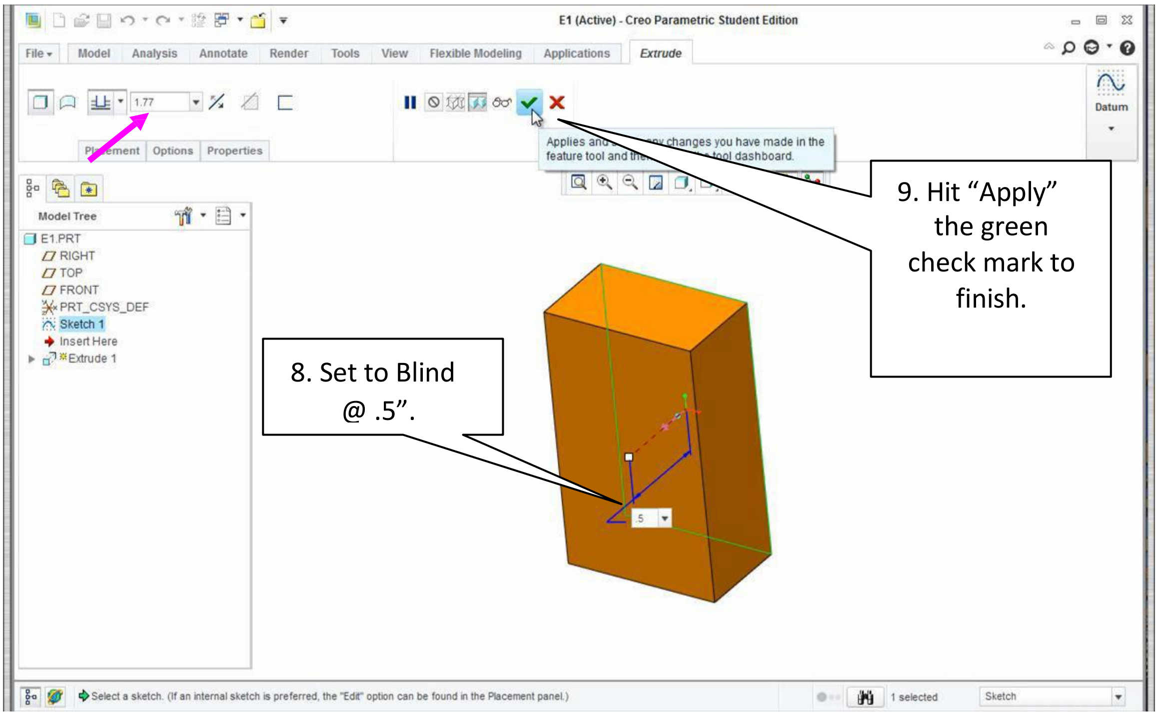
Base Extrude Features *create a 3D solid representation by extruding a 2 dimensional profile of the entity.*

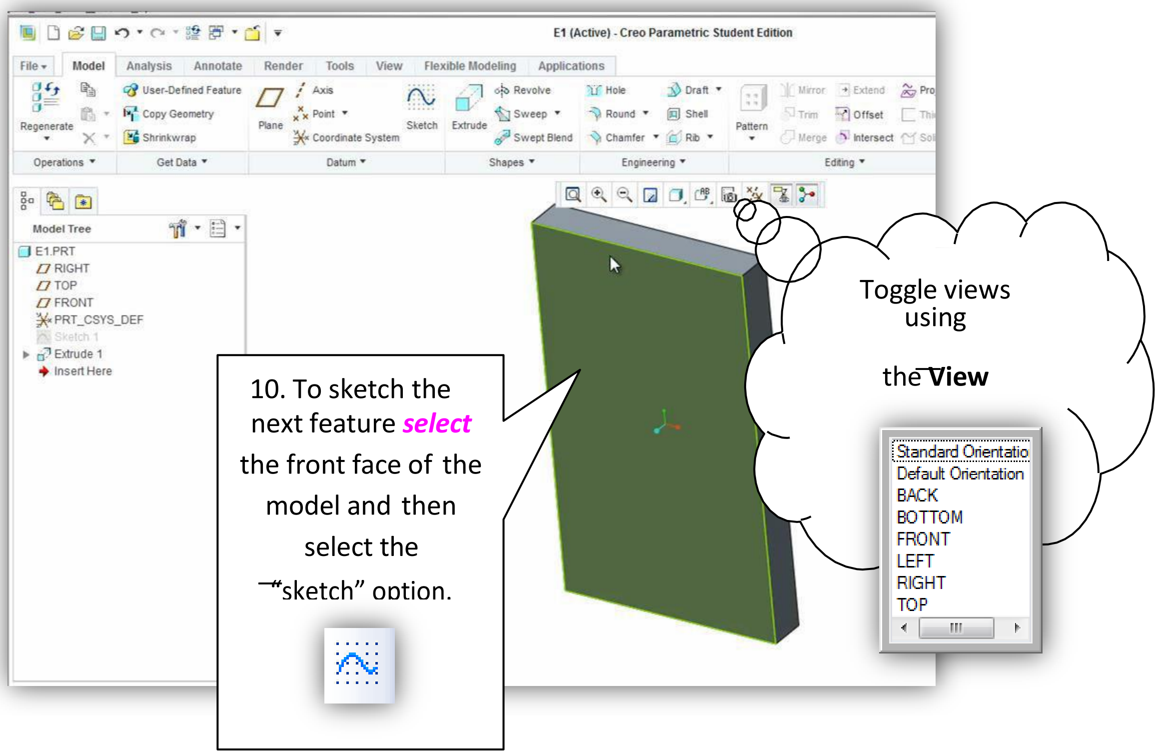




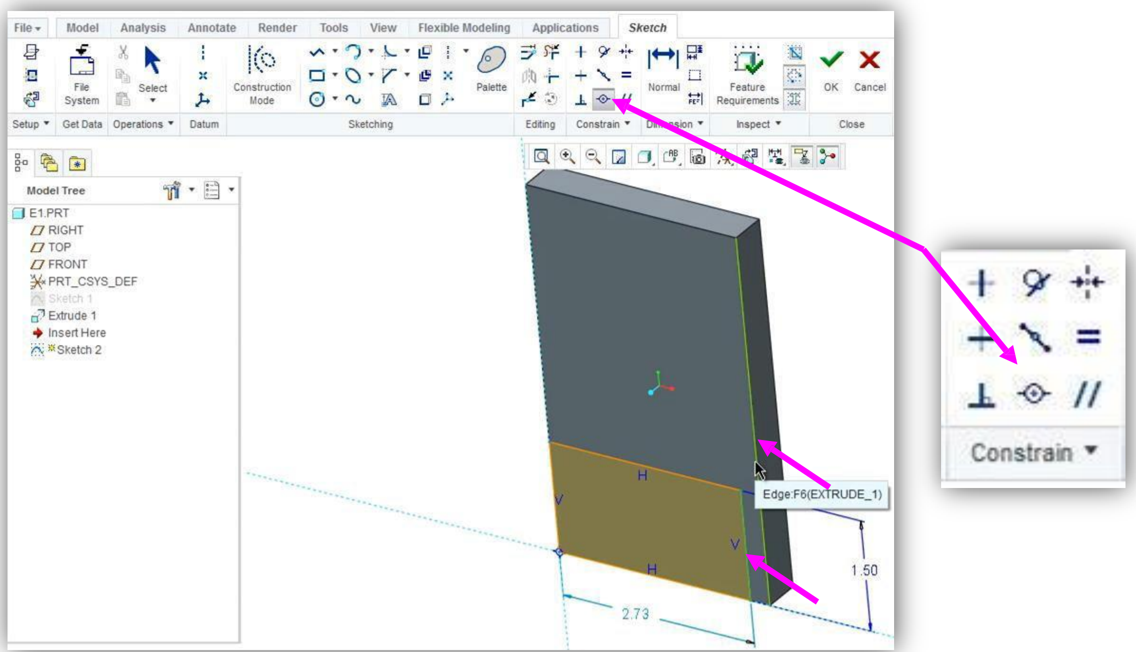


NOTE: When dimensioning use the dimension tool and make edge selections, mouse center button click to apply dimension.

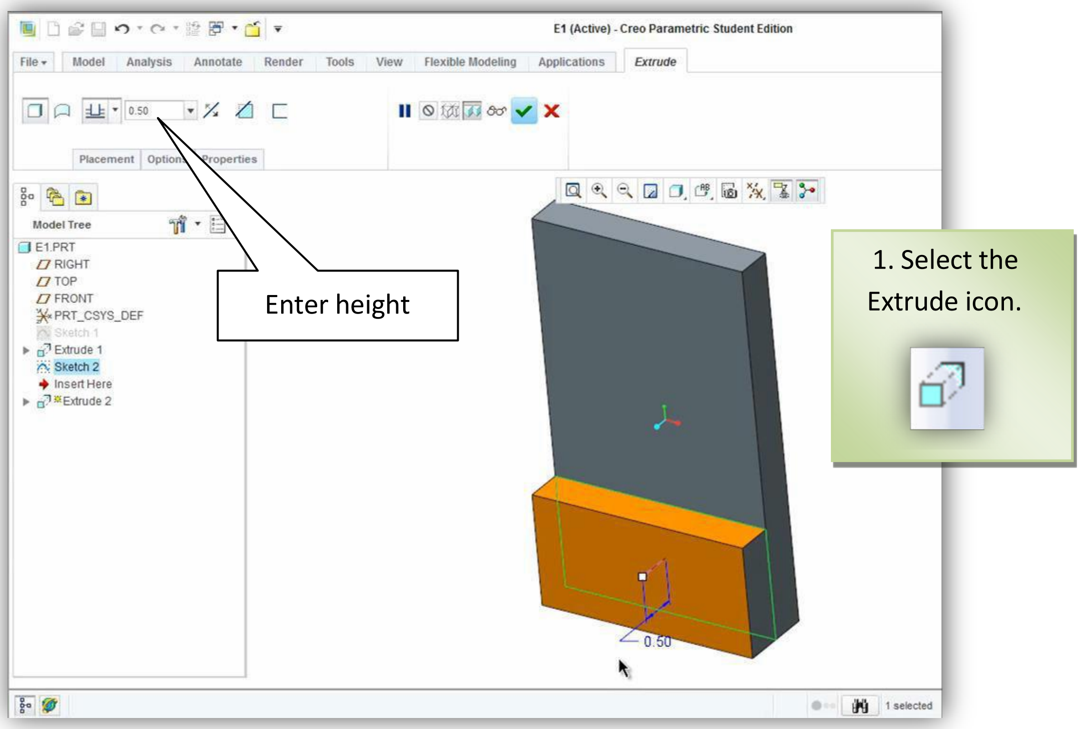




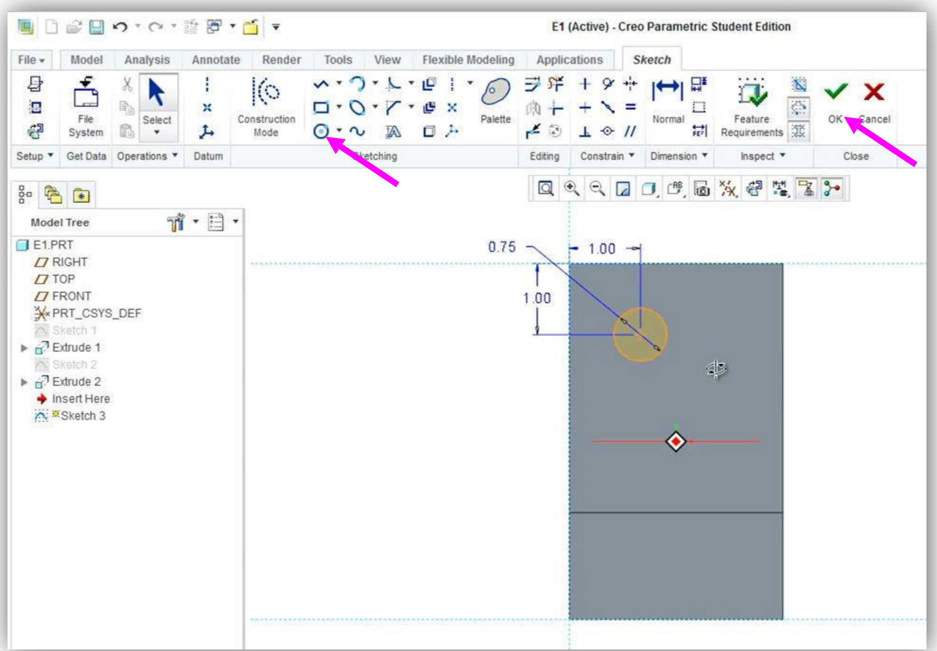
Adding a constraint – Ctrl Select both left edges of sketch and solid. Select Coincident



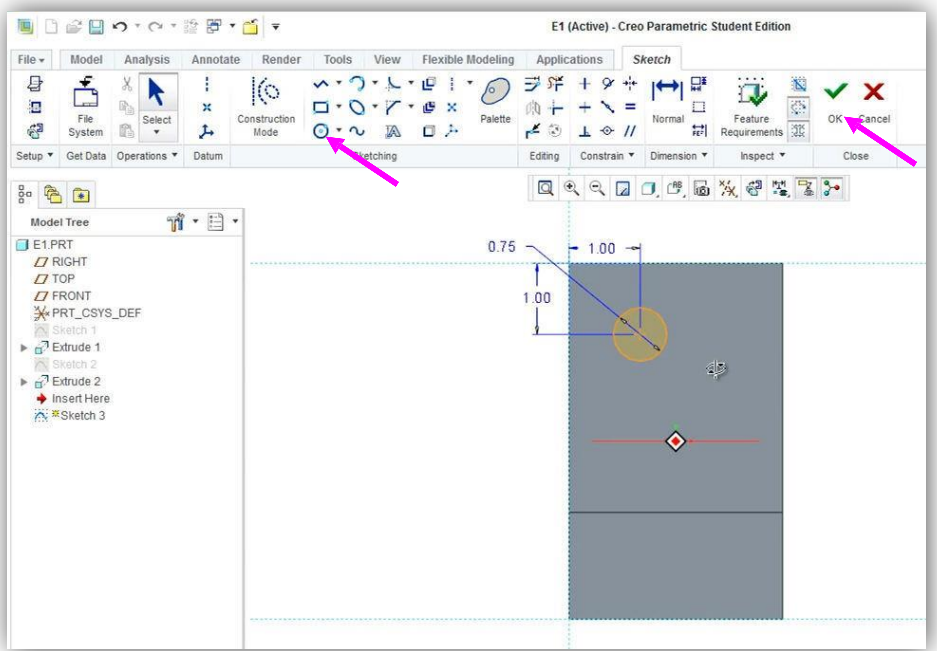
Extrude



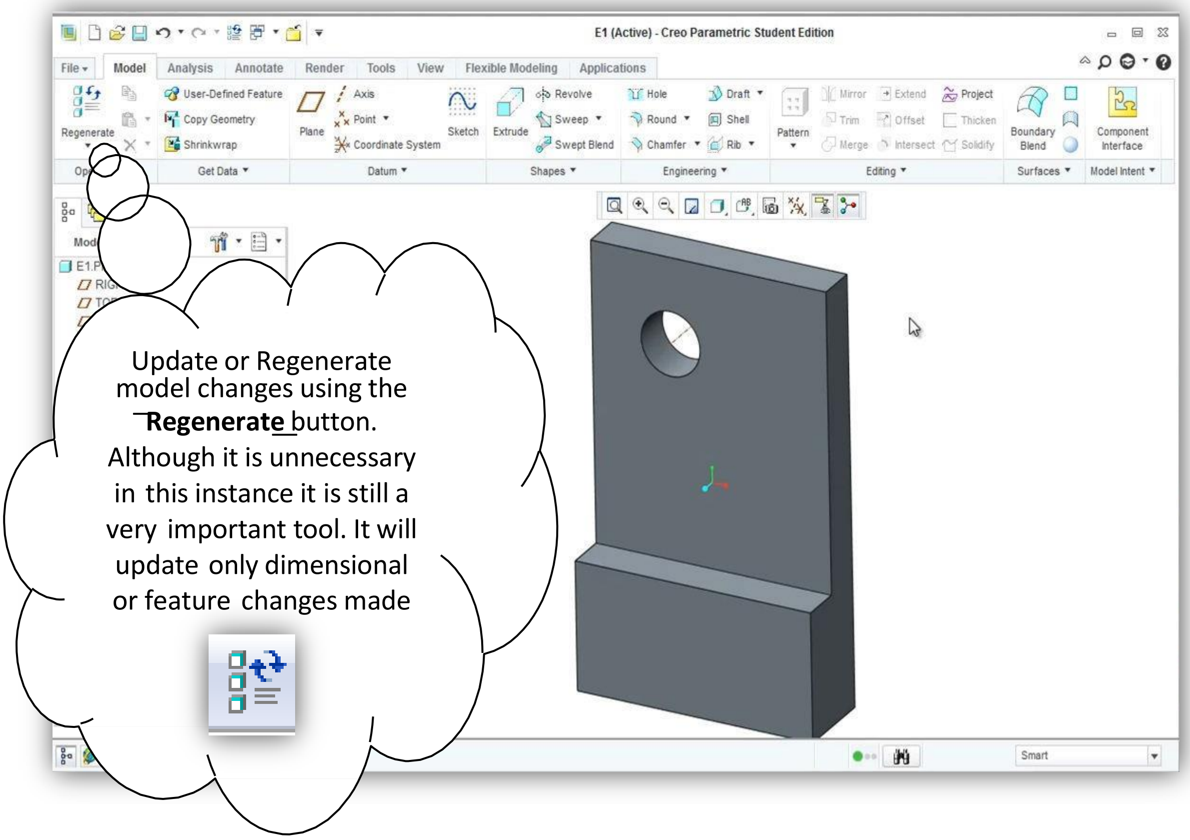
Select the face, select sketch icon and draw a circle on the face. Dimension, Hit OK



Select the face, select sketch icon and draw a circle on the face. Dimension, Hit Ōk_



Go to file save

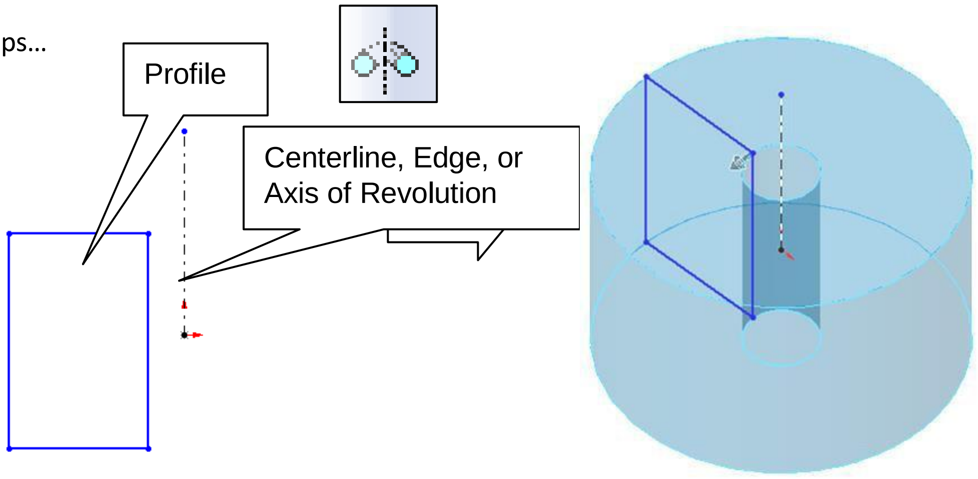

E1 (Active) - Creo Parametric Student Edition
Update or Regenerate model changes using the **Regenerate** button. Although it is unnecessary in this instance it is still a very important tool. It will update only dimensional or feature changes made

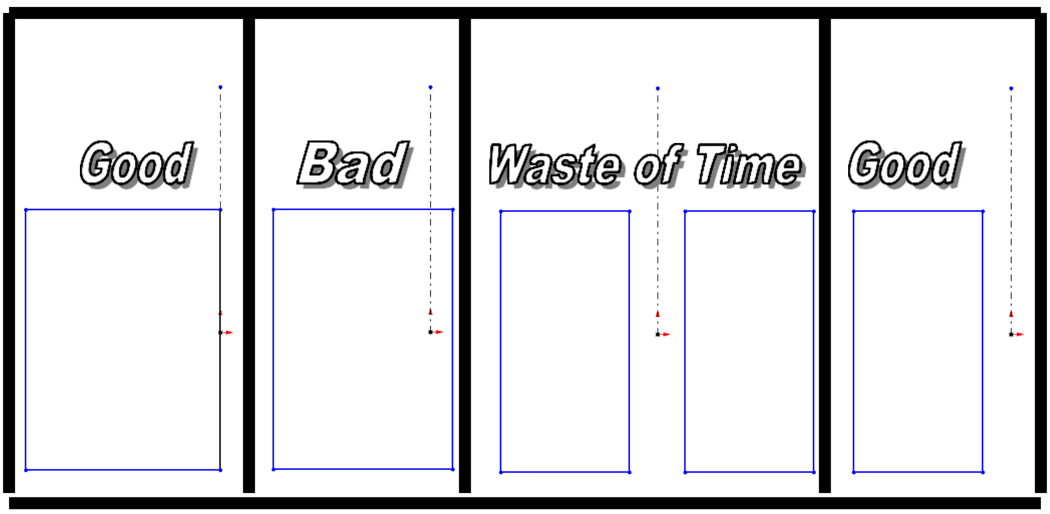
EXERCISE 2

Revolved Features

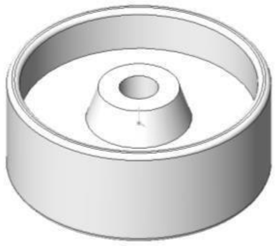
Revolved Feature - creates features that add or remove material by revolving one or more profiles around a centerline. The feature can be a solid, a thin feature, or a surface**.**

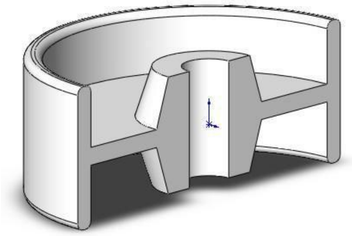
Tips…



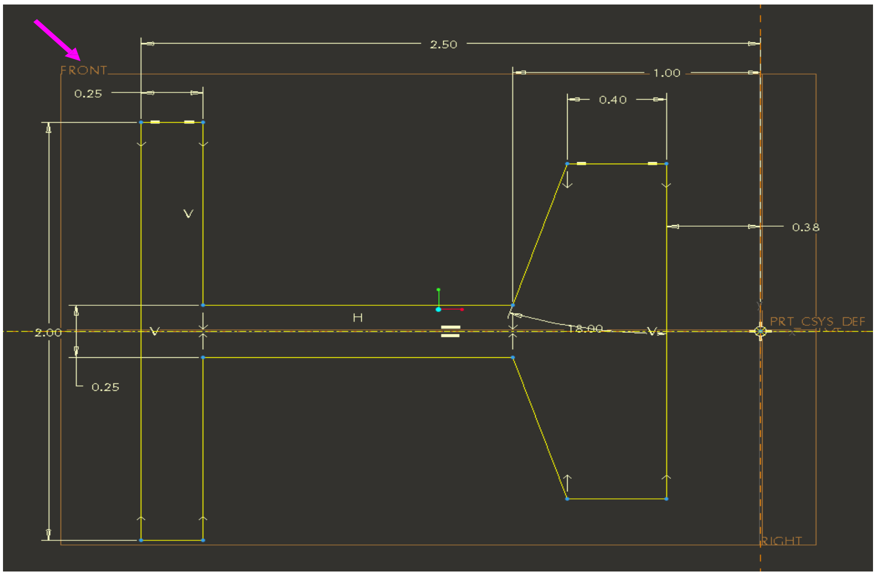


The profile should never cross over the centerline, nor should there be profiles on both sides of the centerline.

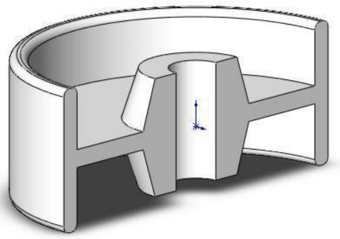


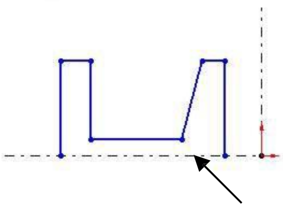


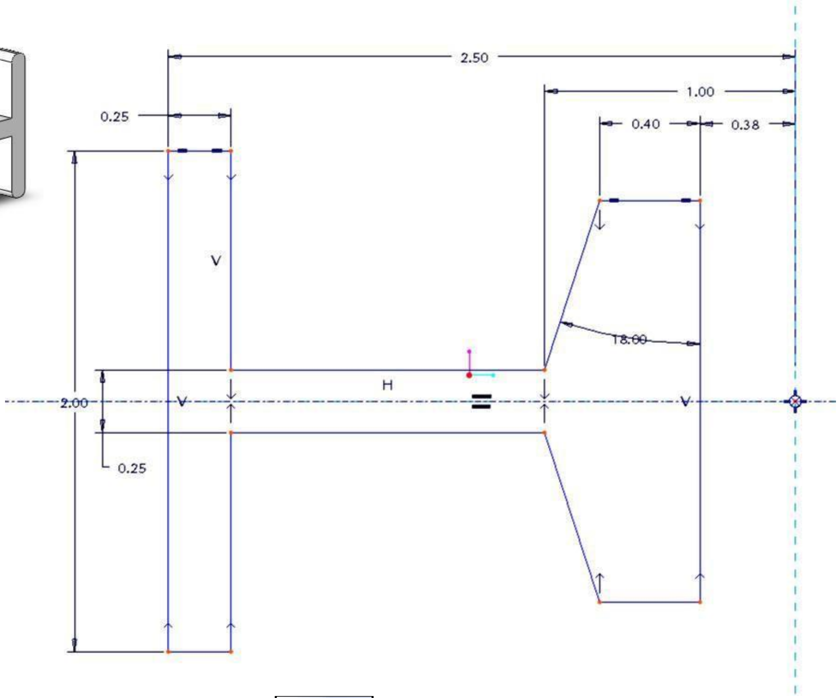
Create a new pat file .

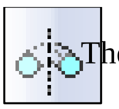


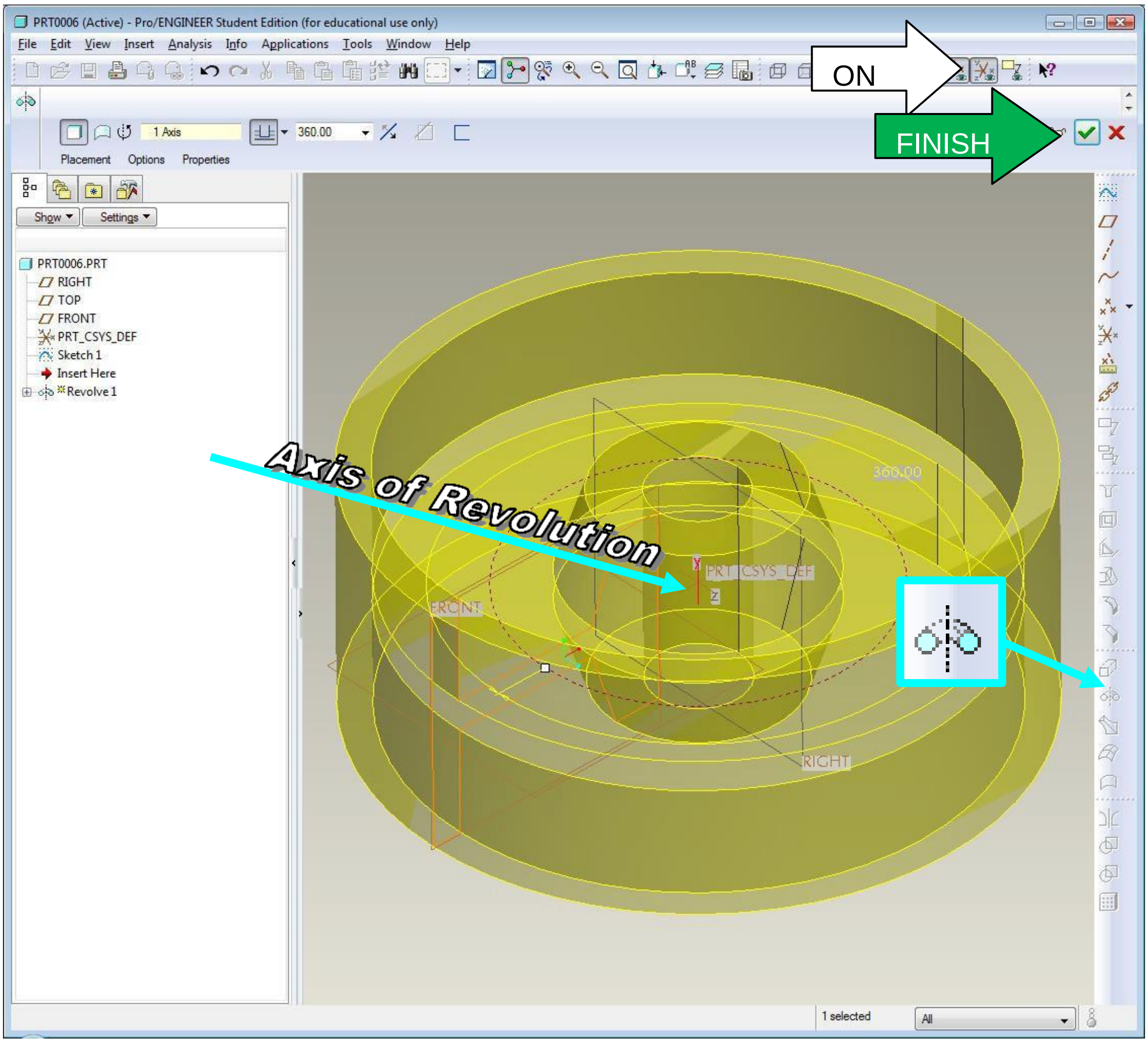
1. Using the dimension tool to create a ¼ of the geometry and then sweep it to the other side. Make sure you finish adding the dimensions.





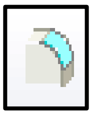


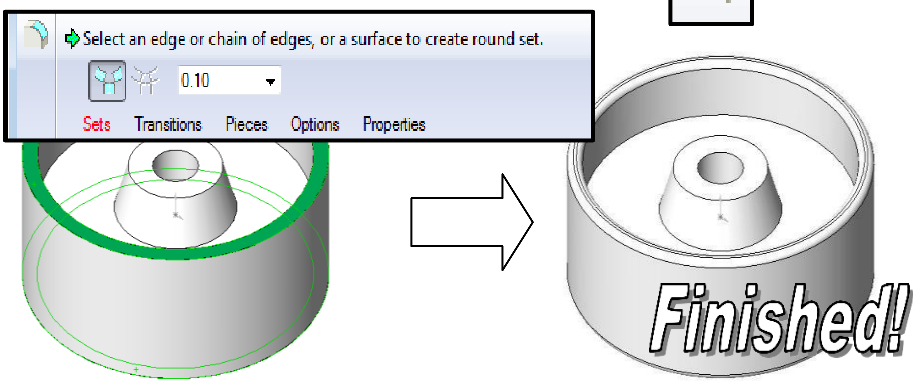
1. Select the **Revolve** feature icon.  Then select the axis/centerline.



Rounds

4. Select the top and bottom edges and add R.6б6_

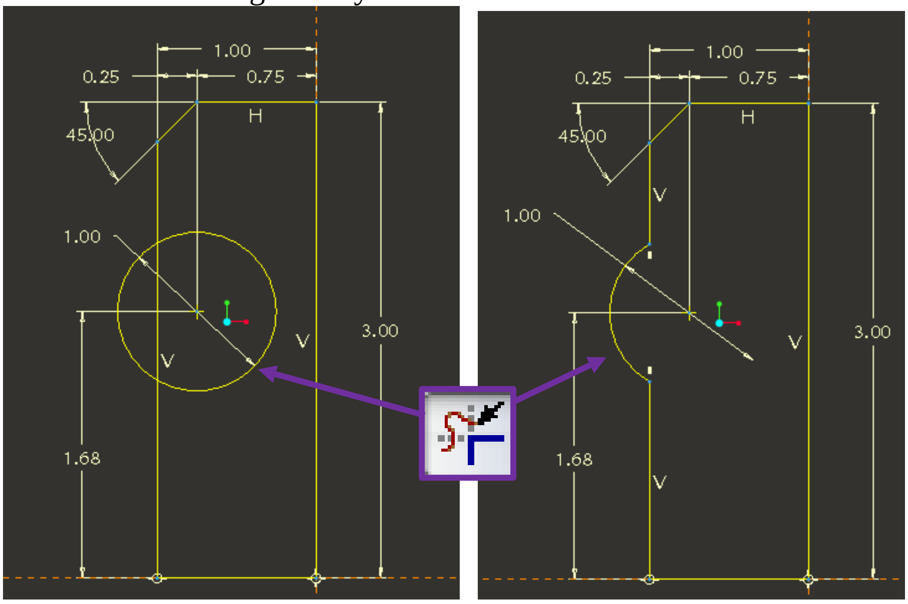




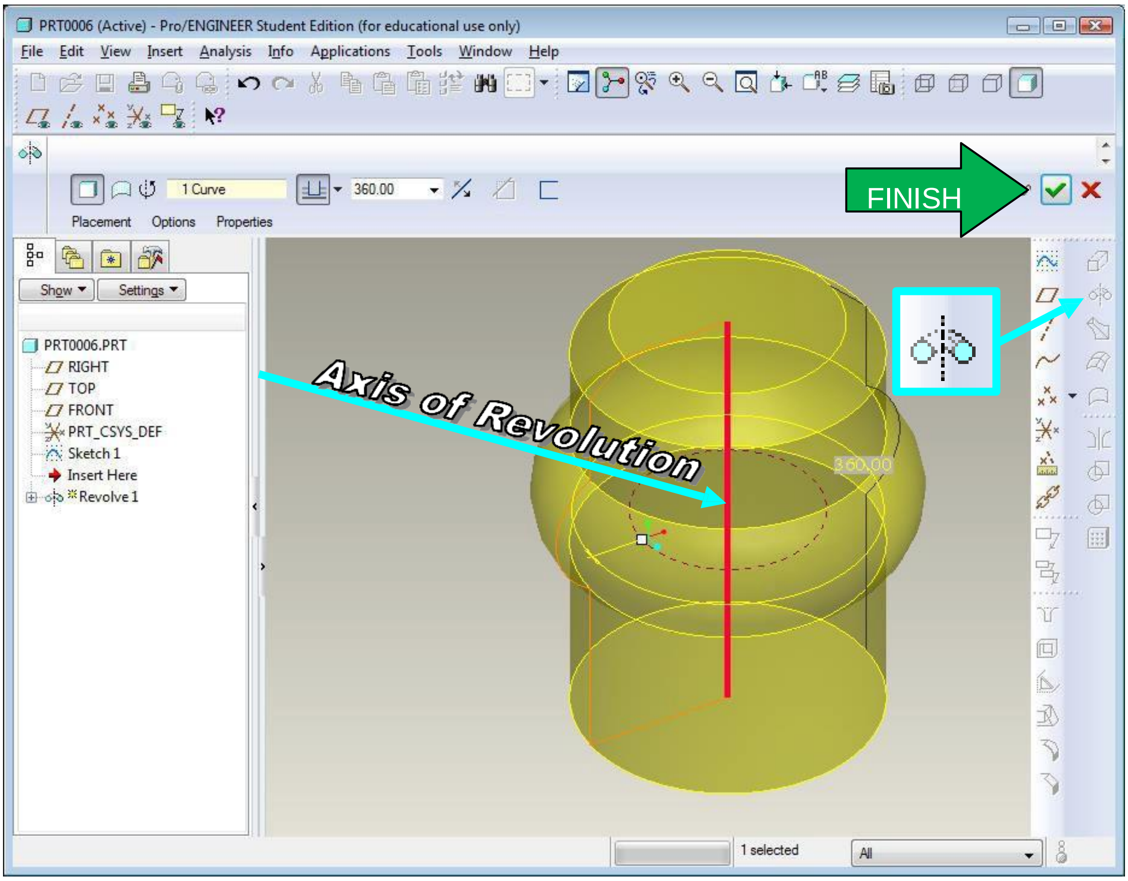
EXERCISE 4

Secondary Feature Modeling

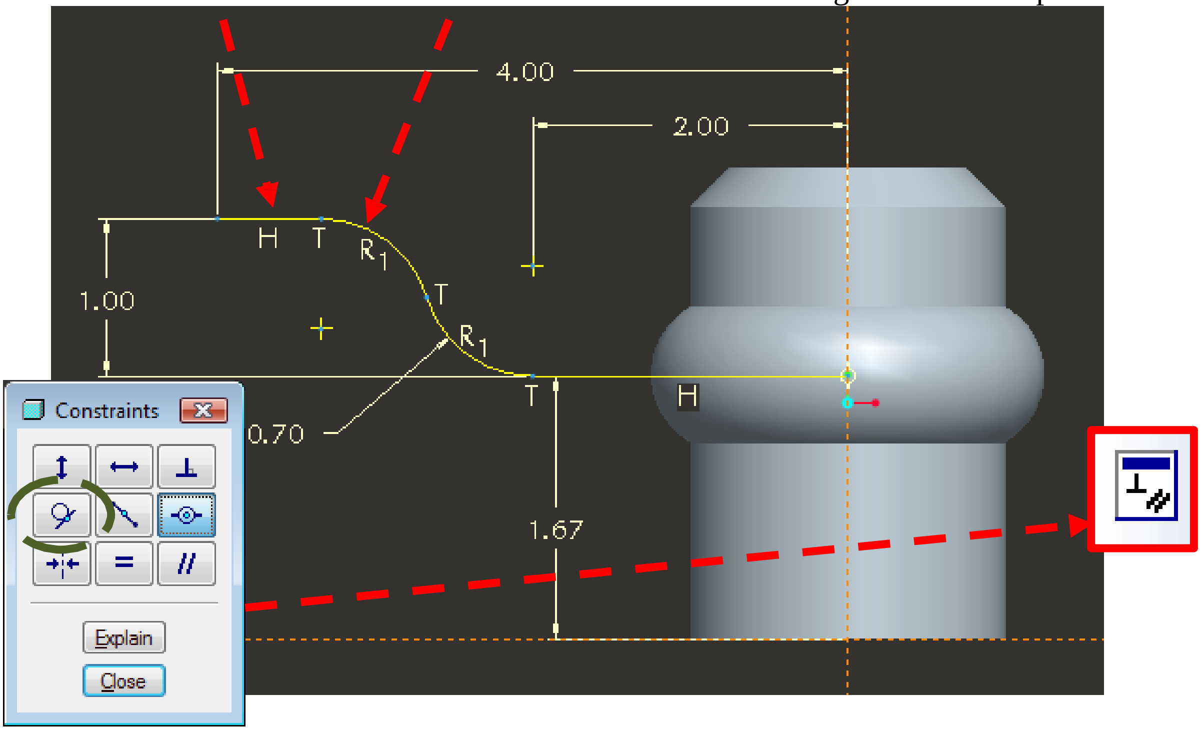
1. sketch the geometry as shown below Then **Trim.**



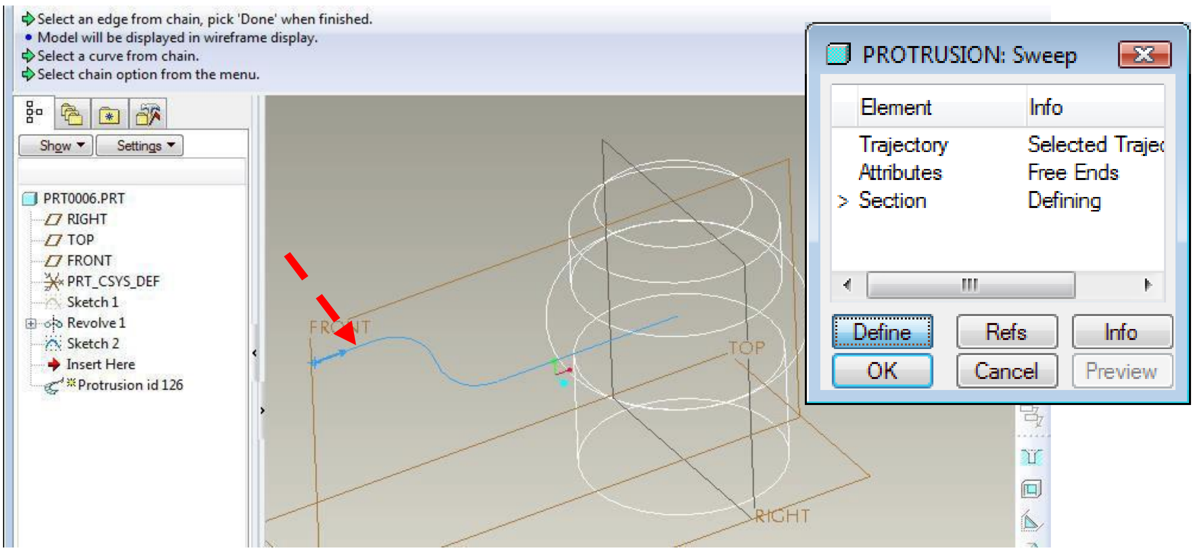
2. **Revolve**.



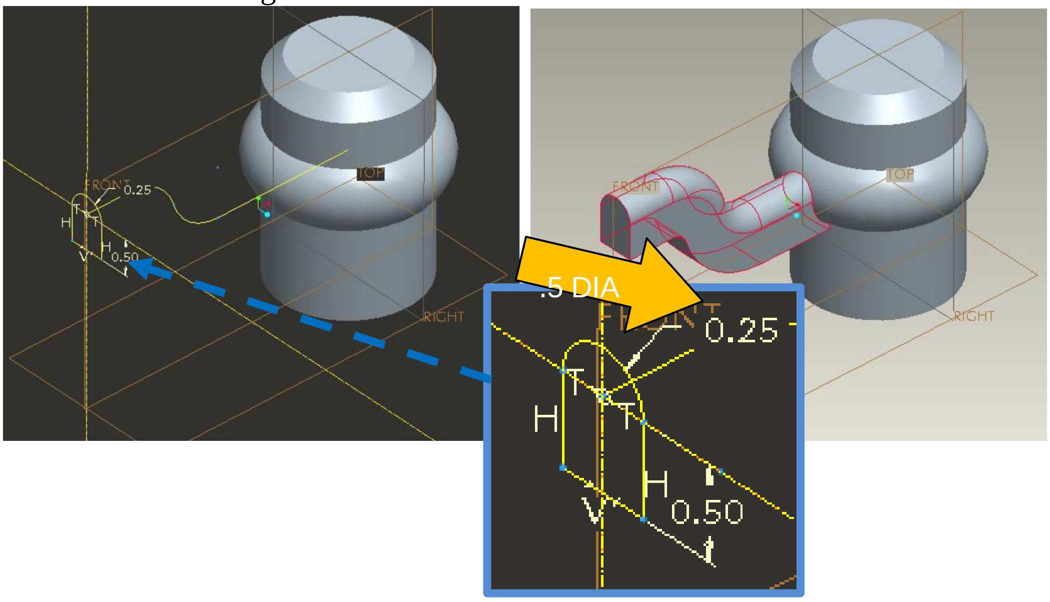
3. **Constraints:** Select the Front datum plane and sketch the following. Use the Constraint tool and select the **Tangent** option. Then select the left most horizontal line and the arc attached to it to establish a tangent relationship.



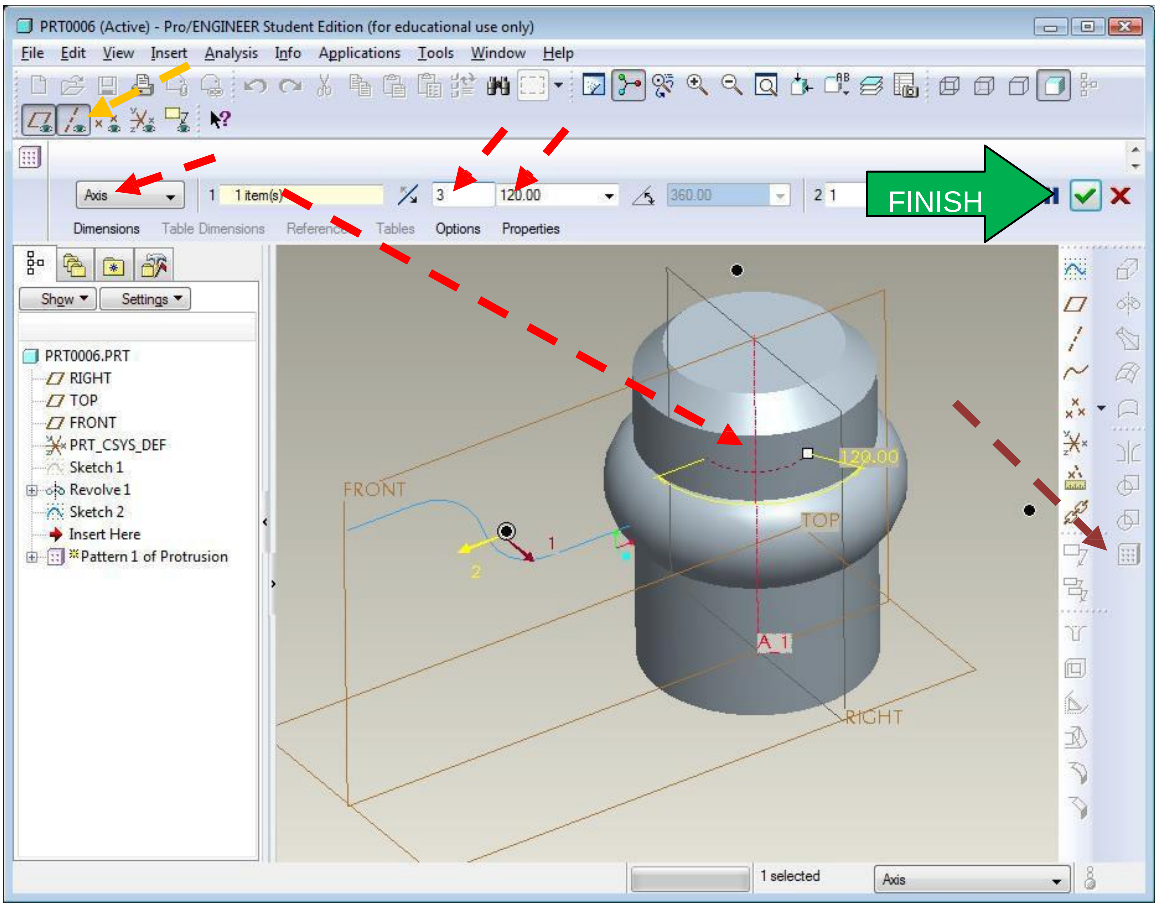
4. **Sweeps**: left side of the curve we just created to create a new sketch datum at the end.
5. Also select: SelectTraj/Curve Chain/Select All/Done/Done



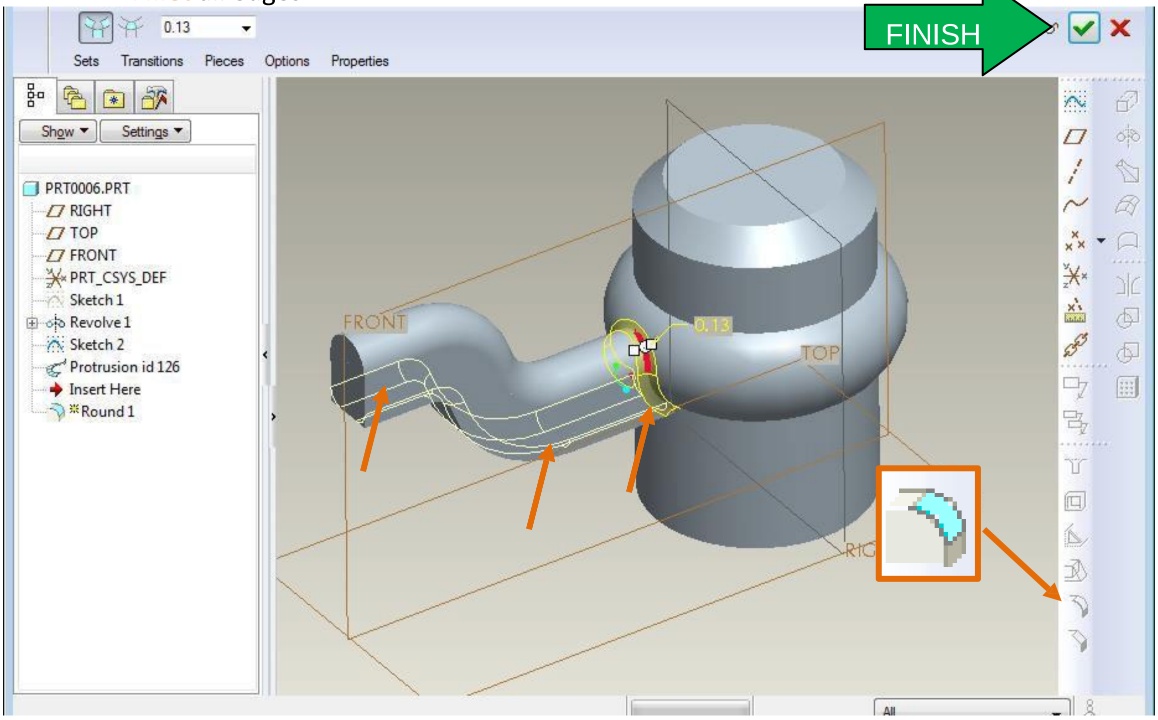
8.Draw the following sketch

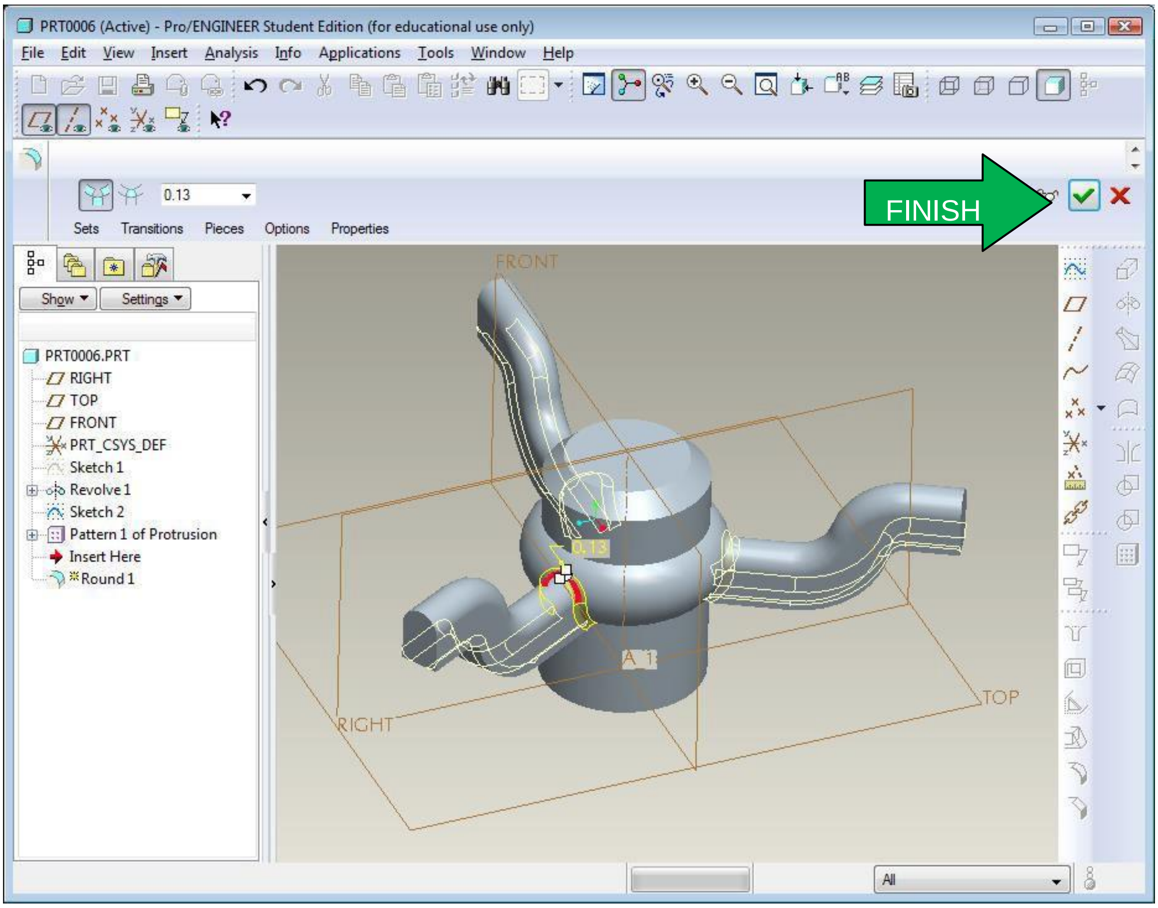


6. **Pattern** *Circular Pattern*: 360°/3 = 120° (*NOTE: First select the spoke to activate the icon.*) “SelectAxis also select the view axis

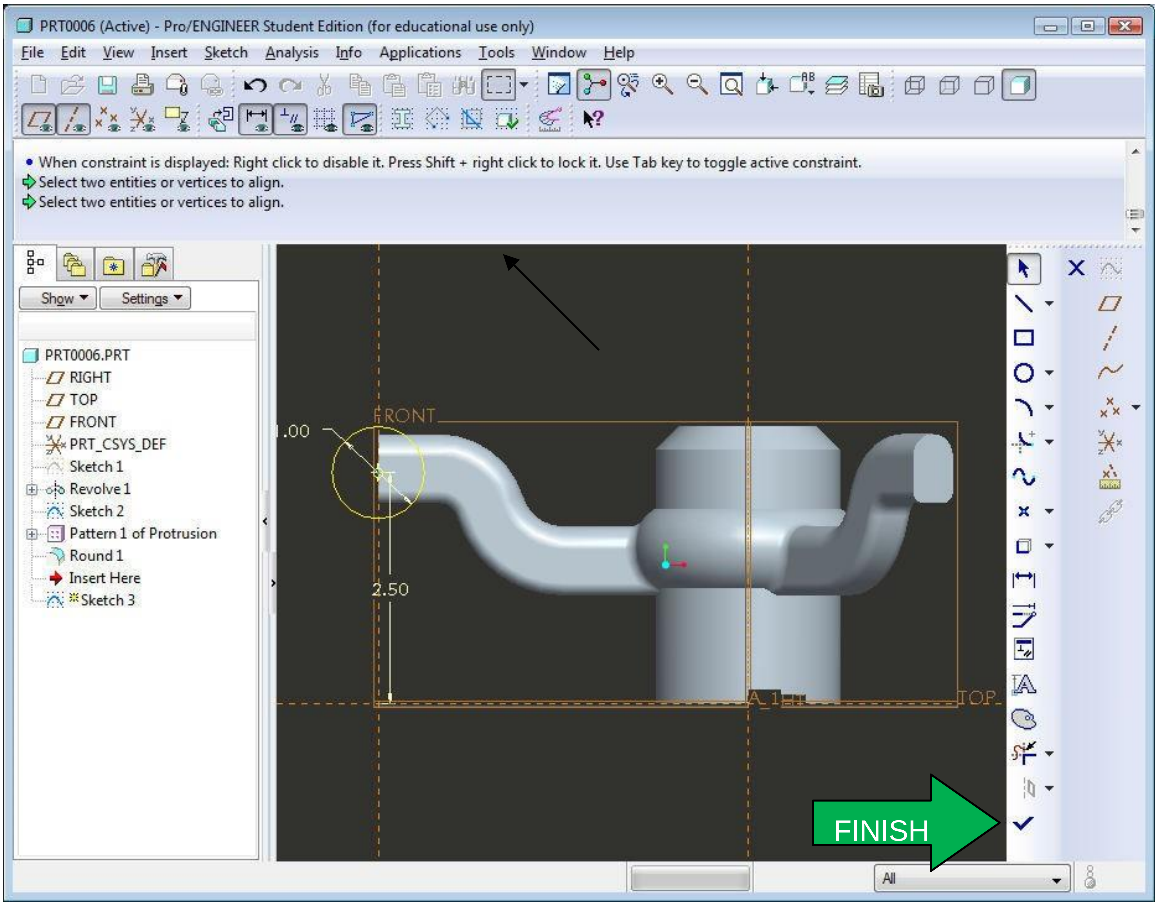


Fillet all edges

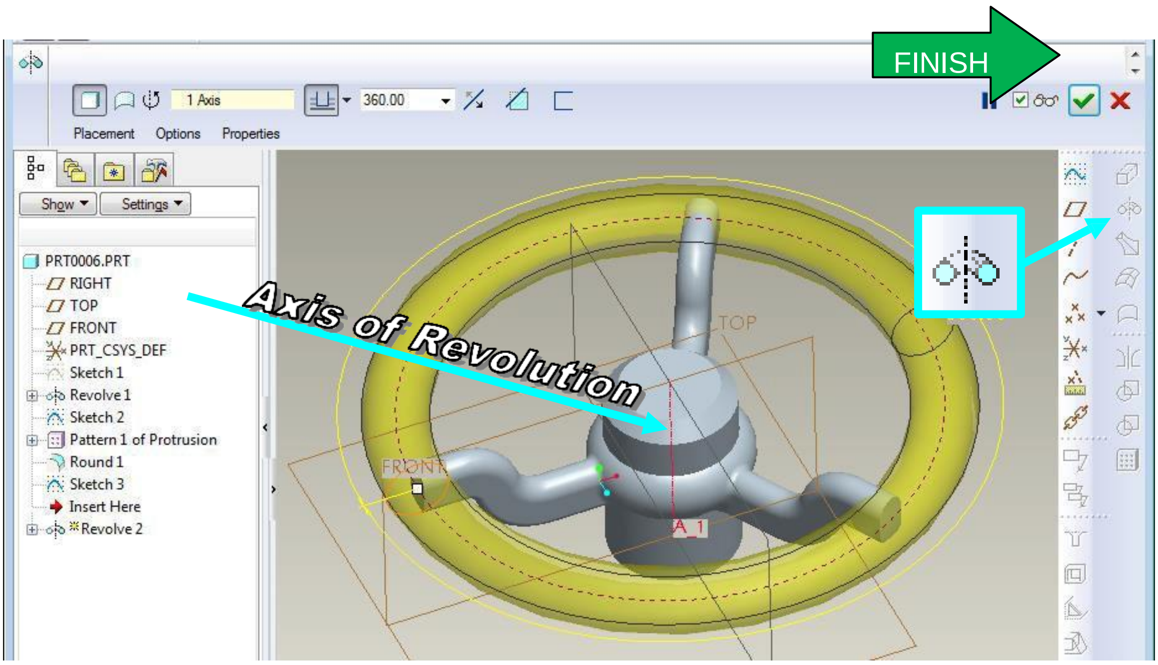




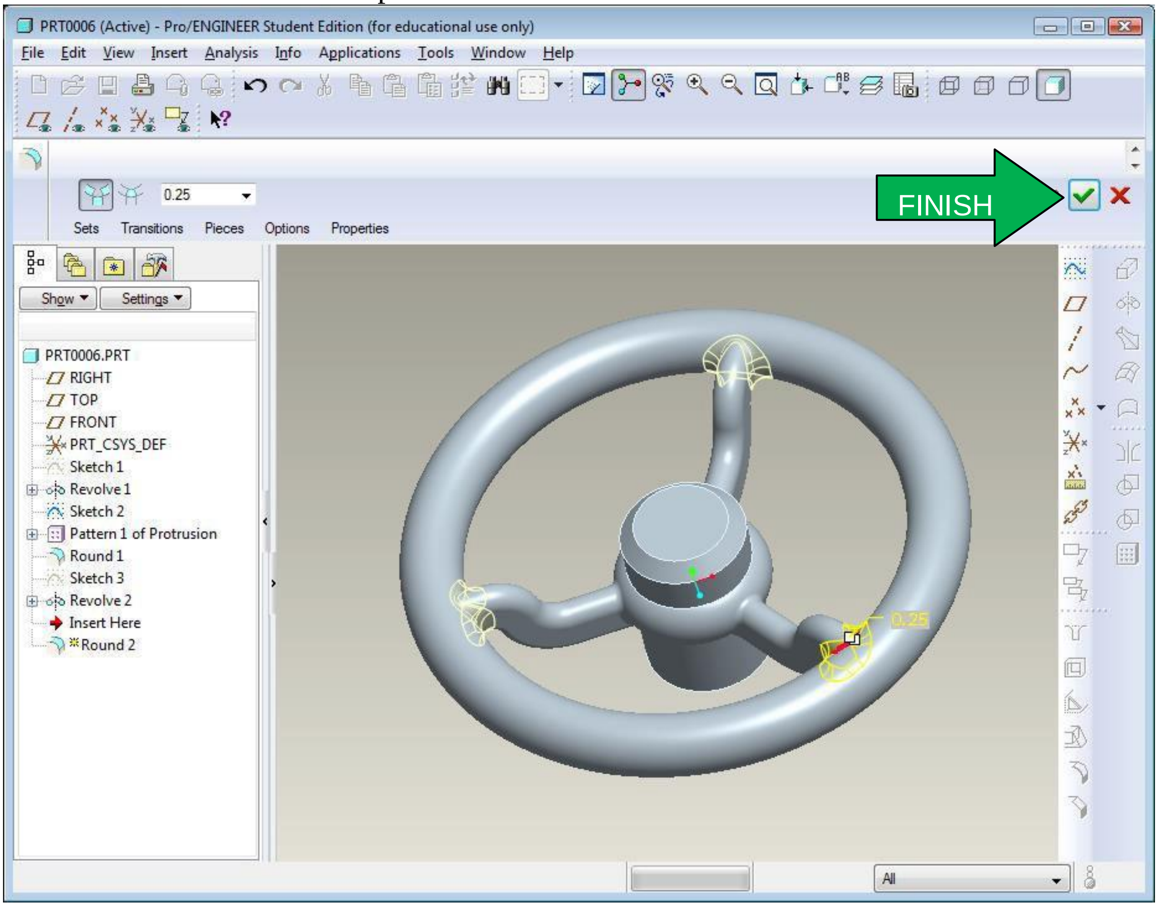
7. Rebuild after completion.



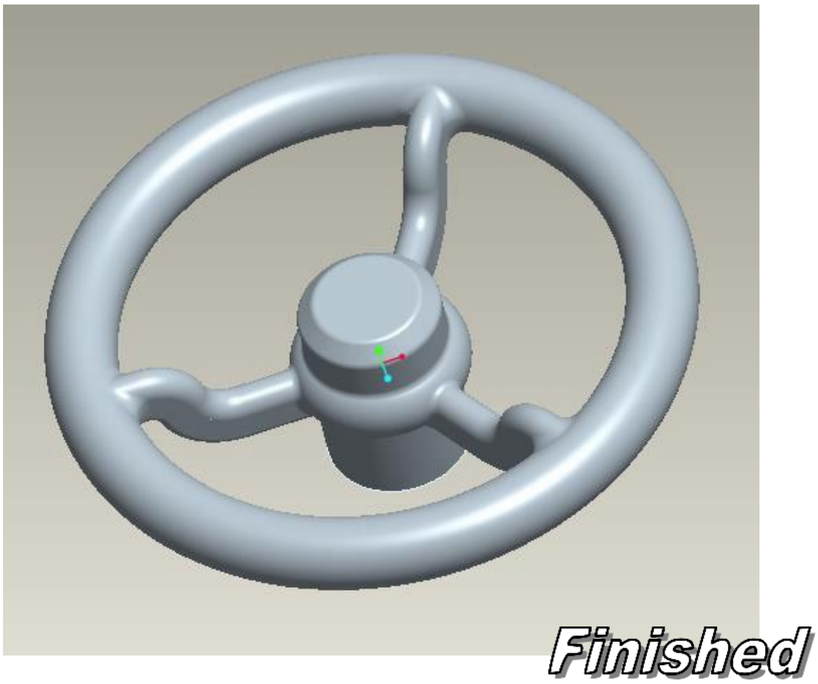
8. **REVOLVE**

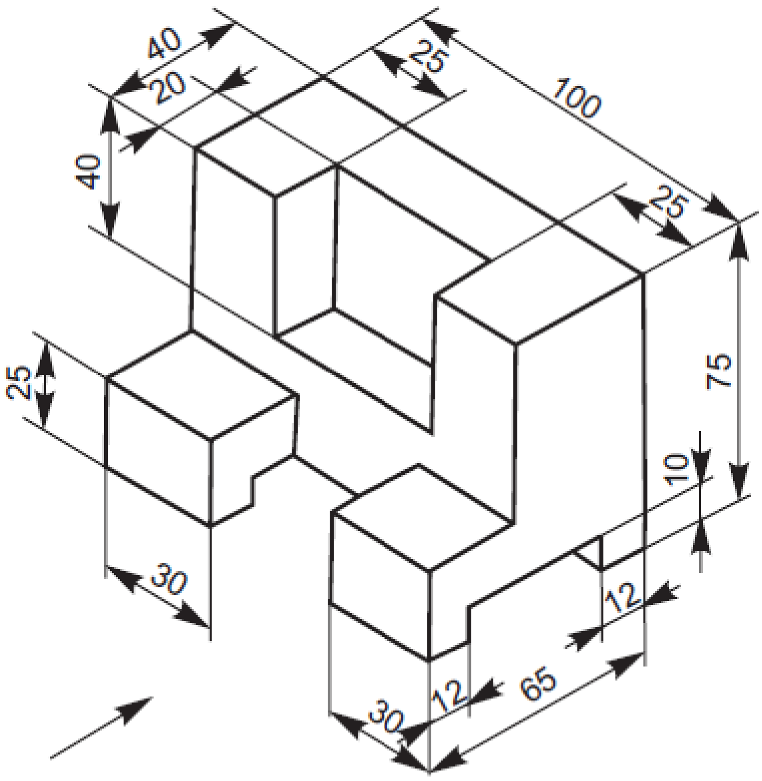


9. Add **Rounds** to the spoke – handle sections.



FINISHED

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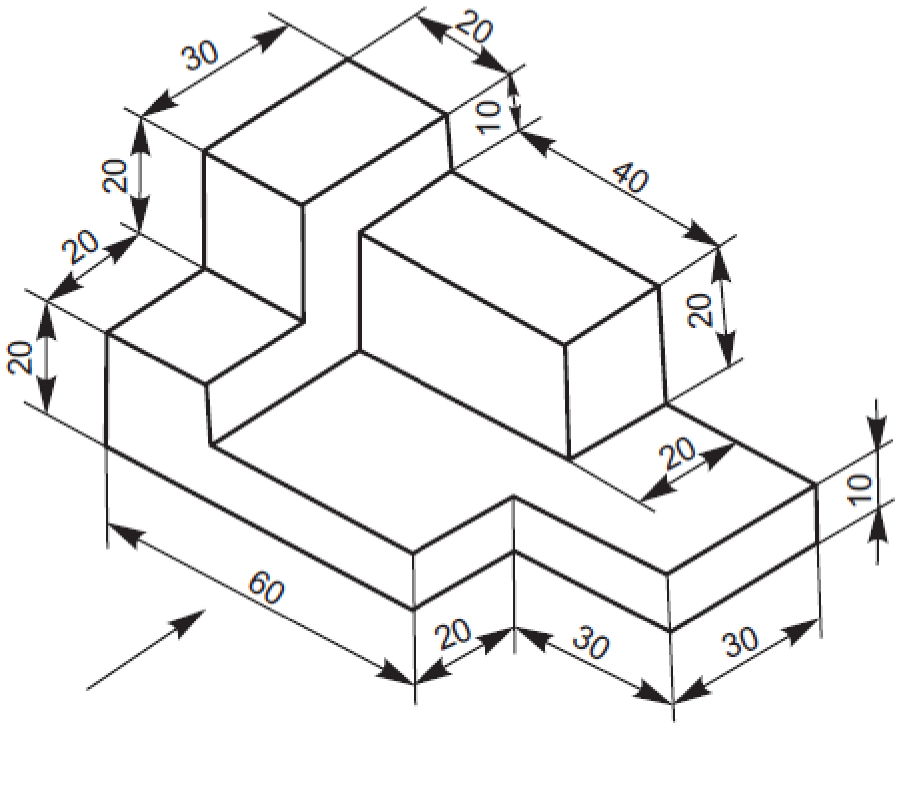
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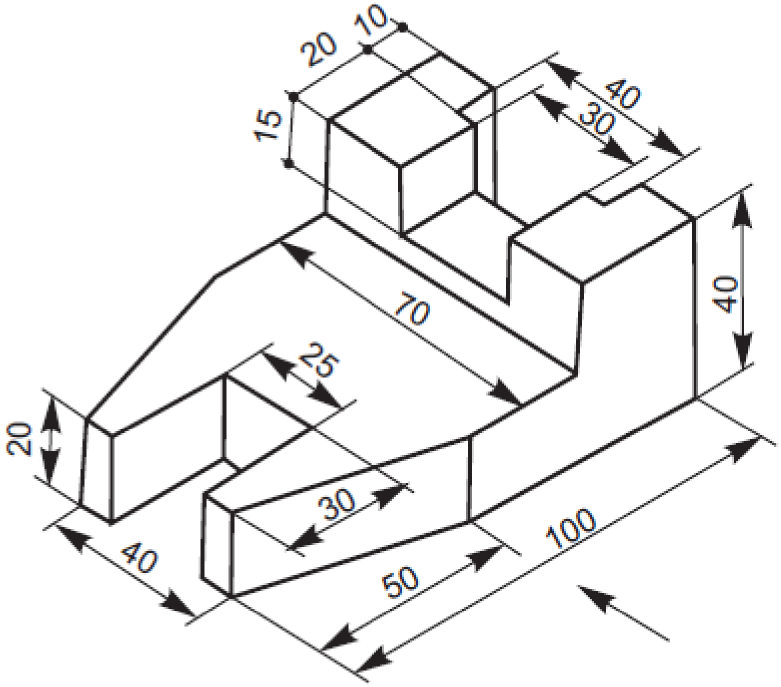

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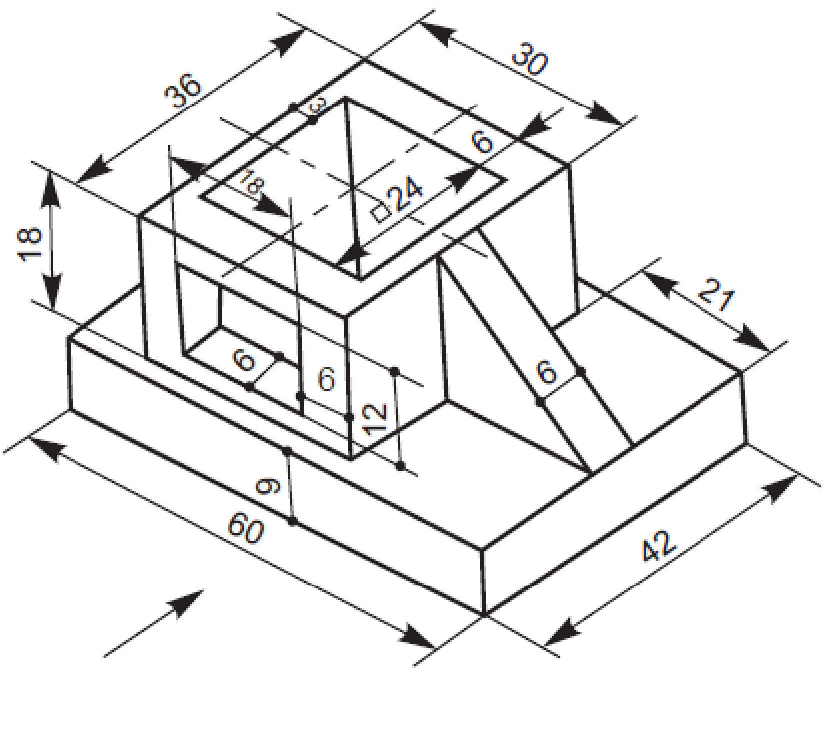

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