Concordia University
Engineering and Computer Science

MECH 211 – Mechanical Engineering Drawing

François Tardy

Credits: 3.5

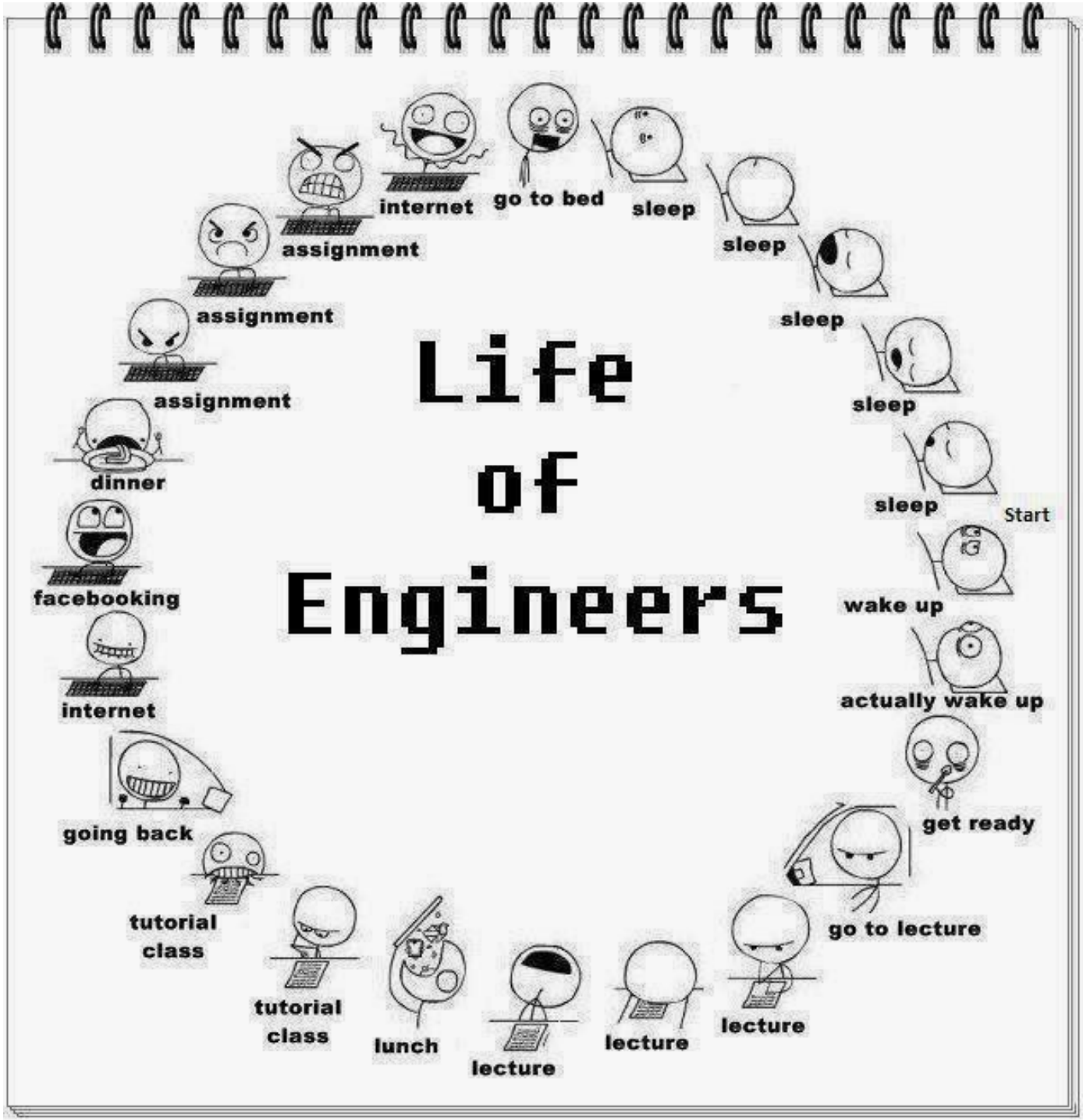
Lecture 9

Content of the lecture

- Dimensions
- Nomenclatures
- Dimensioning practices
- Dimensioning examples
- Dimensioning rules
  - Aligned dimensions
  - Dimensions outside the view
  - The extension line practice
  - The centerline

Content of the lecture

- Radial and diametrical dimensions
- Chained features
- Explanations
- Not to scale designations
- Reference of the extension lines
- General dimensioning
- Contour dimensioning
- Size vs. location


Life
of
Engineers
Start
wake up
actually wake up
get ready
go to lecture
lecture
lecture
lecture
lunch
tutorial
class
tutorial
class
going back
internet
facebooking
dinner
assignment
assignment
assignment
internet
go to bed
sleep
sleep
sleep
sleep
sleep

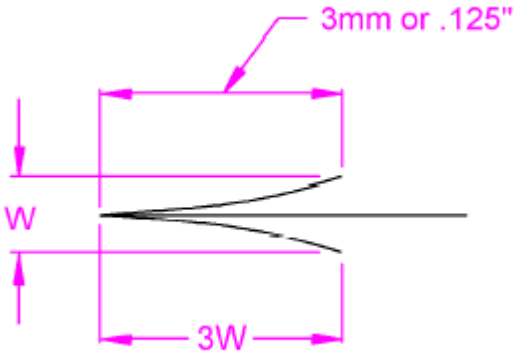
Dimensions

Dimensions are the numerical value that define the size, shape, location, surface texture, or geometric characteristic of a feature.

What should be defined before dimensioning?

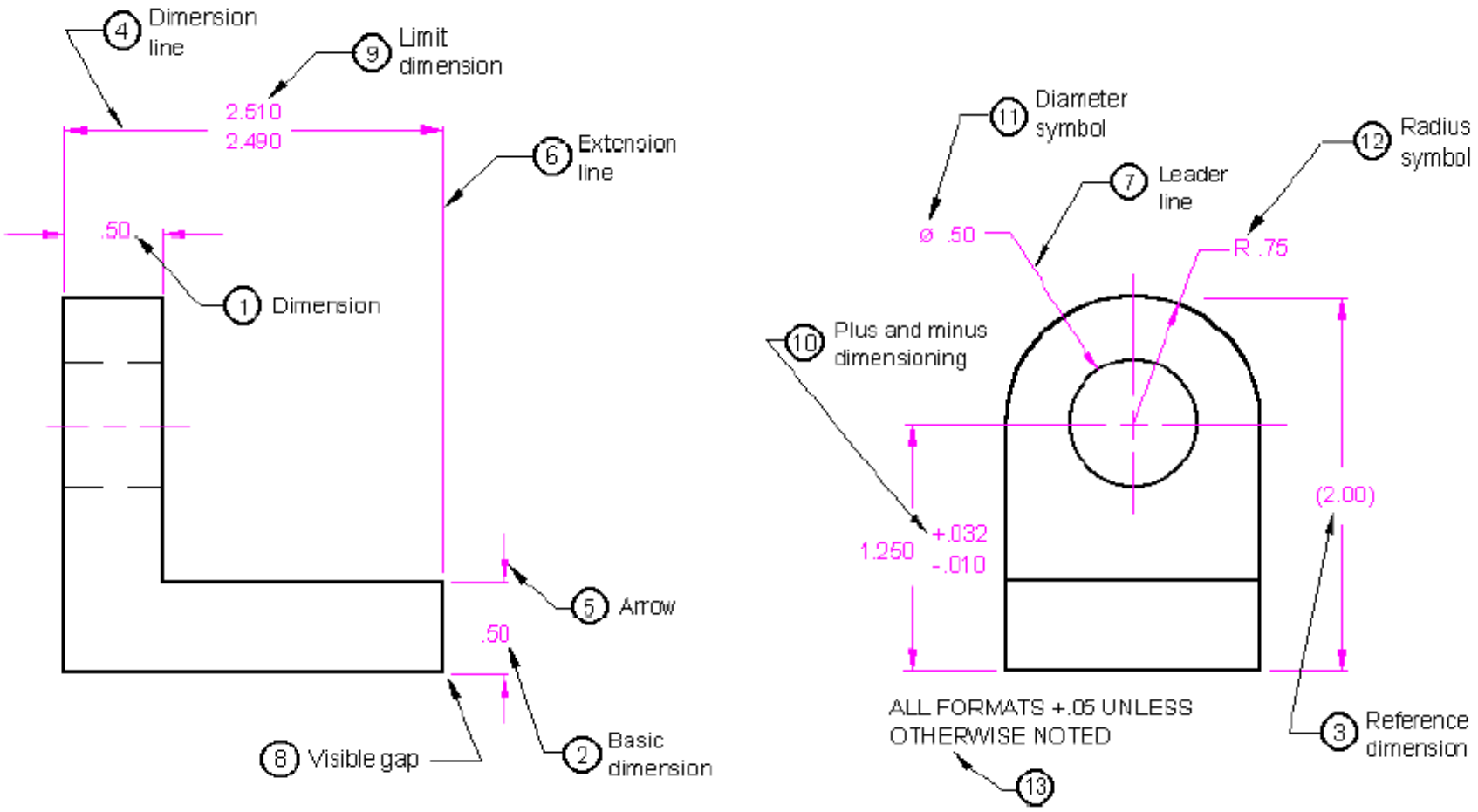
- Perfect understanding of the shapes
- Units
- Minimizing the data to geometrically define the part
- Features to be defined
- Dimensioning rules

Start with a base of reference.



Dimensions – cont'd

- Shape, location, relative position of features in one part.
- Locate the position of the tool with respect to the blank – manufacturing hints
- Feature sizes (dimensions: linear, angular)
- Position or location (linear, angular)


4 Dimension line
9 Limit dimension
2.510
2.490
6 Extension line
.50
1 Dimension
5 Arrow
.50
8 Visible gap
2 Basic dimension
11 Diameter symbol
12 Radius symbol
7 Leader line
ø .50
R .75
10 Plus and minus dimensioning
1.250 +.032 -.010
(2.00)
ALL FORMATS +.05 UNLESS OTHERWISE NOTED
3 Reference dimension
13

Nomenclature

- **Dimension**: A numerical value which defines size or relative position.
- **Basic dimension**: Theoretically exact size of the feature.
- **Reference dimension**: Dimension not direct used, but indicated.
- **Dimension line**: Line that shows the extent and the direction of the feature.
- **Arrow**: Symbol @ the end of dimension lines.

Nomenclature cont’d

Extension line: Line that shows which feature is associated with the size

Visible gap: Gap between corners of the feature and extension lines

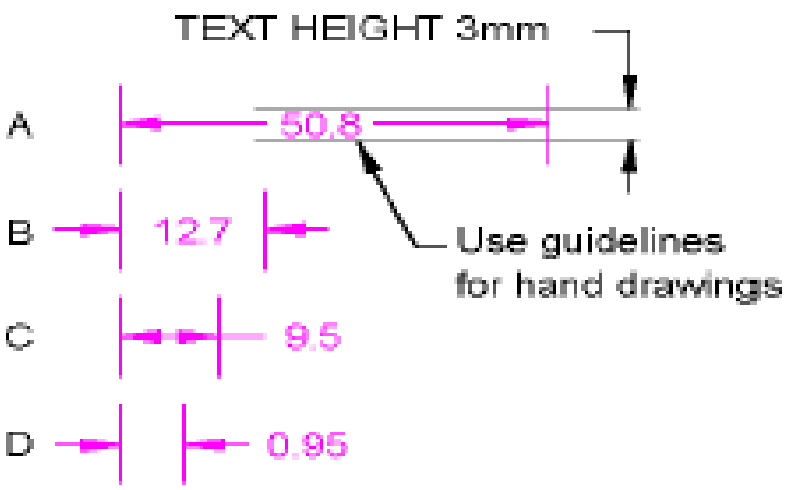
Leader line: Extension line that shows the size of a inaccessible feature

Diameter/Radius symbols: d/r followed by a number, the size of the feature

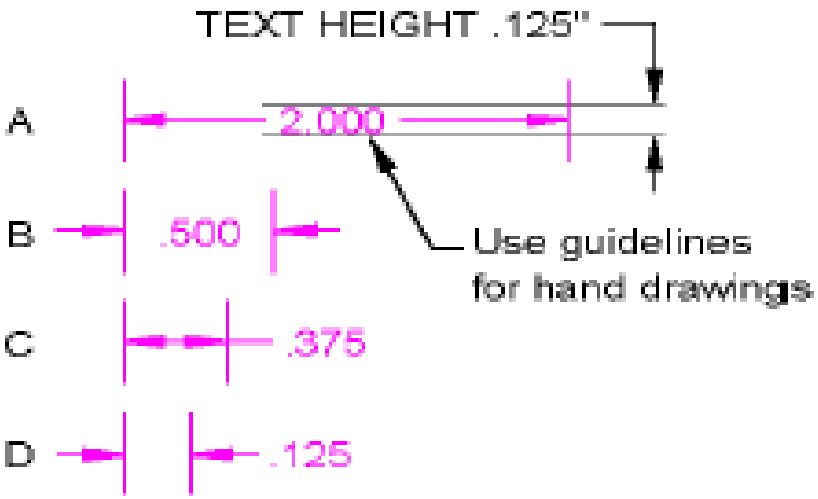
Datum: A reference line/surface for datum dimensioning.

Dimensioning Practice

- Texts are 3mm or 0.125’’ high
- The space between lines of text is 1.5 mm or 0.0625’’



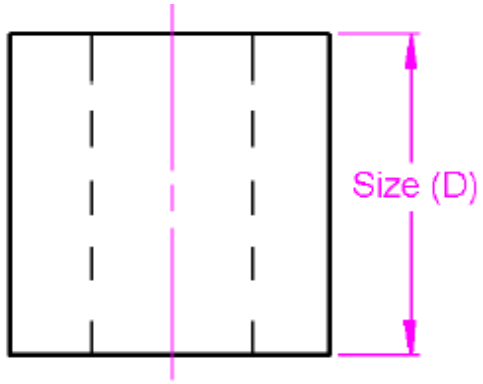
Millimeter dimensioning

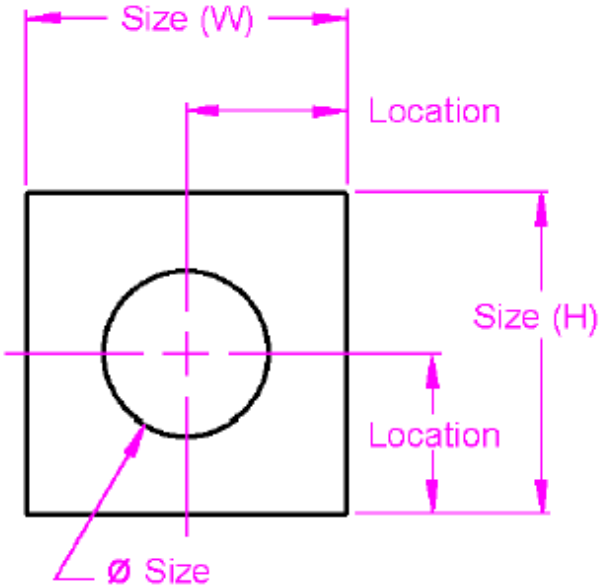


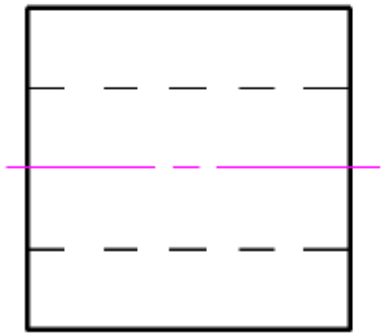
Decimal dimensioning

Dimensioning Practice

REMEMBER
Sizes and dimensions are used for manufacture purposes







Dimensioning Practice – cont'd

Size dimensions

1. Horizontal
2. Vertical
3. Diameter
4. Radius

Position dimensions

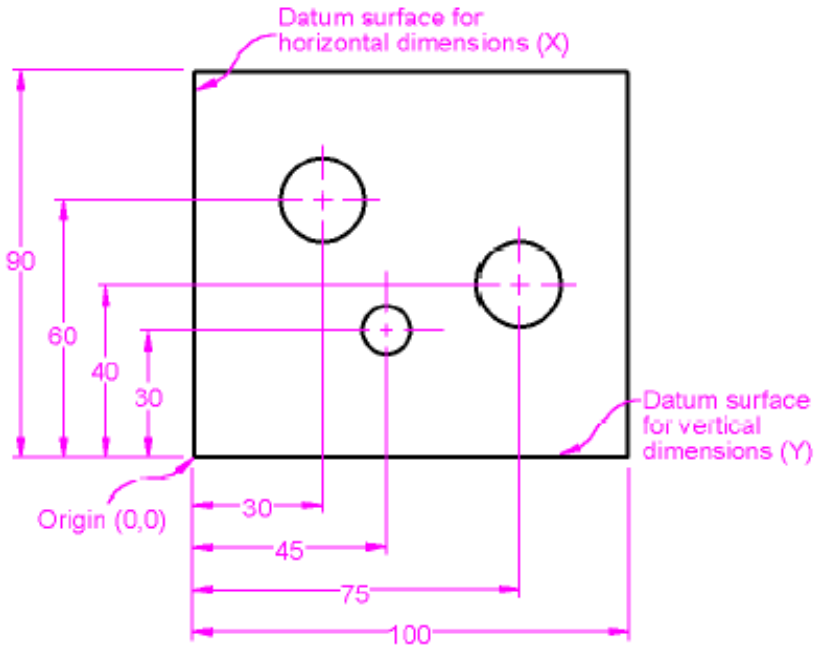
Horizontal position

Vertical position

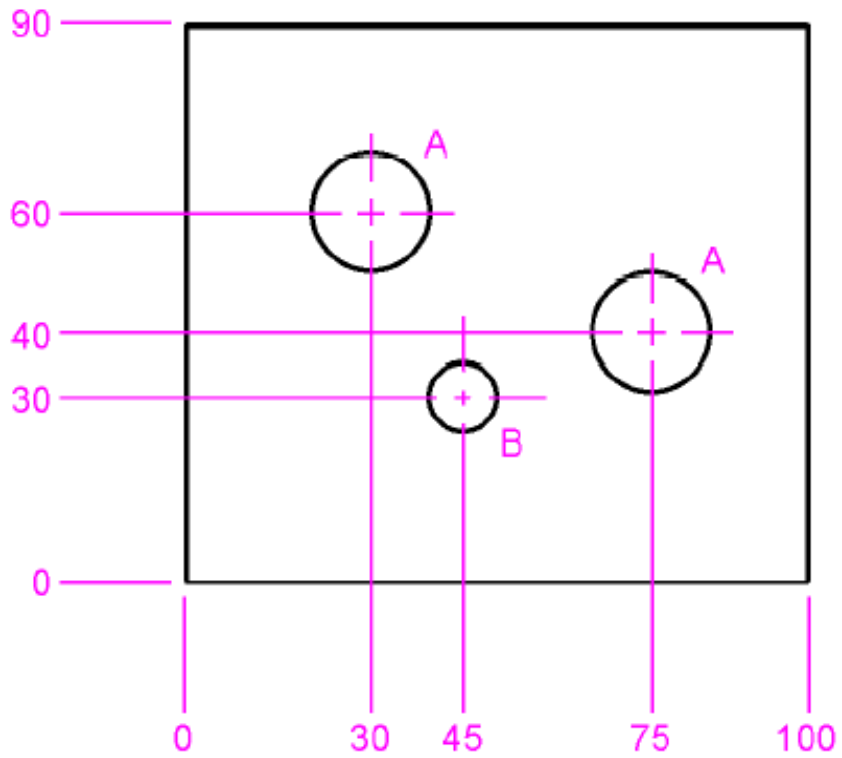
Angle

- Standard practice in dimensioning – promotion of clarity.
- Make sure that the part can be built to dimensions.
- The provided dimensions must be measurable.

Dimensioning Example



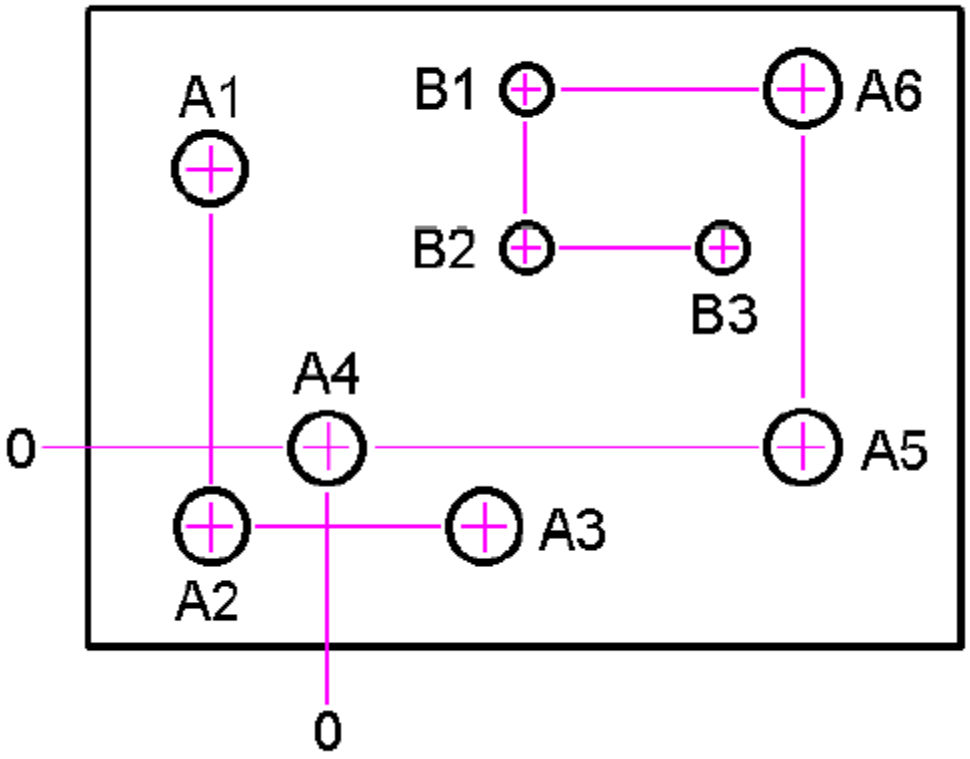
Symbol	A	B
Hole diameter	20	10



Tabular dimensioning

Dimensioning Example

Coordinate Dimensioning



Hole	X	Y	Size
A1	-1.00	2.00	Ø.50
A2	-1.00	- .50	Ø.50
A3	1.10	- .50	Ø.50
A4	0	0	Ø.50
A5	3.38	0	Ø.50
A6	3.38	2.62	Ø.50
BI	1.50	2.62	Ø.25
B2	1.50	1.50	Ø.25
B3	2.88	1.50	Ø.25

Dimensioning Rules

- Each feature of an object is dimensioned once and only once.
- Dimensions should be selected to suit the function of the object.
- Dimensions should be placed in the most descriptive view of the feature being dimensioned.
- Dimensions should specify only the size of a feature. The manufacturing method should only be specified if it is a mandatory design requirement.
- Angles shown on drawings as right angles are assumed to be 90 degrees unless otherwise specified, and they need not be dimensioned.

Dimensioning Rules cont'd

- Dimensions should be located outside the boundaries of the object whenever possible.
- Dimension lines should be aligned and grouped where possible to promote clarity and uniform appearance.
- Crossed dimension lines should be avoided whenever possible. When dimension lines must cross, they should be unbroken.
- The space between the first dimension line and the object should be at least 3/8 inch (10mm). The space between dimension lines should be at least ¼ inch (6mm).
- There should be a visible gap between the object and the origin of an extension line.

Dimensioning Rules cont'd

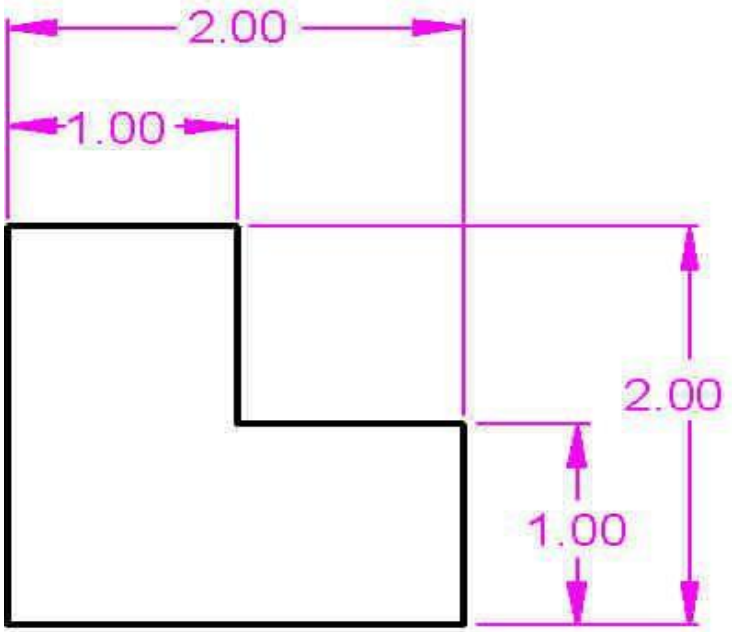
- Extension lines should extend 1/8 inch (3mm) beyond the last dimension line.
- Extension lines should be broken if they cross or are close to arrowheads.
- Leader lines used to dimension circles or arcs should be radial.
- Dimensions should be oriented to be read from the bottom of the drawing.
- Diameters are dimensioned with a numerical value preceded by the diameter symbol.

Dimensioning Rules cont'd

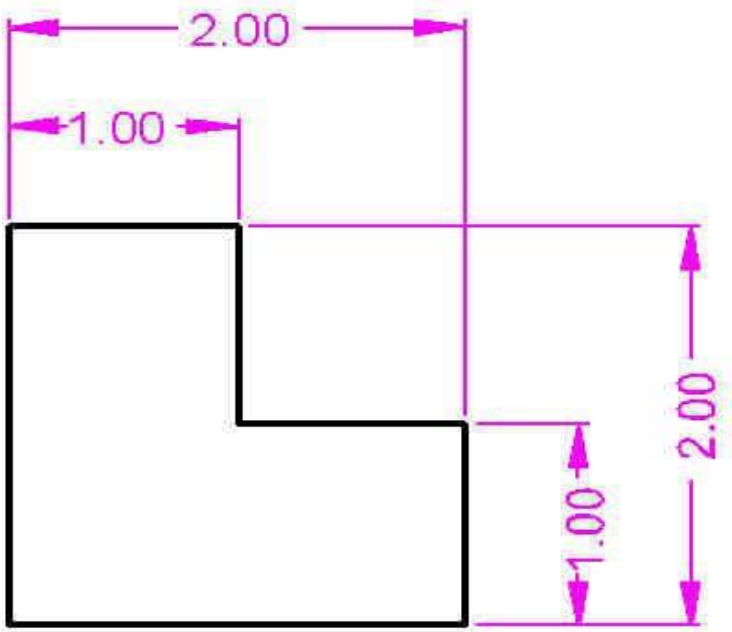
- Concentric circles should be dimensioned in a longitudinal view whenever possible.
- Radii are dimensioned with a numerical value preceded by the radius symbol.
- When a dimension is given to the center of an arc or radius, a small cross is shown at the center.
- The depth of a blind hole may be specified in a note. The depth is measured from the surface of the object to the deepest point where the hole still measures a full diameter in width.
- Counterbored, spotfaced, or countersunk holes should be specified in a note.

Aligned Dimensions

Aligned Dimensions have text placed parallel to the dimension line, with vertical dimensions read from the right of the drawing.

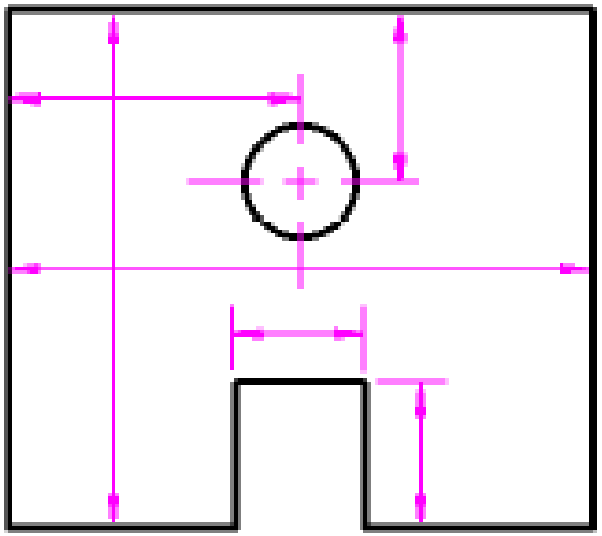


Unidirectional
Current standard

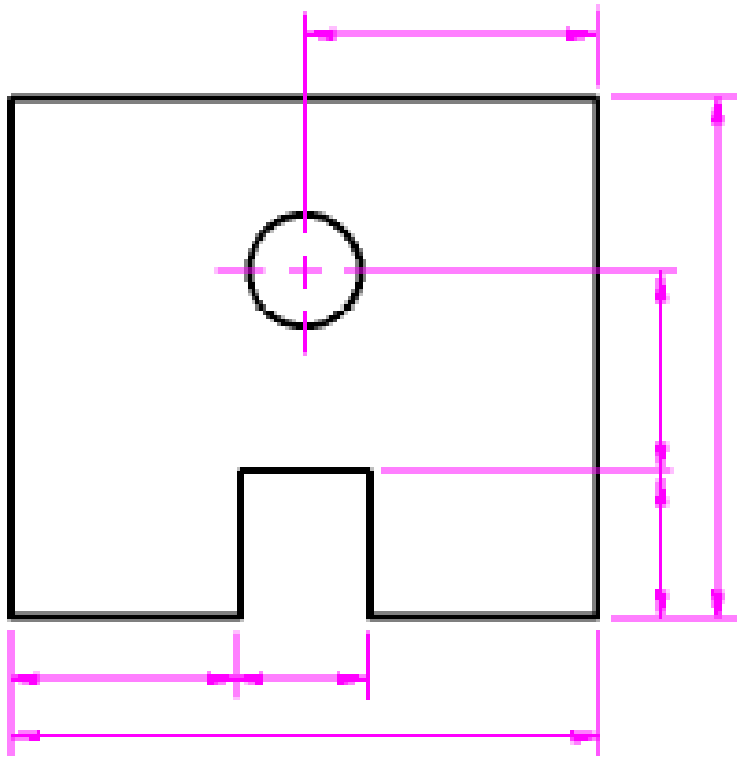


Aligned
Old standard

Dimension Outside the View

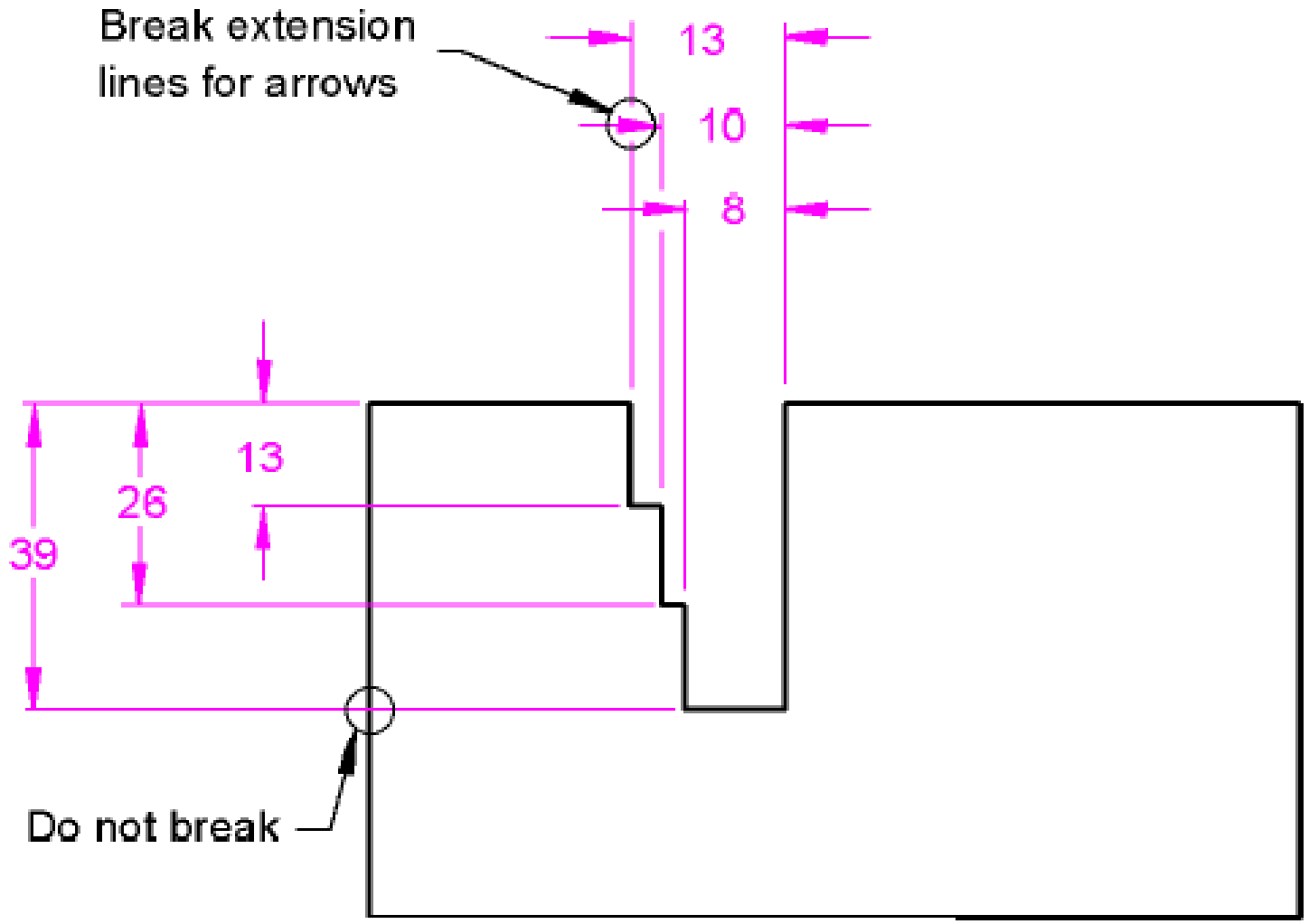


(A)
Avoid



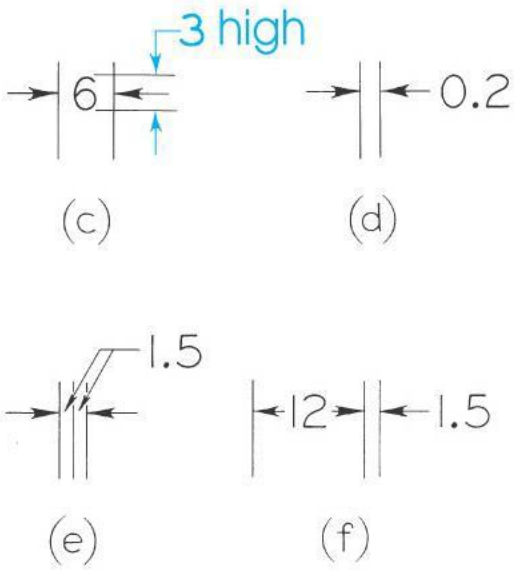
(B)
Good Practice

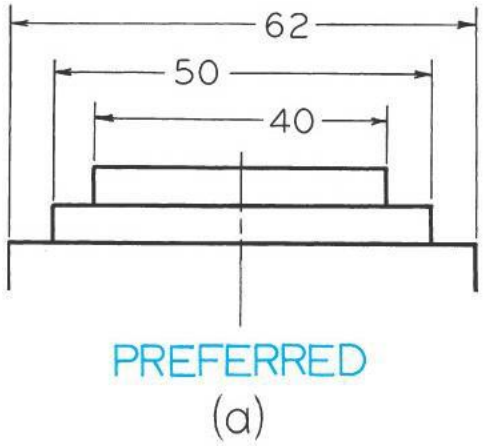
Extension Line Practice

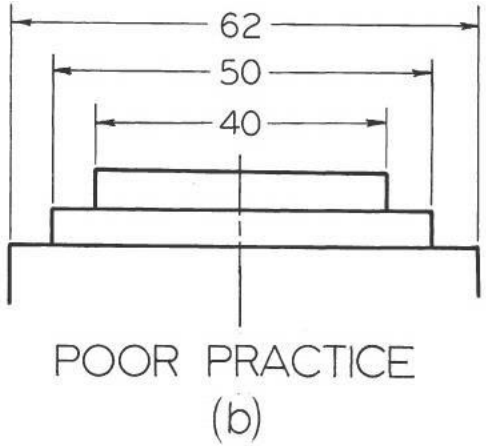


Extension Line Practice

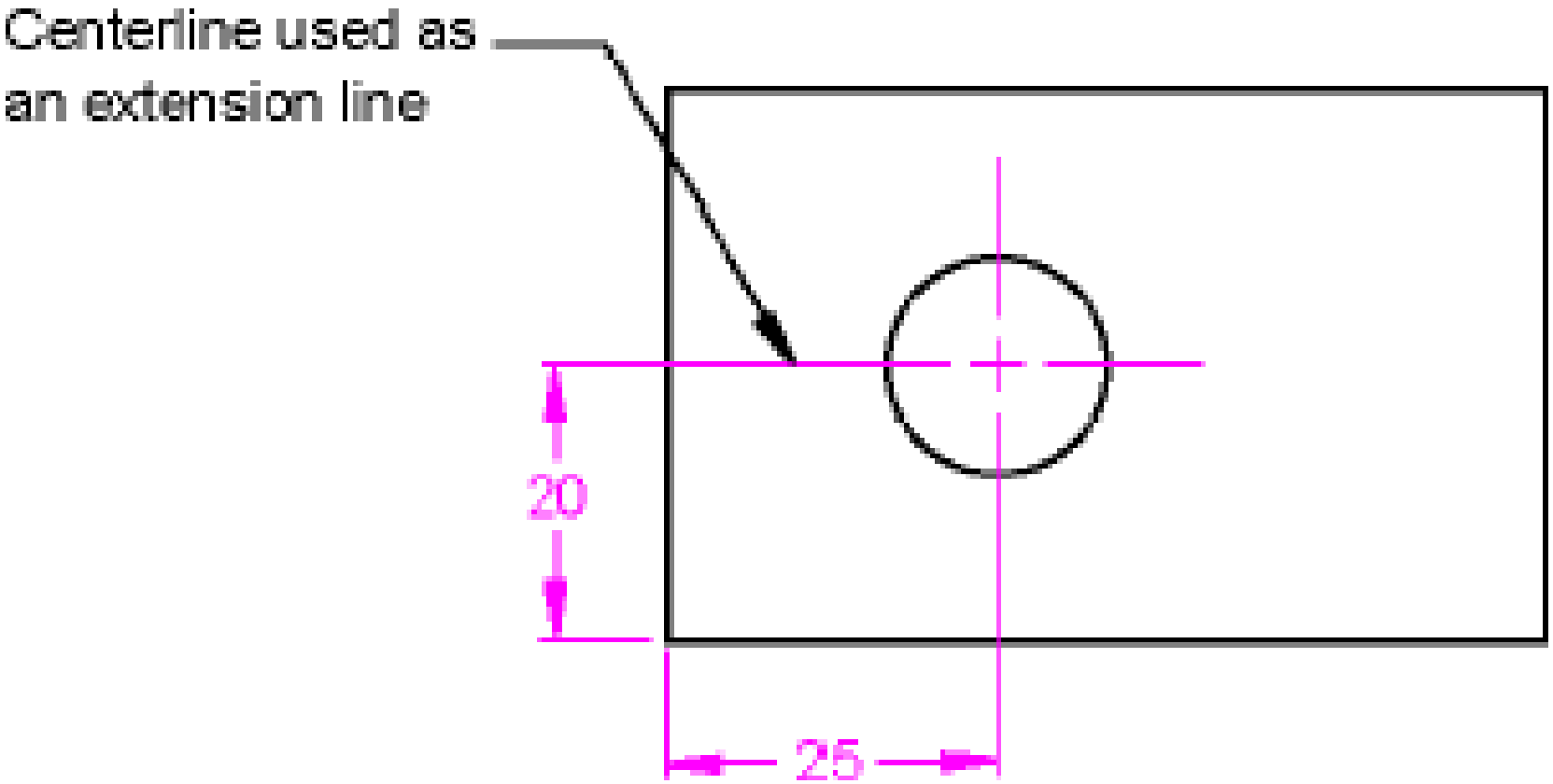
- Use any of the four methods, as long as they are readable.
- Stagger dimensions when grouping.
- Do not break dimension lines for object lines.





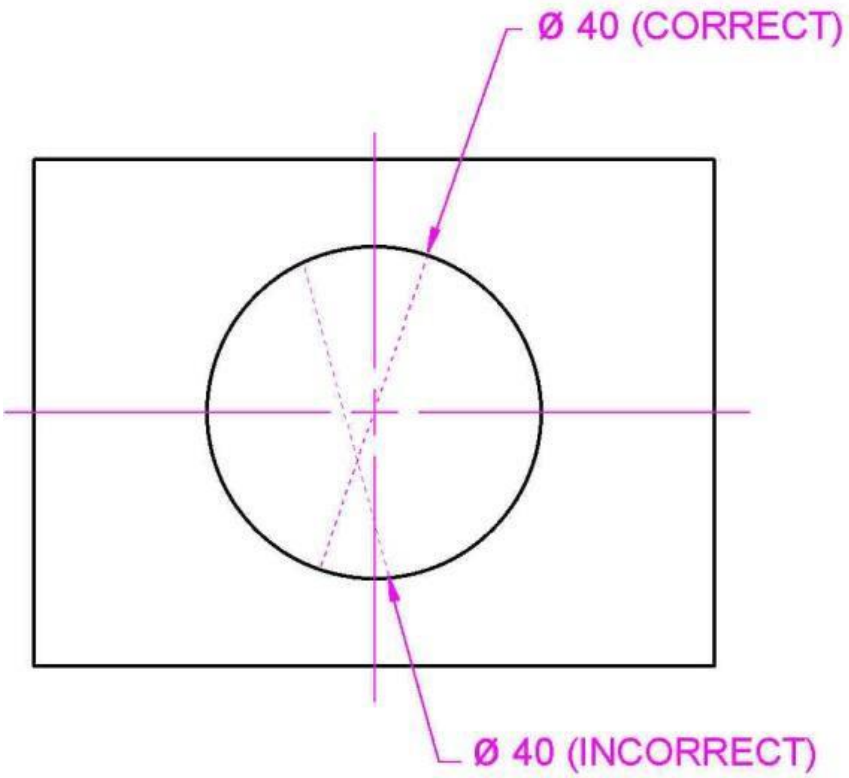


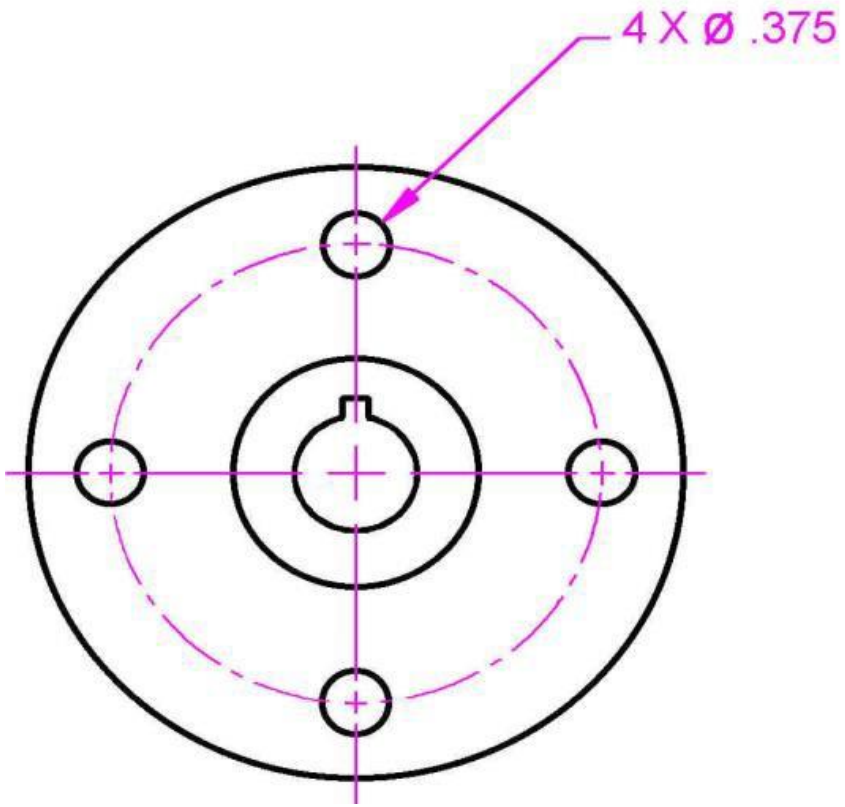
The Center Line Practice



Radial and Diametrical Dimensions

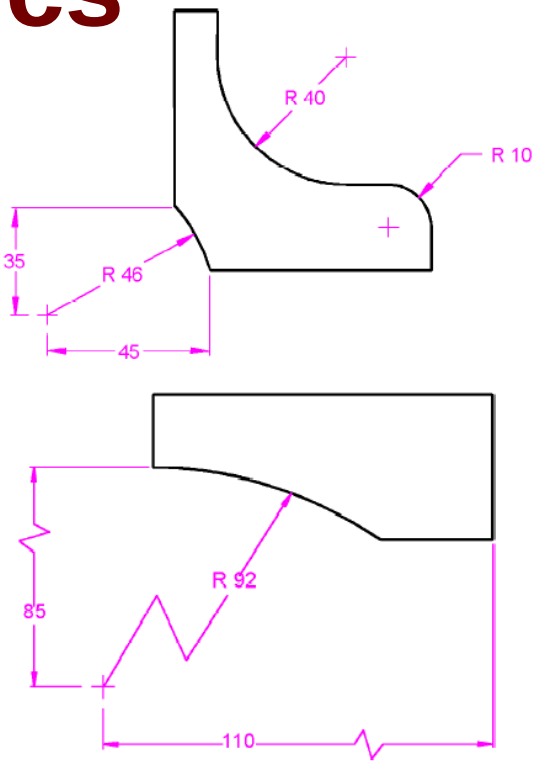
- More than half a circle: **diameter**
- Less than half a circle: **radius**
- Leaders to point towards centre of the circle or arc (Radial)

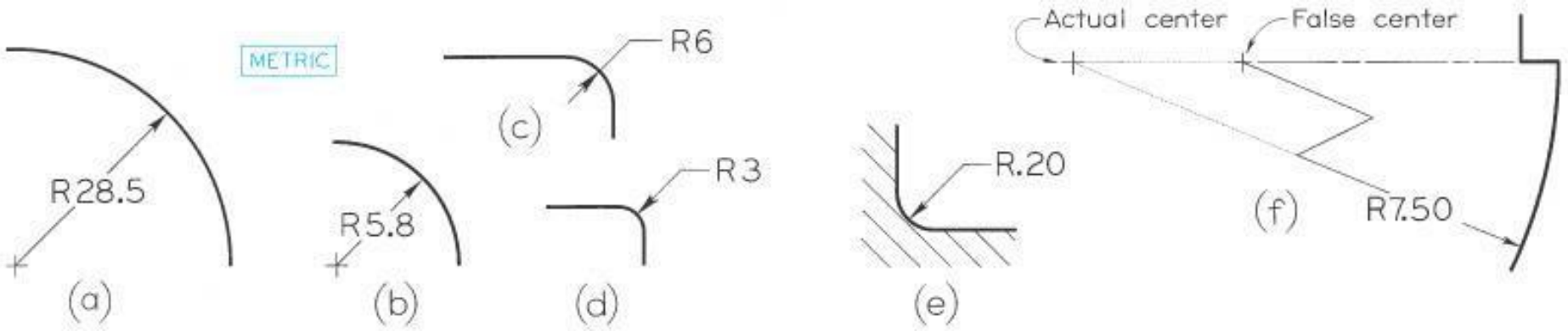




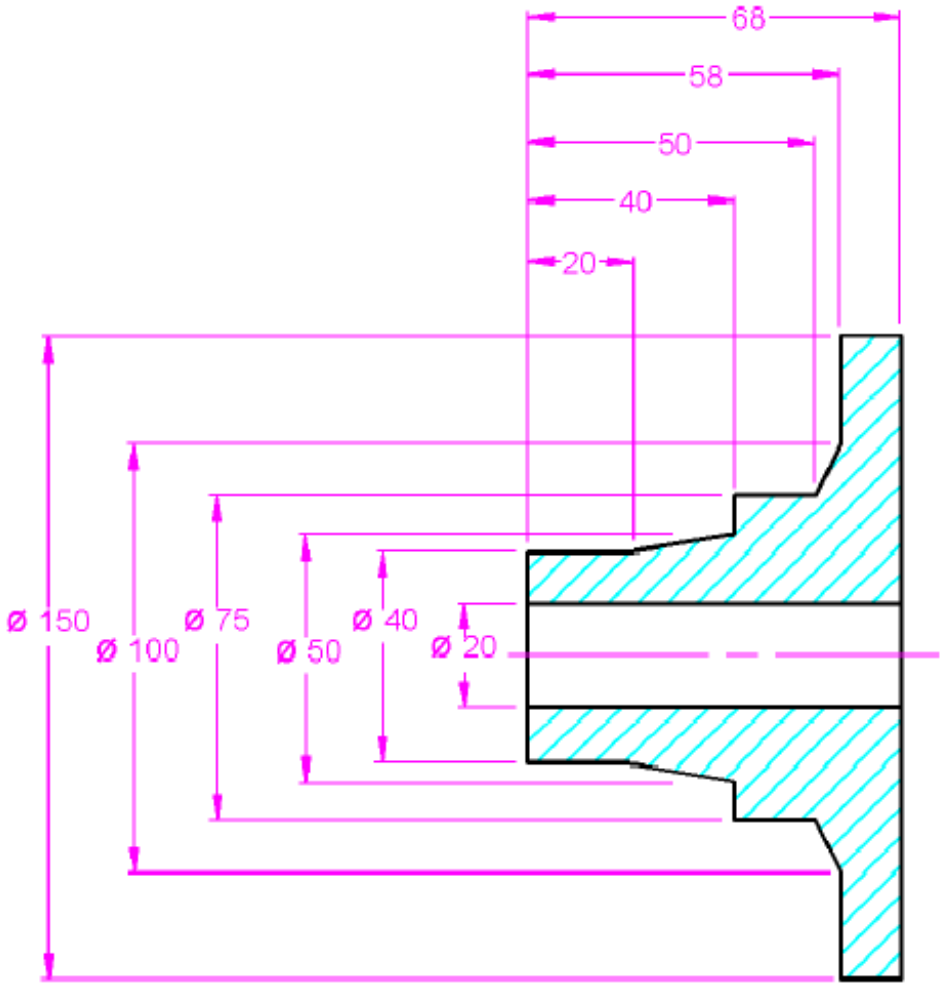
Dimensioning Arcs

- Arcs are dimensioned in a view where a true shape is seen.
- The leader and the value are located inside the arc if space is available. If not, the value is moved out with or without the leader.
- A cross indicates the centre of all arcs with or without dimensions, except for small and unimportant radii.
- False center with jogged leader can be used for long radii.

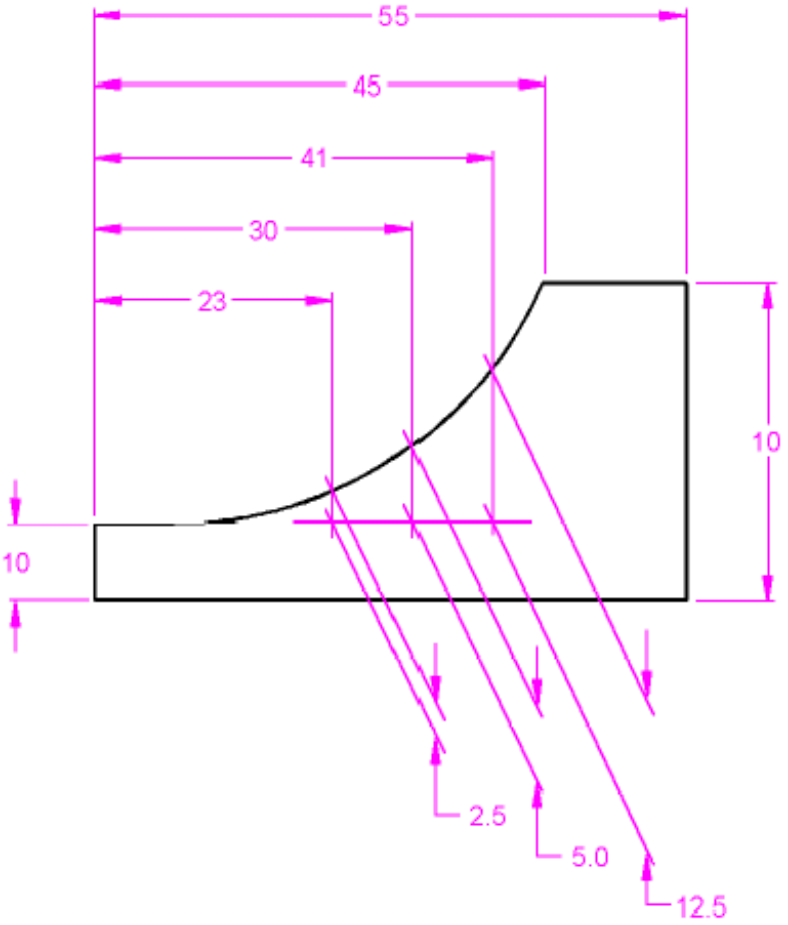




Dimensioning of Chained Features



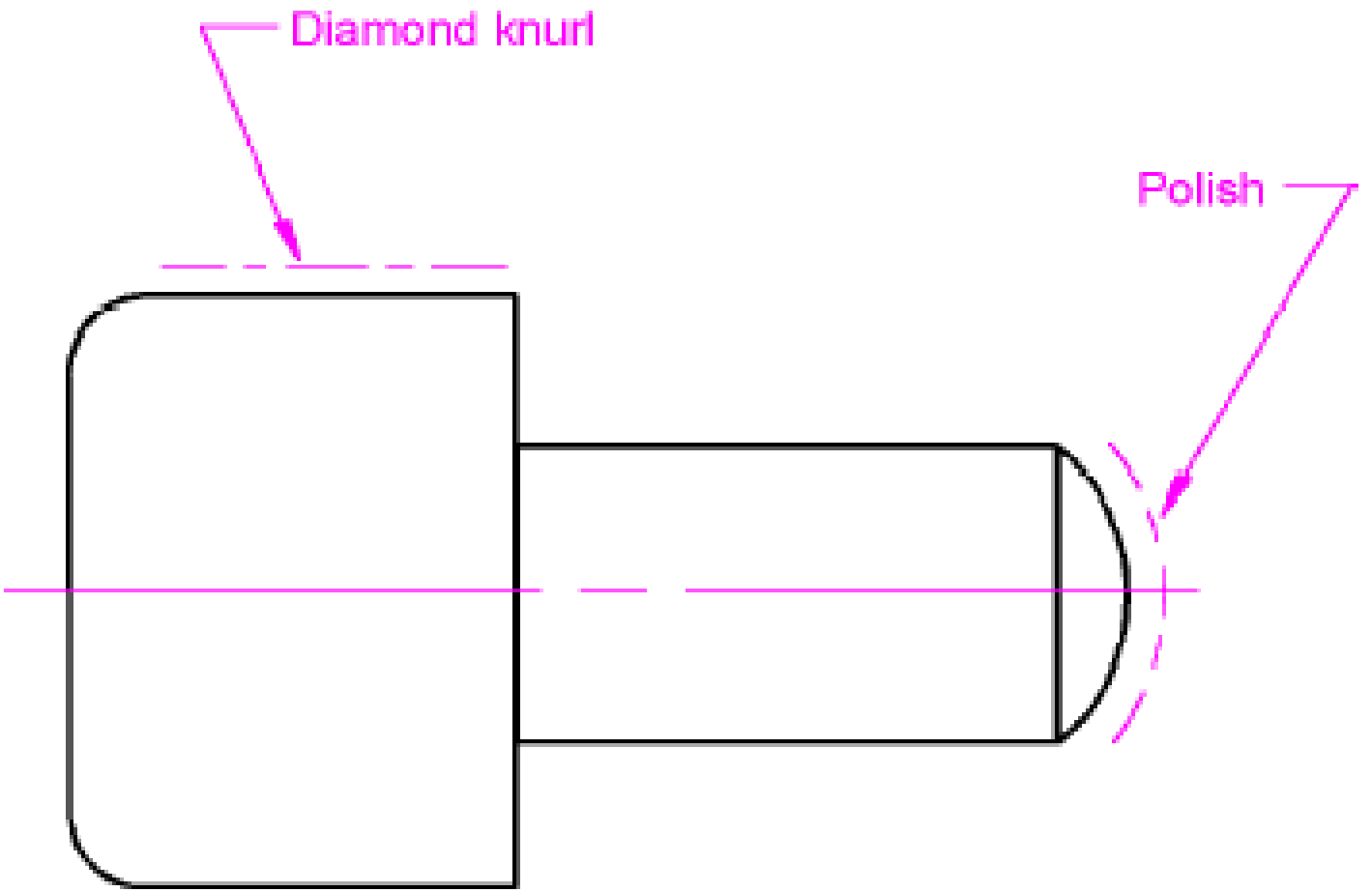
Staggering dimension text



Aligning dimension lines

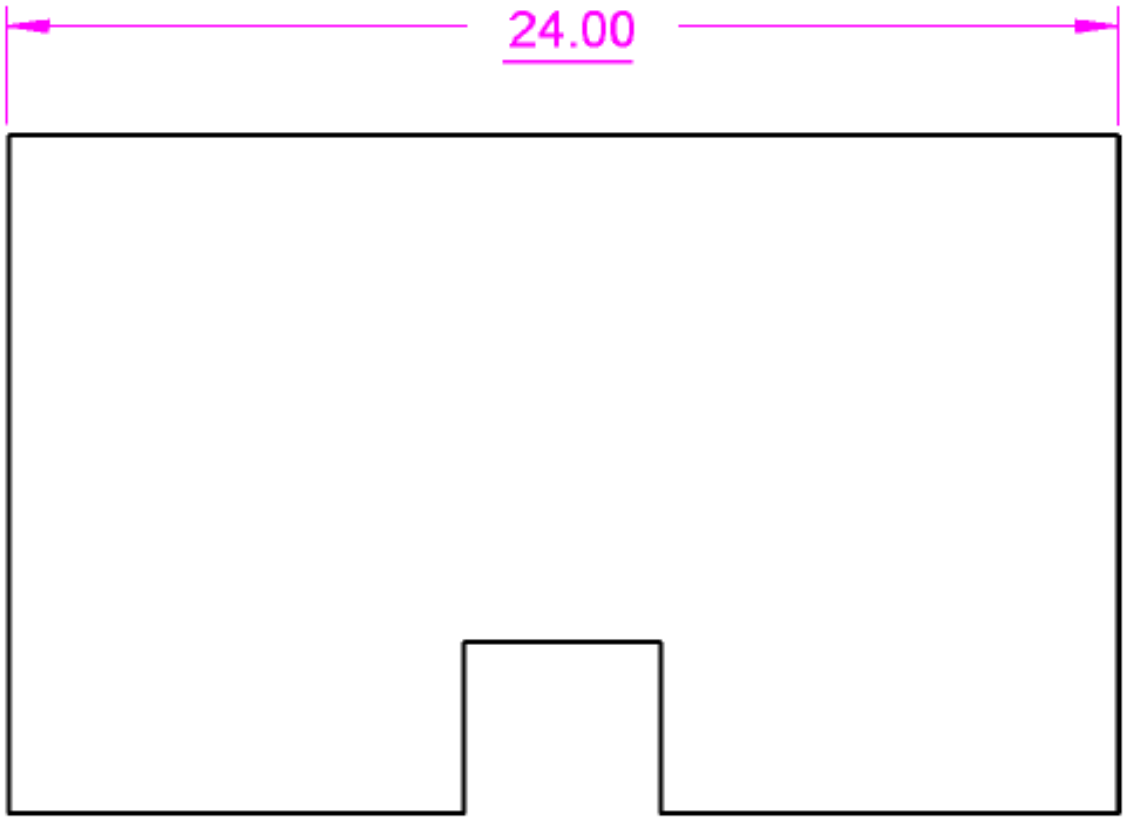
Detailed Explanations

Extension lines and line indicators are used to detail manufacturing requirements.



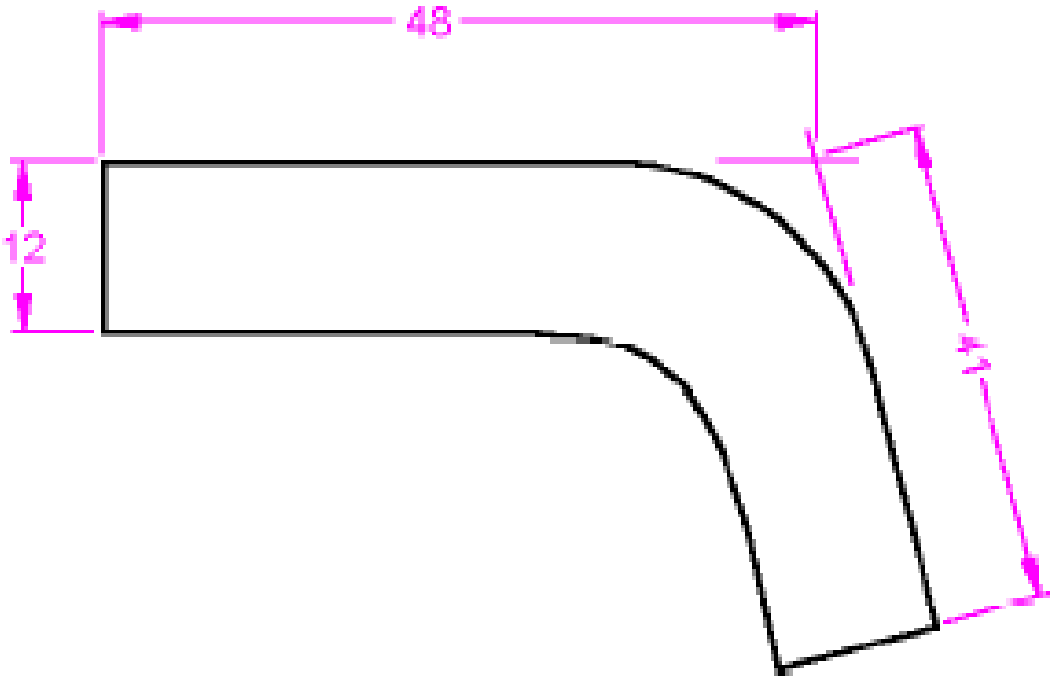
Not to Scale Designation

- All features in drawings are scaled accordingly
- Unscaled features could be also represented and indicated.



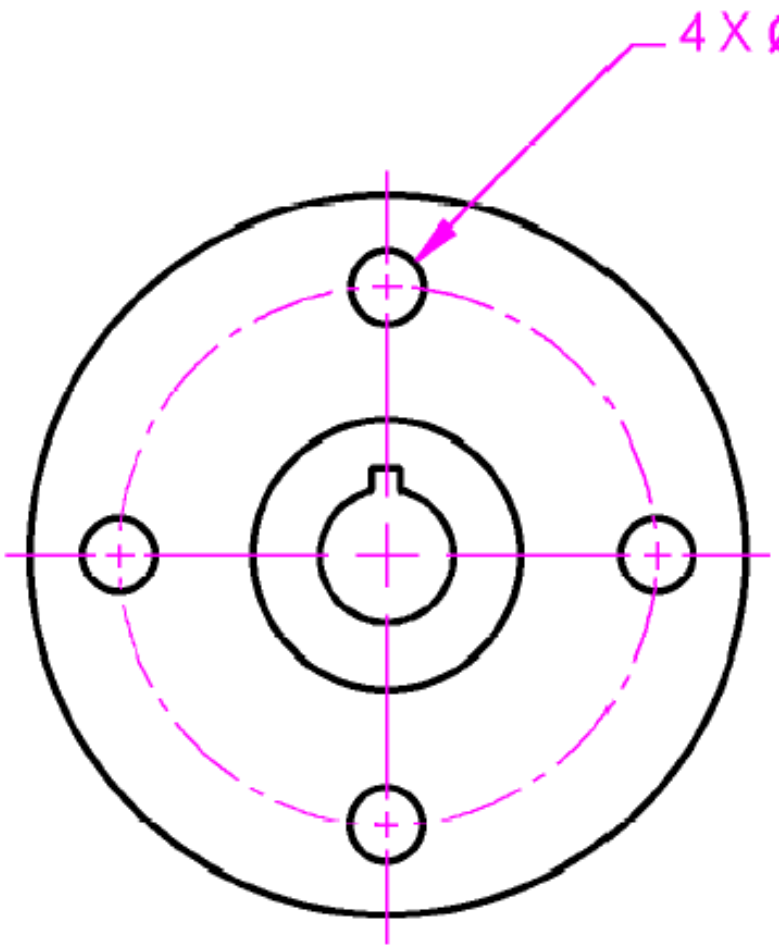
Reference for the Extension Line

- Dimensioning is always done between crisp surfaces.
- Sometimes, such surfaces are not available and dimensions are indicated to facilitate the manufacturing process.



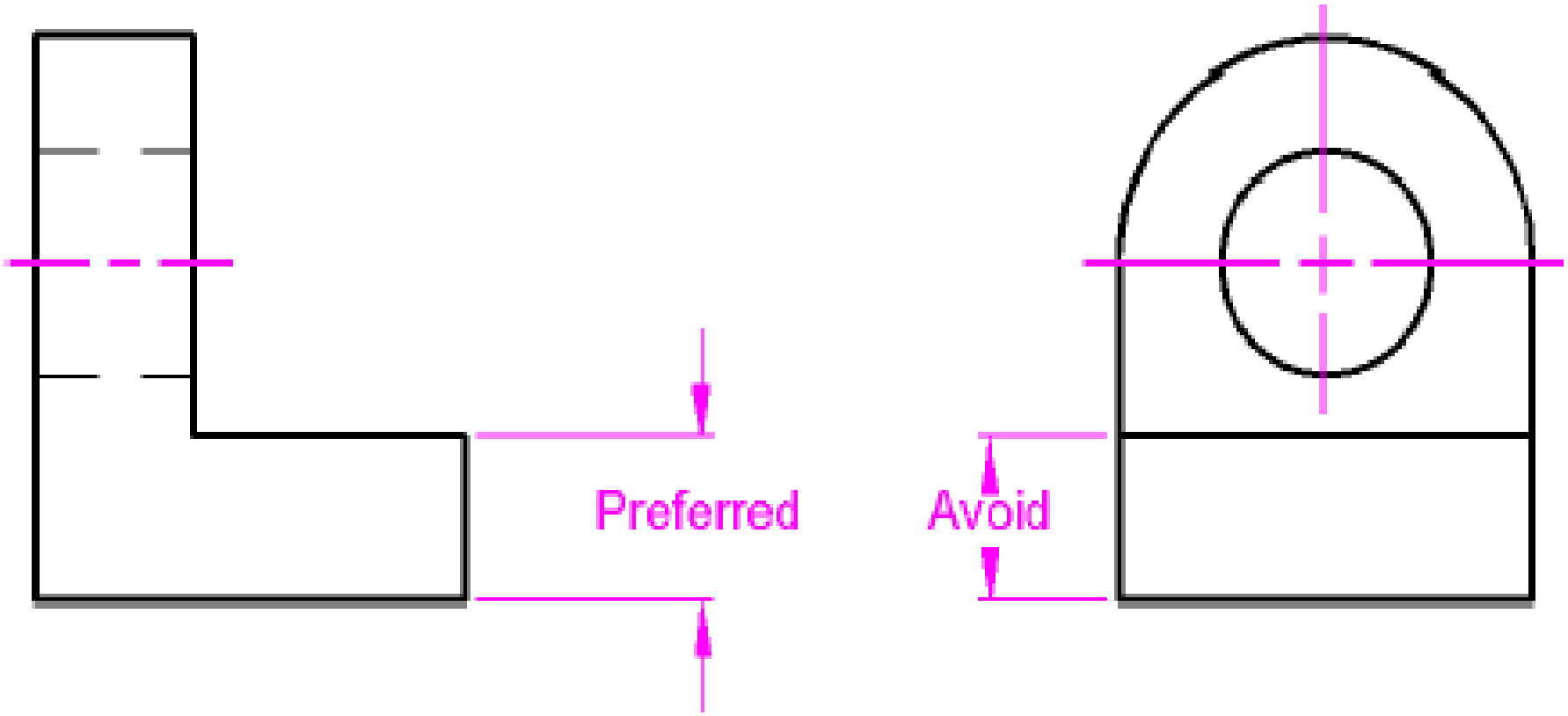
General Dimensioning

Holes should be dimensioned in the view in which they are best seen.



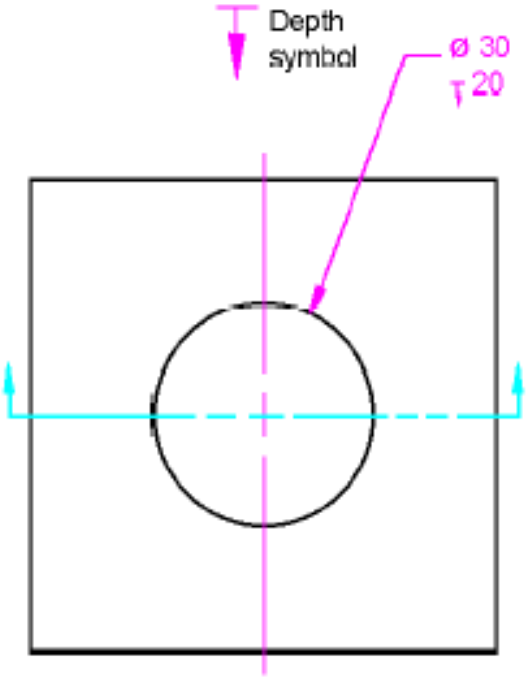
General Dimensioning cont'd

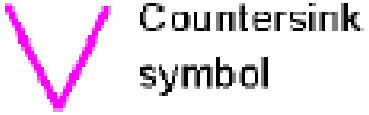
Features should be dimensioned in the views in which they are best seen.

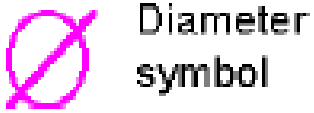


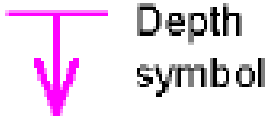
General Dimensioning cont'd

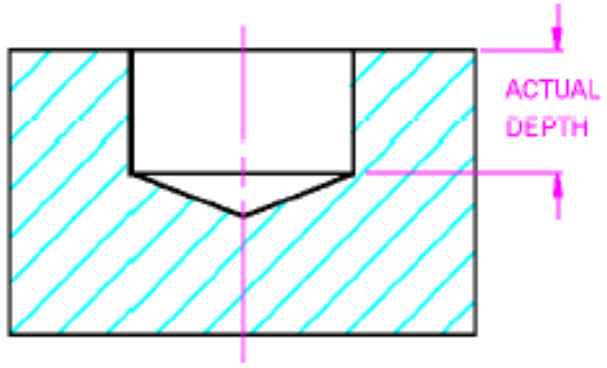
Do not draw a view/section for a feature that could be indicated by a symbol.



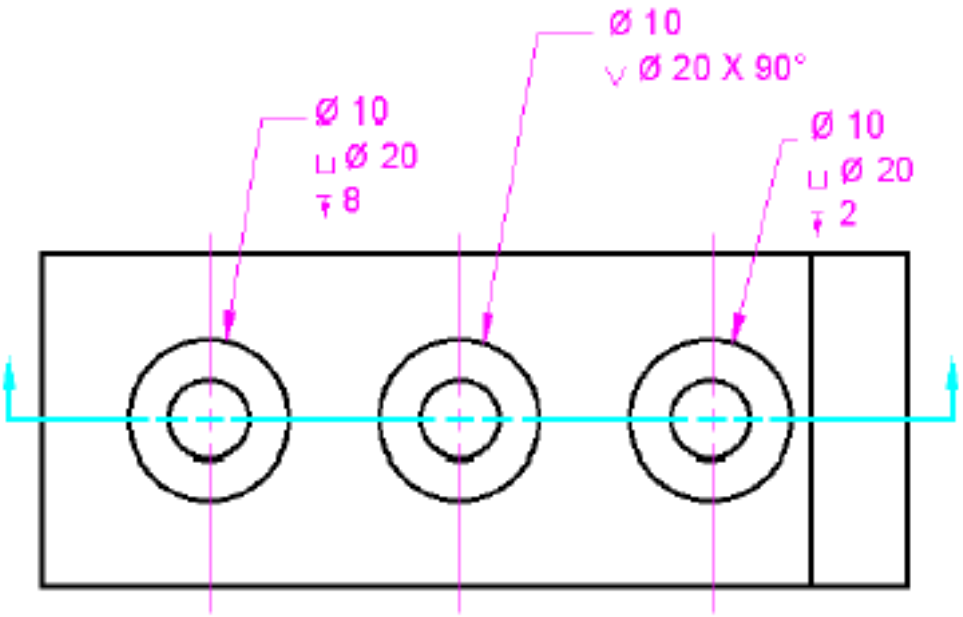




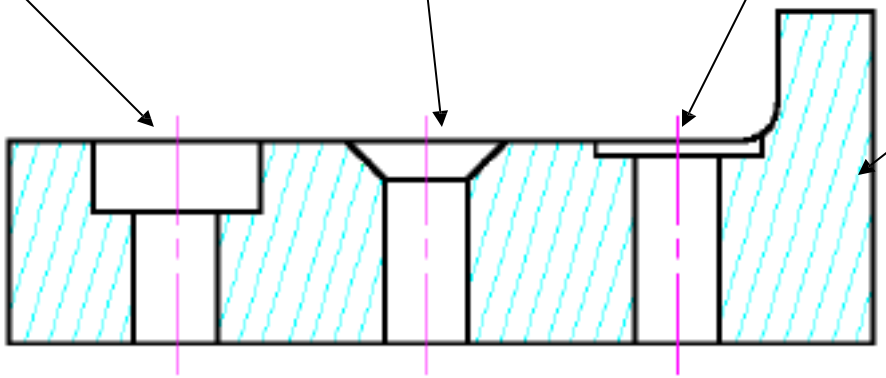




General Dimensioning – Cont'd



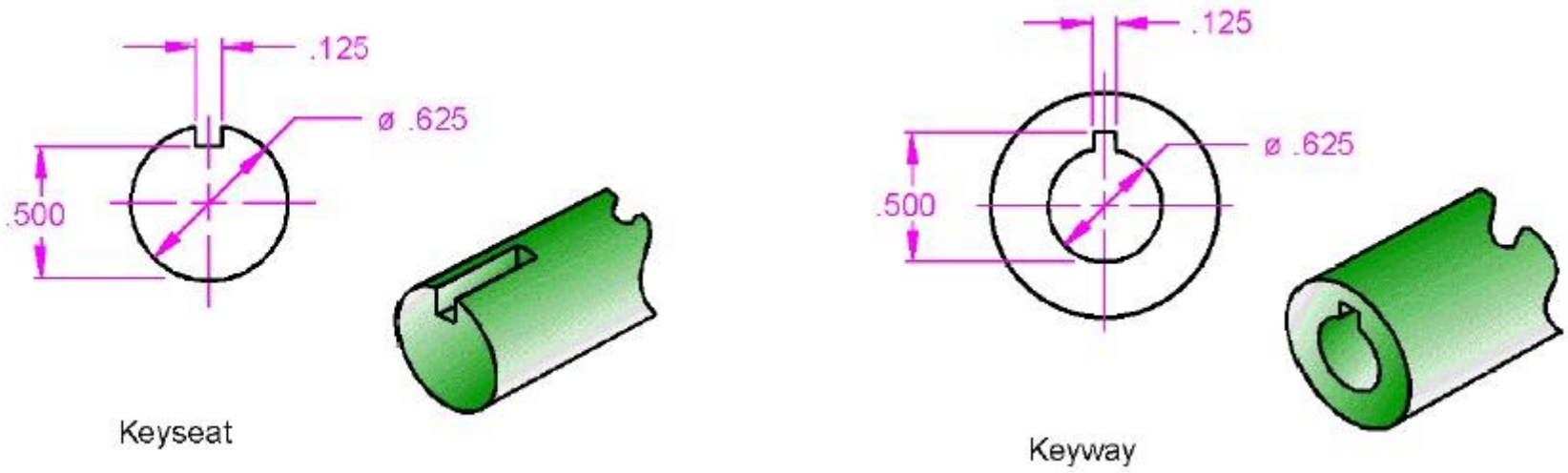
Counterbore Countersunk Spotface



A section view is not necessary as symbols in the top view indicate these shapes.

General Dimensioning – cont'd

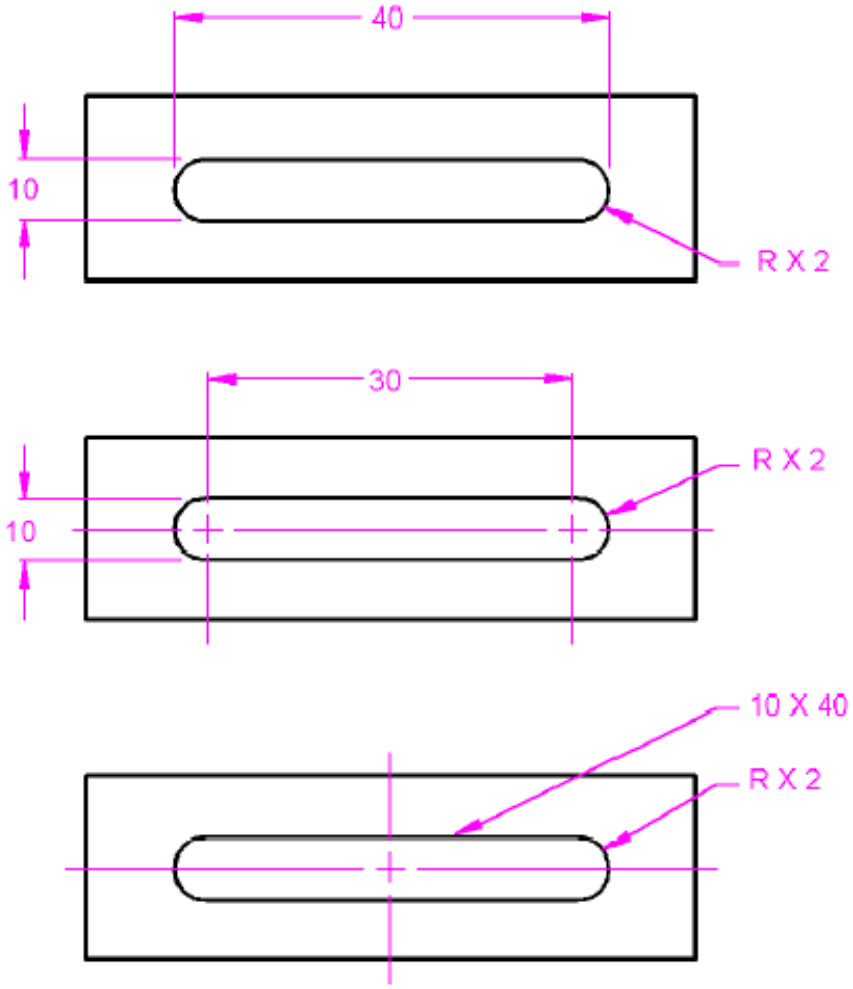
- Dimension keyseats from the bottom of the keyseat to opposite end of the shaft.
- For key seat, from top of keyway to bottom of hole.



Keyseat and **keyway**

General Dimensioning – cont'd

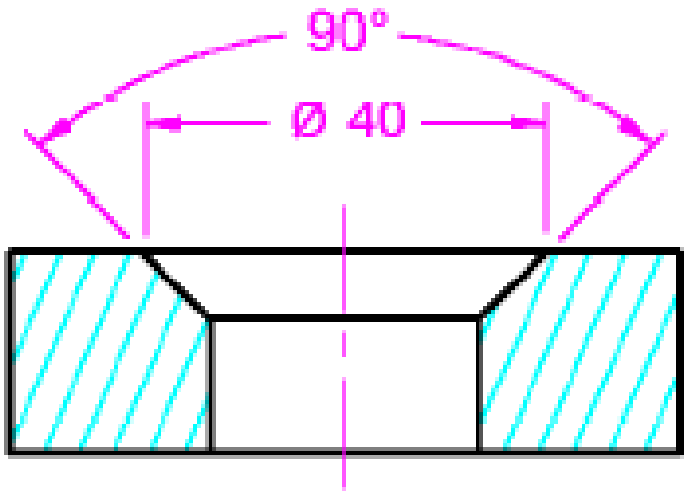
- Dimension by giving centre to centre distances and radii of ends.
- Only one radius dimension is required, but the amount of instances must be mentioned.

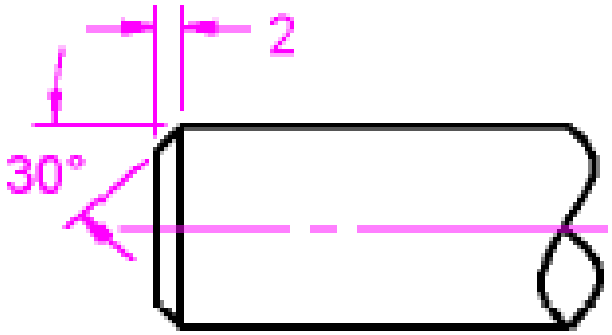


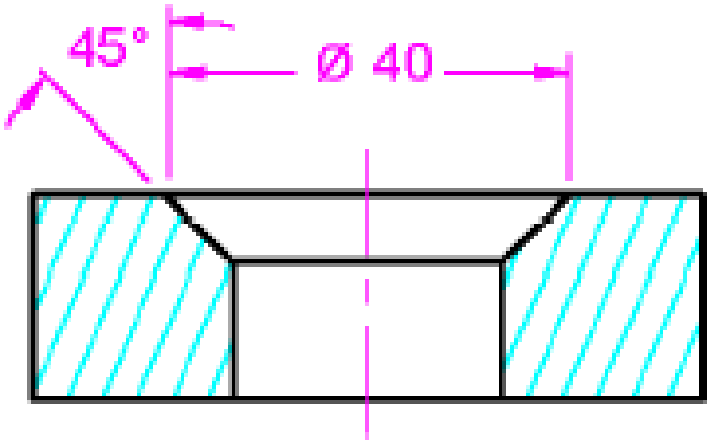
Slot cuts

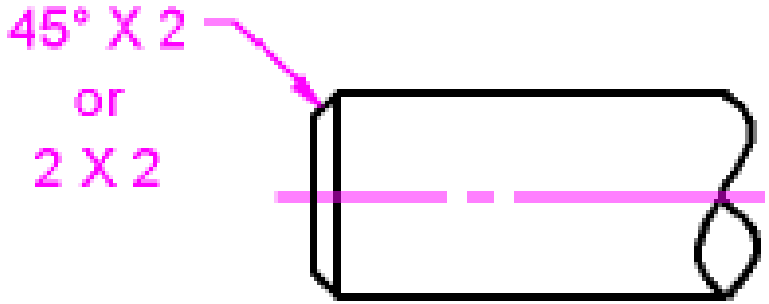
General Dimensioning cont'd

Chamfers





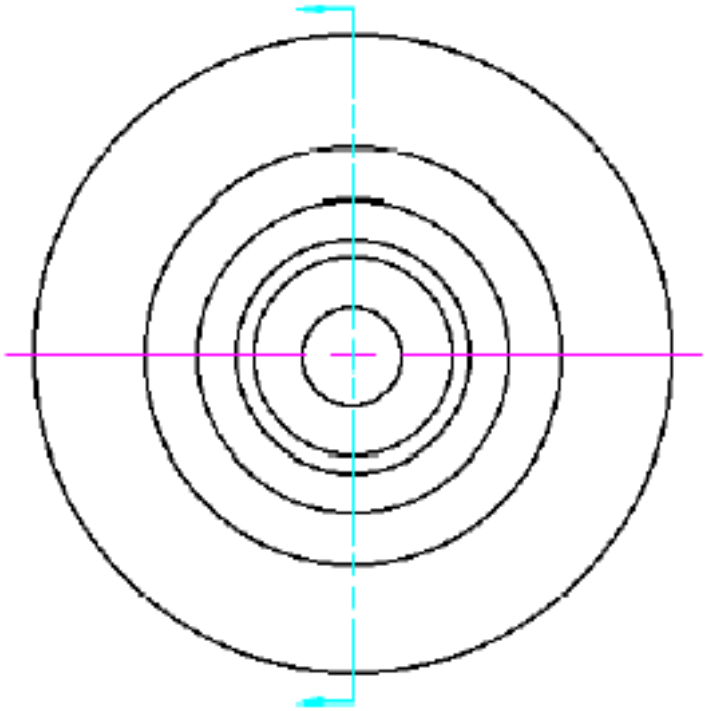


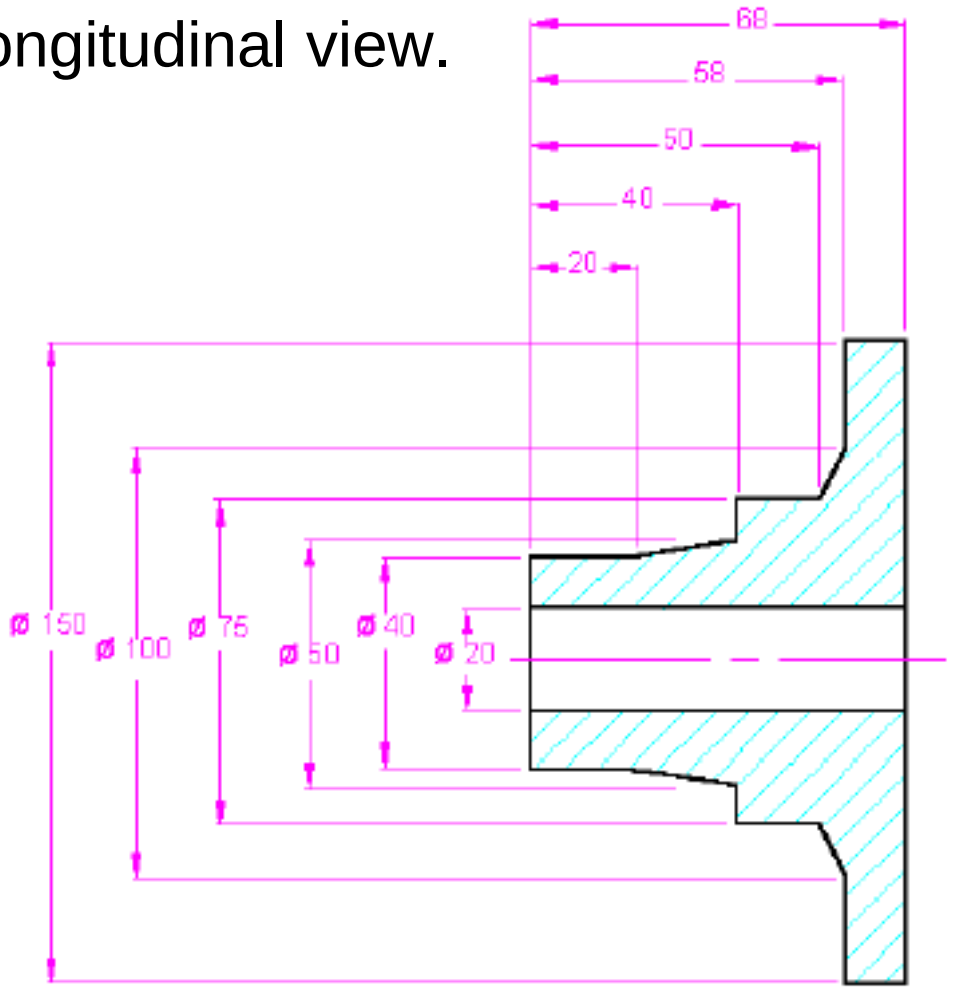


General Dimensioning cont'd

Concentric circles

- Dimensioned in the longitudinal view.

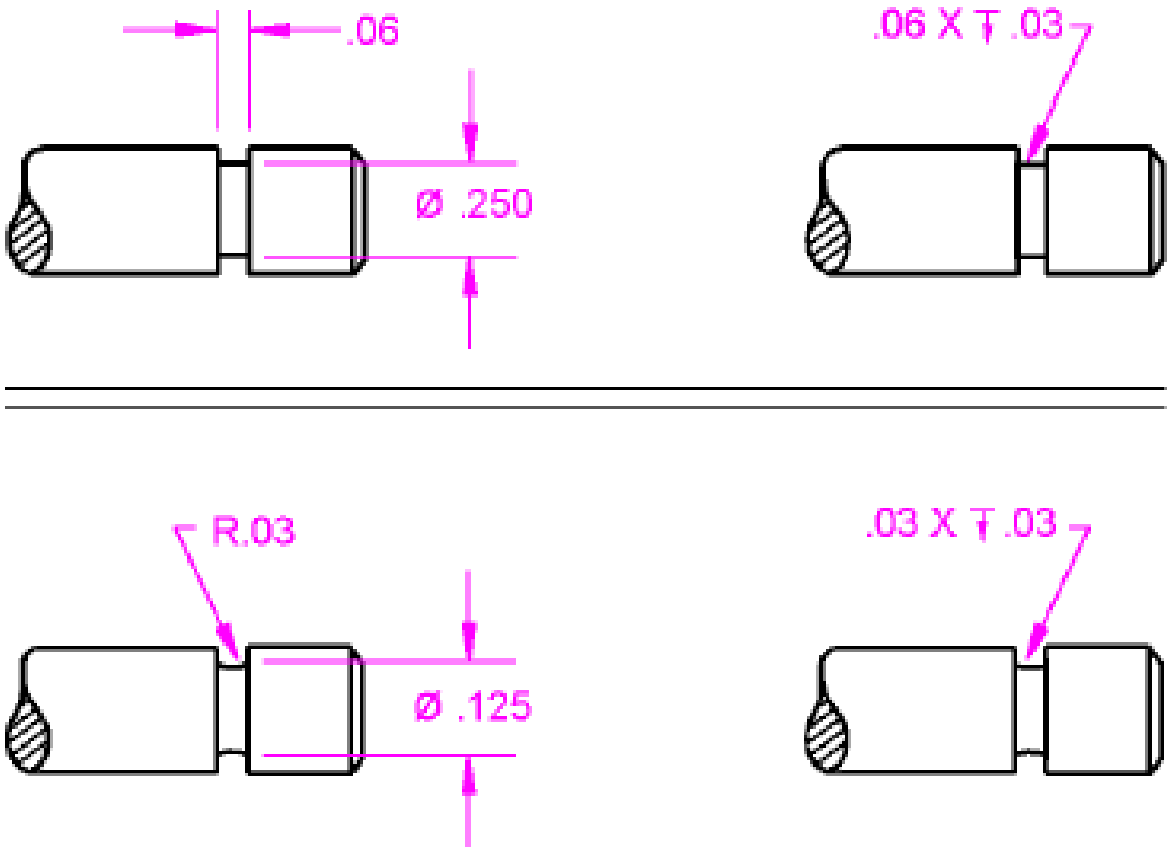




General Dimensioning cont'd

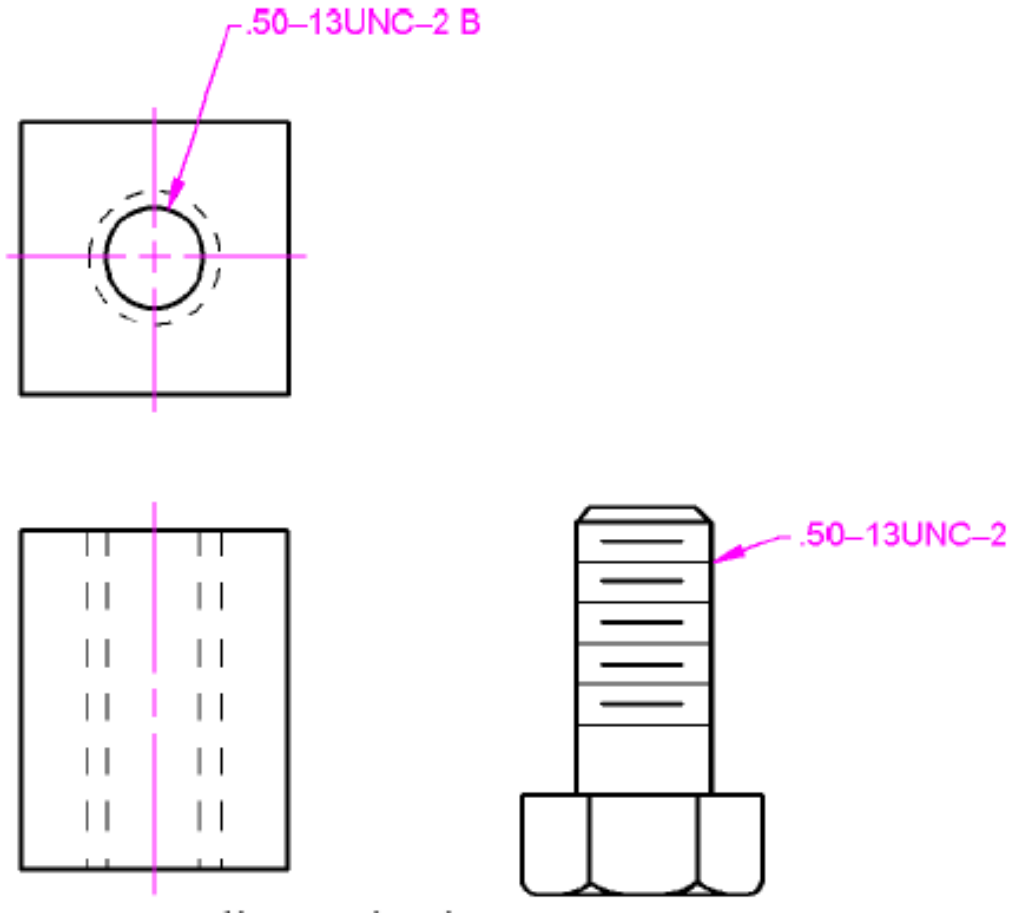
Grooves

Dimensioned with local notes or by showing the dimensions of the **undercut depth** and the **distance**.

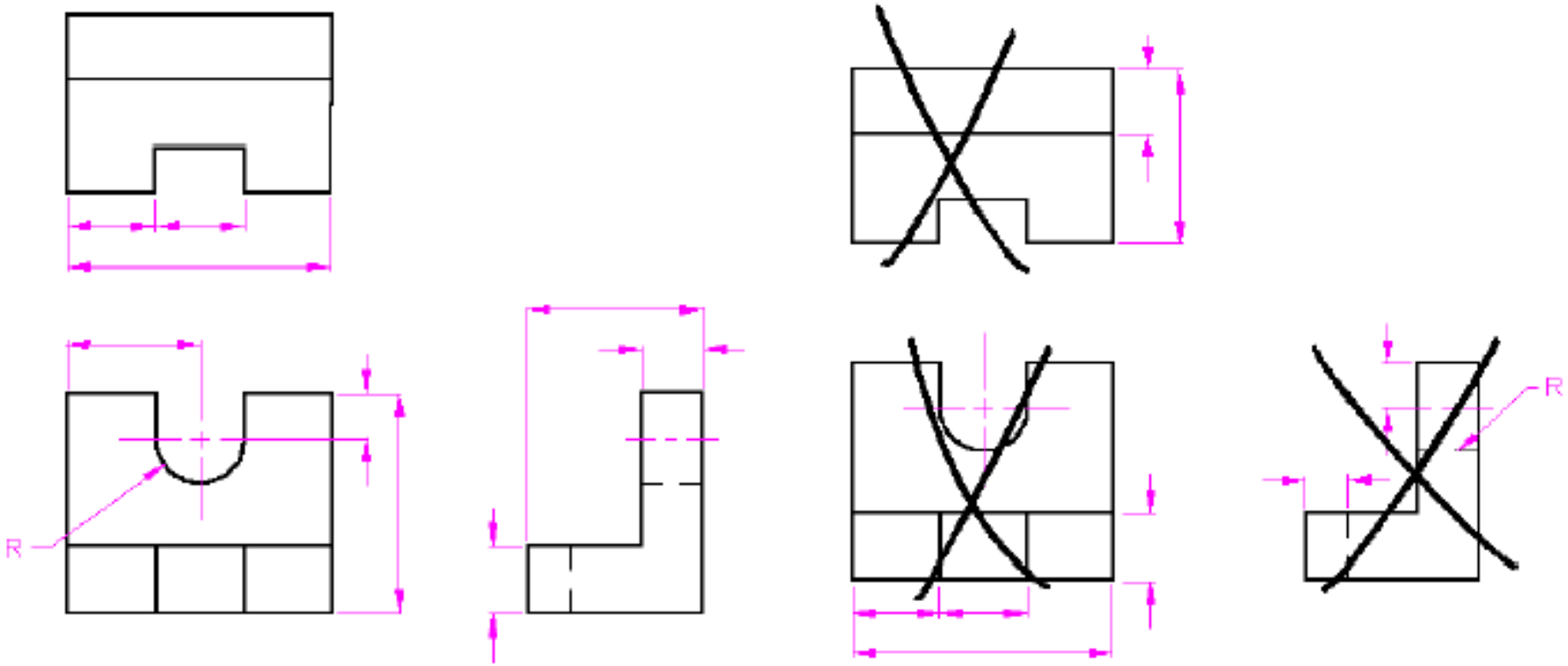


General Dimensioning - cont'd

- Threads are dimensioned with local notes.
- Internal or tapped threads are shown on the circular view.
- External threads are shown on the longitudinal view.

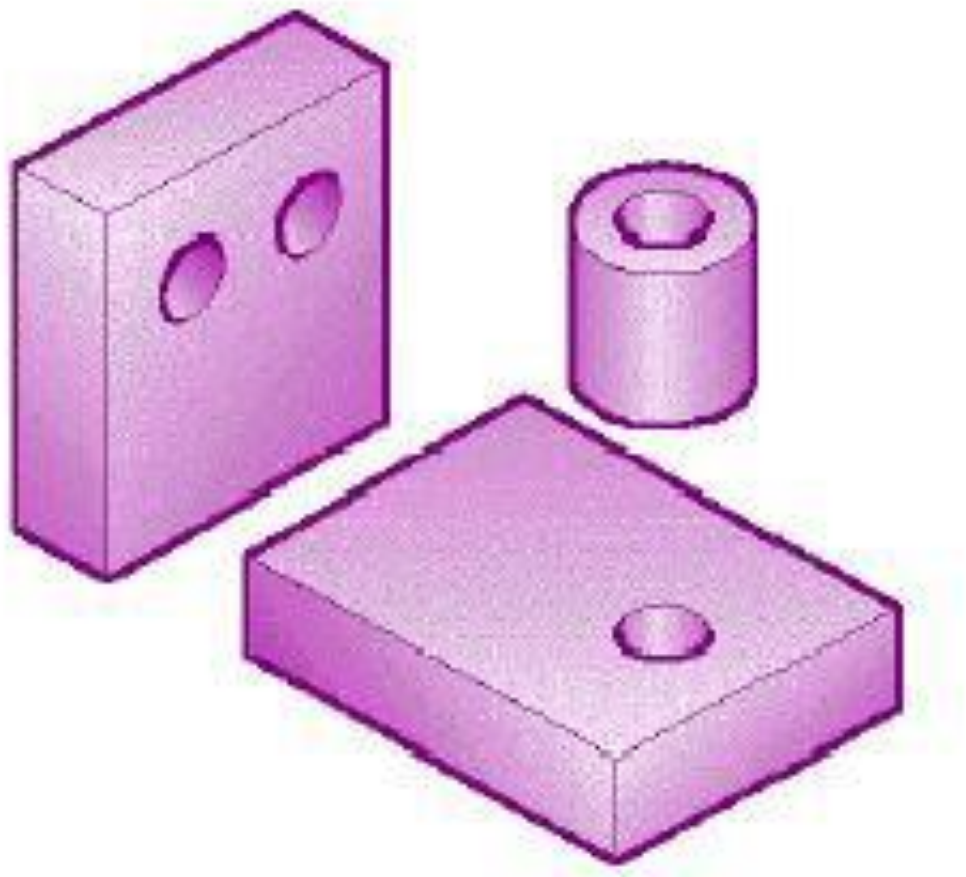


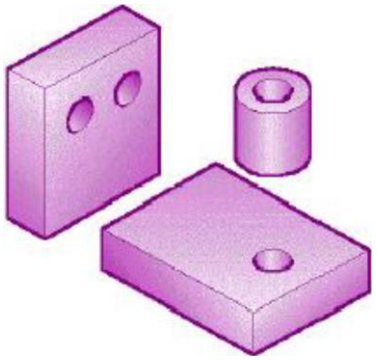
Contour Dimensioning



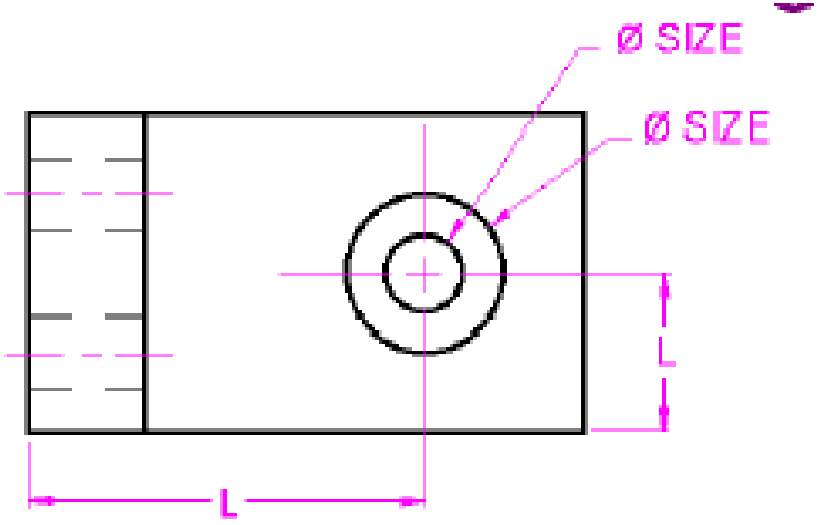
Size vs. Location

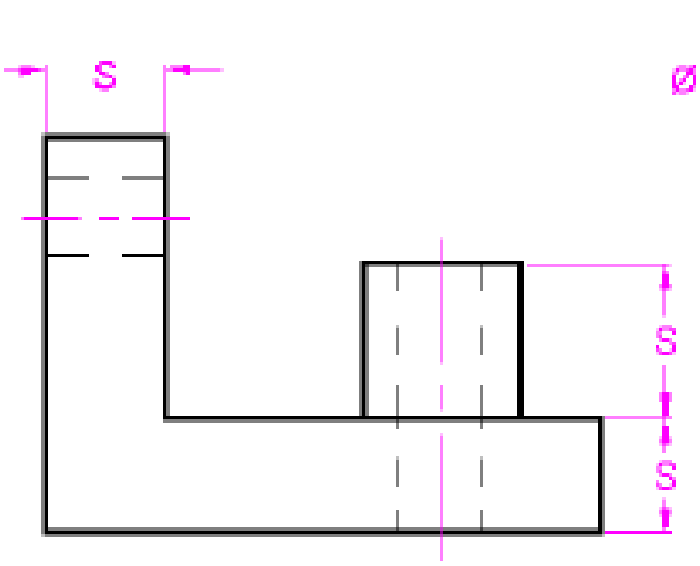
Both size and location dimensions have to be provided to avoid any confusion

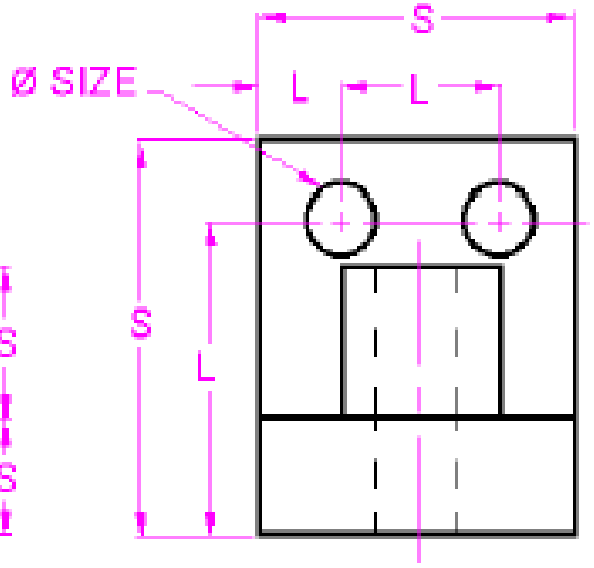




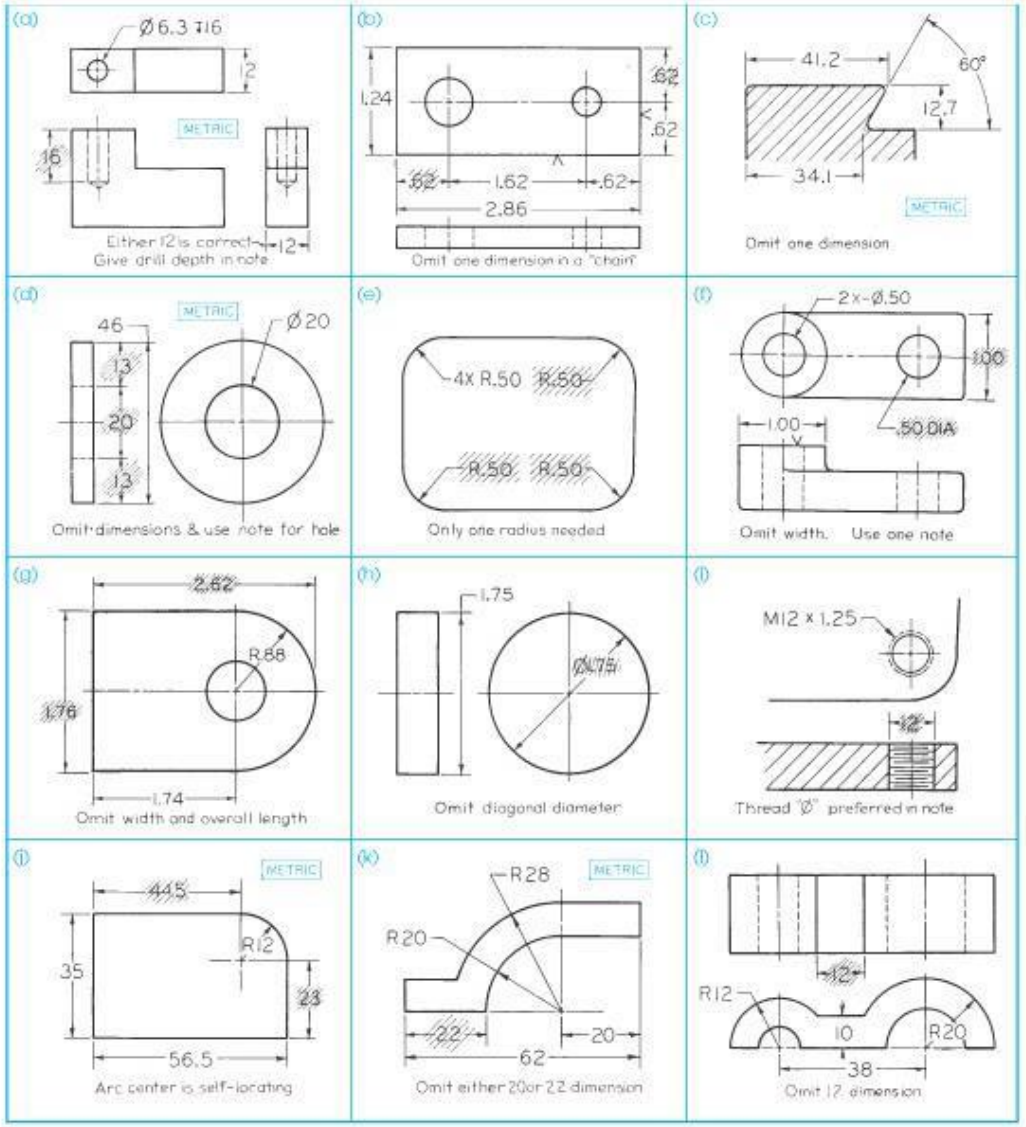
Size vs. Location



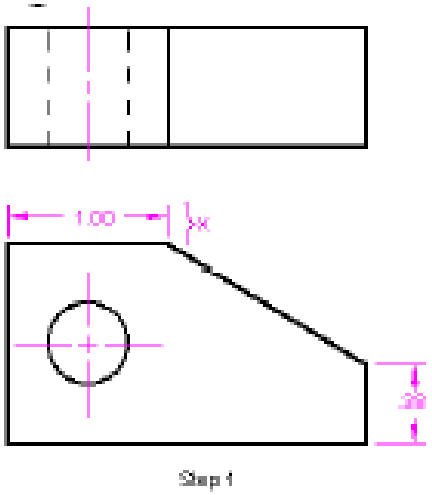




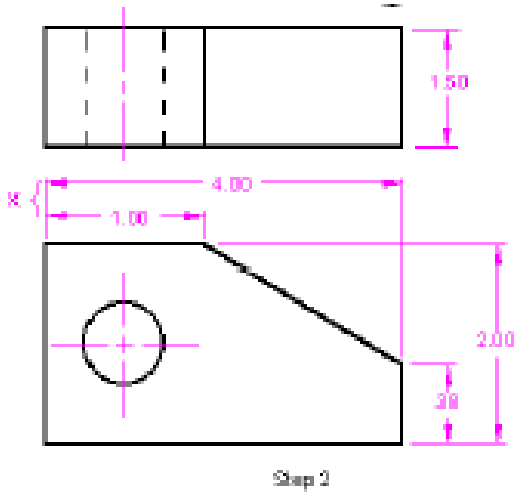
Superfluous Dimensions



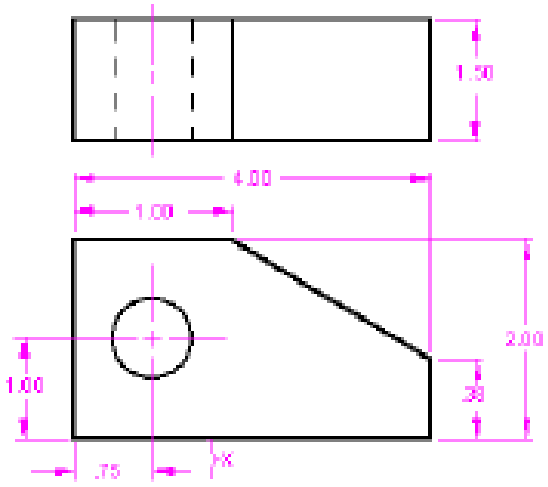
Creating a Dimensioned Drawing



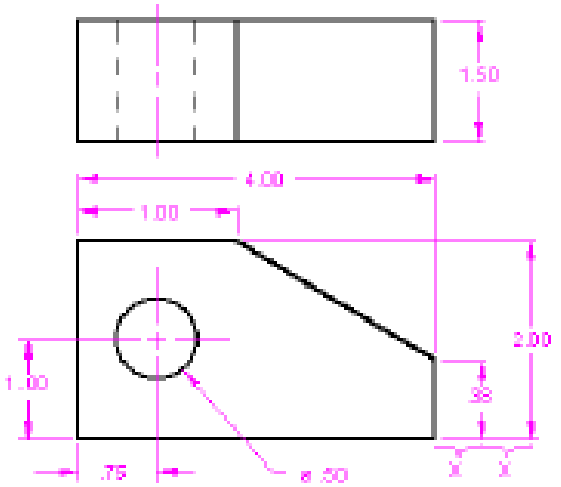
Step 1



Step 2



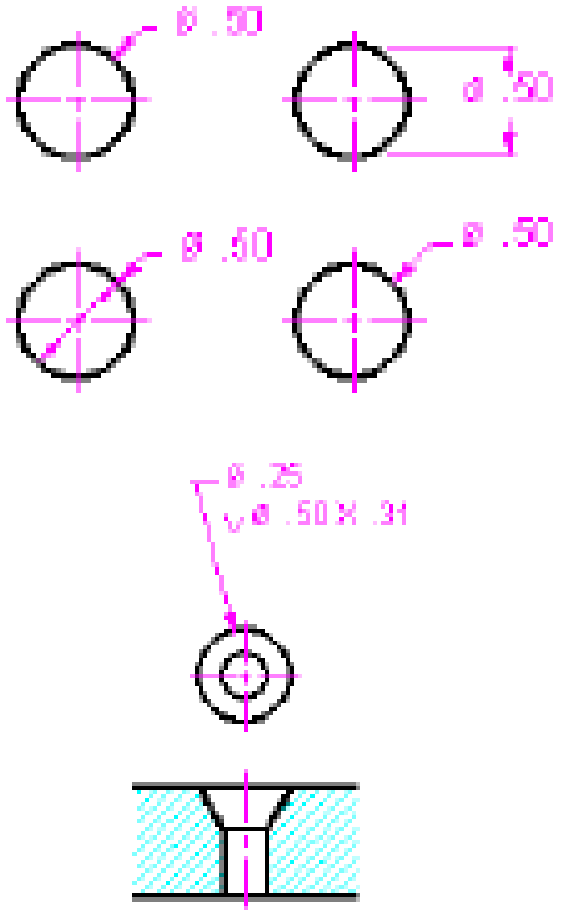
Step 3



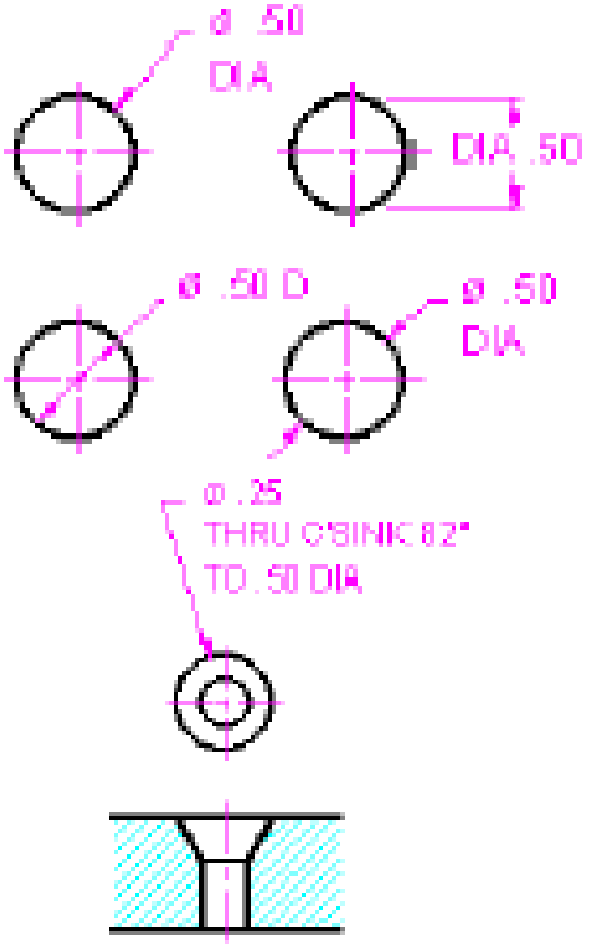
Step 4

Representation Standards – ANSI Y 14.5 M

New standard



Old standard



REMINDER

- Each feature of an object is dimensioned once and only once.
- The location and/or size dimensions for a feature should be placed in the view in which that feature is most clearly seen i.e. where its shape description is most complete.
- Any dimension specified should correspond to a range of dimensions in the final product, i.e. each dimension should include an appropriate tolerance

REMINDER

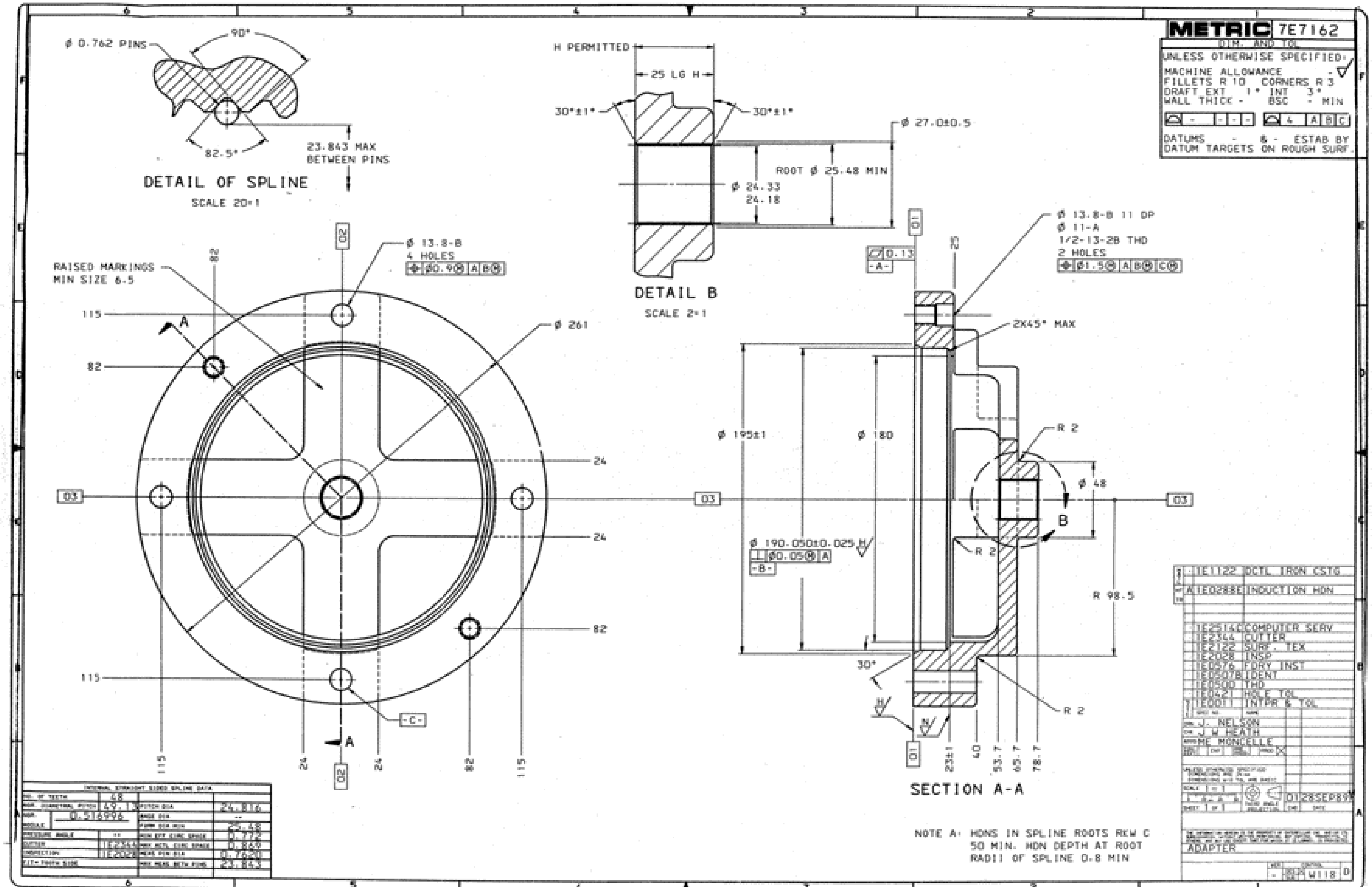
- Dimensions lines should never coincide with object lines or other extension lines,
- Dimension lines should be unbroken except for the number between the arrowheads,
- There should be a visible gap between the object and the origin of an extension line,
- Crossing of dimension lines should be avoided wherever possible.

REMINDER

- Dimensions should reference object lines rather than hidden lines,
- Dimensions should be placed in spaces as close as possible to their point of application,
- When dimensions are "nested", the smaller dimension should be placed closer to the object to avoid unnecessary crossing,
- Dimensions should be located outside the boundaries of the object wherever possible.

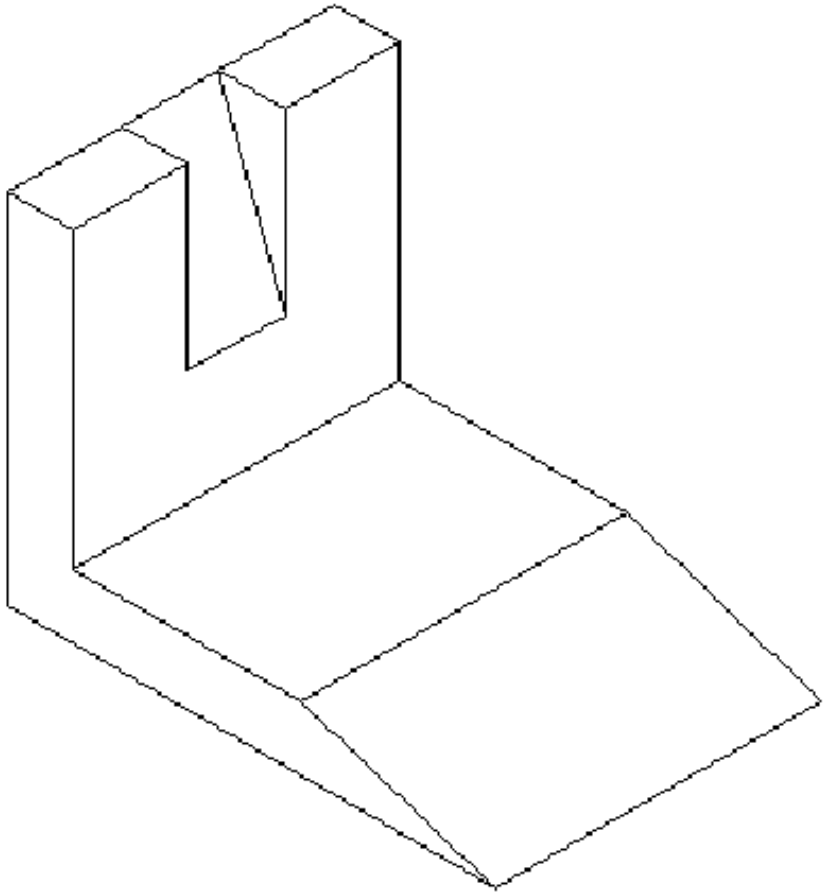
Working Drawings

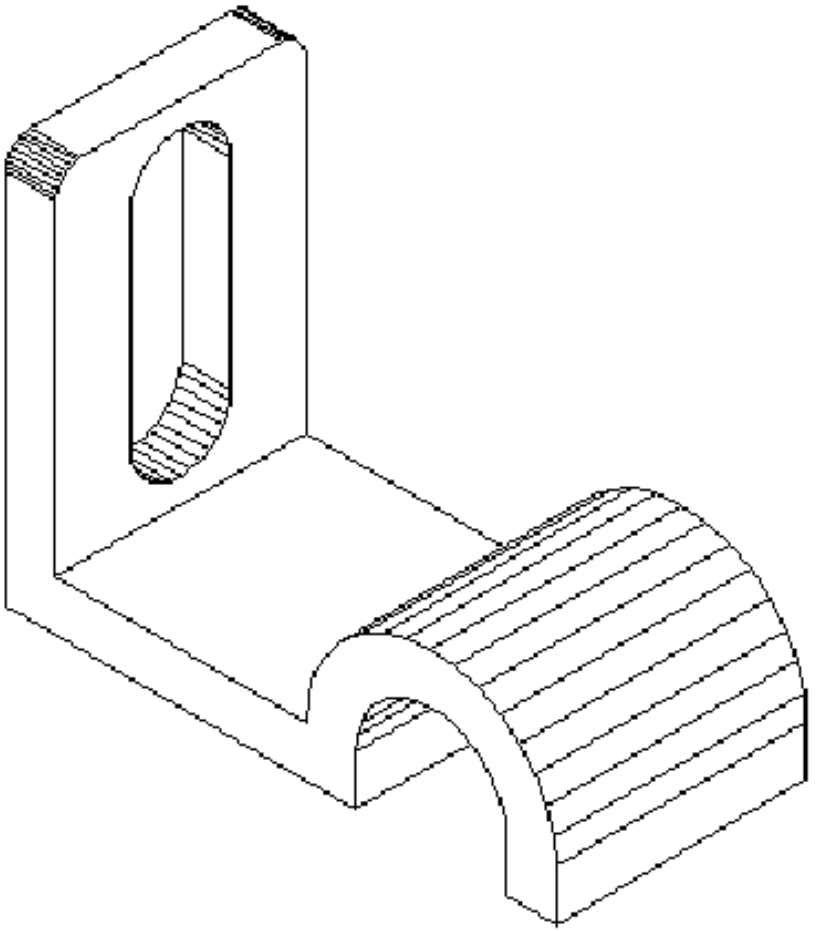
- Used to describe all the necessary details that enable the fabrication of the component,
- Use the necessary views and sections,
- Provide all the necessary dimensions only in one single location.

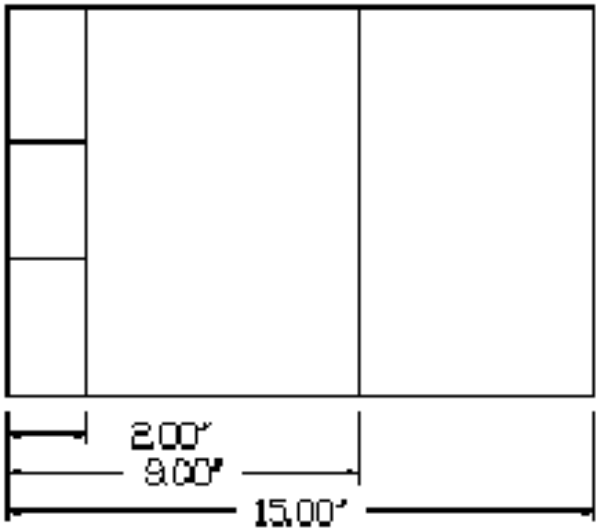

METRIC 7E7162
DIM. AND TOL
UNLESS OTHERWISE SPECIFIED:
MACHINE ALLOWANCE
FILLETS R 10 CORNERS R 3
DRAFT EXT 1° INT 3°
WALL THICK - BSC - MIN
DATUMS - & - ESTAB BY
DATUM TARGETS ON ROUGH SURF.
ø 0.762 PINS
90°
82.5°
23.843 MAX
BETWEEN PINS
DETAIL OF SPLINE
SCALE 20:1
H PERMITTED
25 LG H
30°±1°
30°±1°
ø 27.0±0.5
ROOT ø 25.48 MIN
ø 24.33
24.18
DETAIL B
SCALE 2:1
RAISED MARKINGS
MIN SIZE 6.5
ø 13.8-B
4 HOLES
ø 261
ø 13.8-B 11 DP
ø 11-A
1/2-13-2B THD
2 HOLES
2X45° MAX
R 2
ø 195±1
ø 180
ø 48
ø 190.050±0.025
R 98.5
30°
SECTION A-A
23±1
40
53.7
65.7
78.7
NOTE A: HONS IN SPLINE ROOTS RKW C
50 MIN. HON DEPTH AT ROOT
RADII OF SPLINE 0.8 MIN
INTERNAL STRAIGHT SIDED SPLINE DATA
NO. OF TEETH 48
PITCH DIA 24.816
0.516996
FORM DIA MIN 25.48
0.772
0.869
1E2344
1E2028
MEAS PIN DIA 0.7620
MAX MEAS BETW PINS 23.843
1E1122 DCTL IRON CSTG
A 1E0288E INDUCTION HDN
1E2514C COMPUTER SERV
1E2344 CUTTER
1E2122 SURF. TEX
1E2028 INSP
1E0576 FDRY INST
1E0507B IDENT
1E0500 THD
1E0421 HOLE TOL
1E0011 INTPR & TOL
J. NELSON
J W HEATH
ME MONCELLE
01 28SEP89
ADAPTER
W118
D

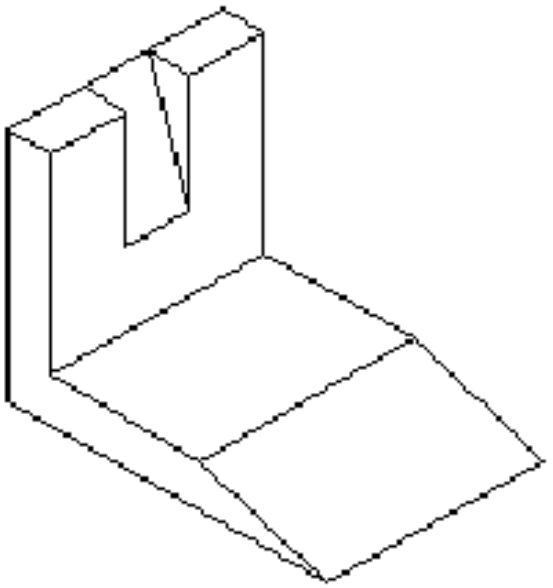
Problems

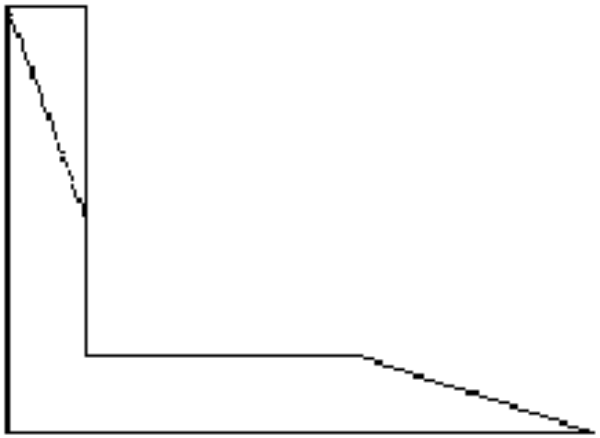
Draw all the necessary views/sections that would enable the correct dimensioning of the following parts:

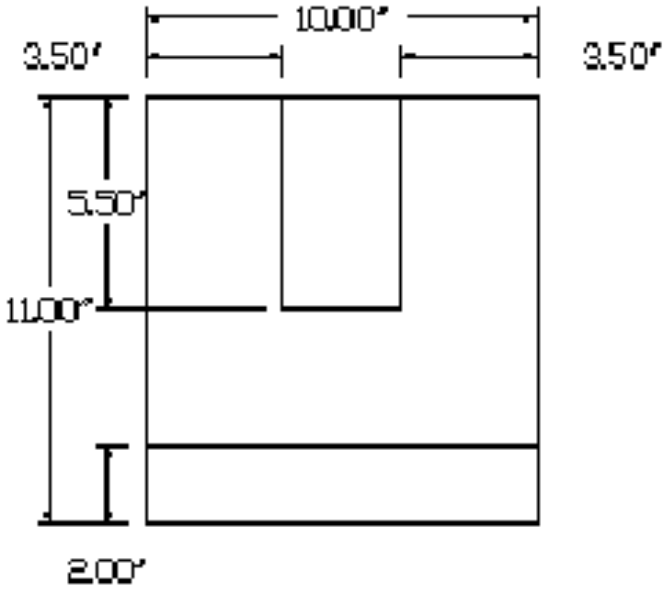


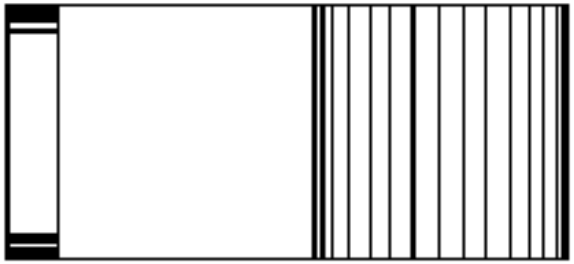


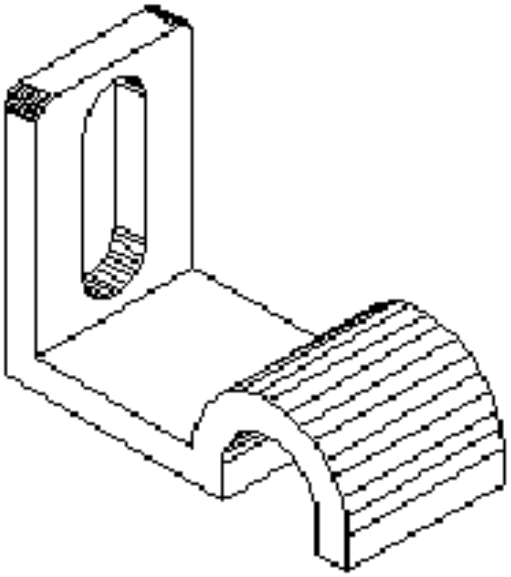

2.00"
9.00"
15.00"

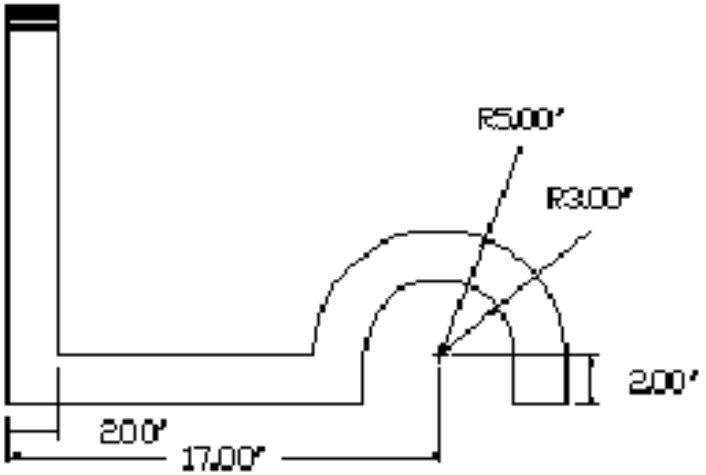


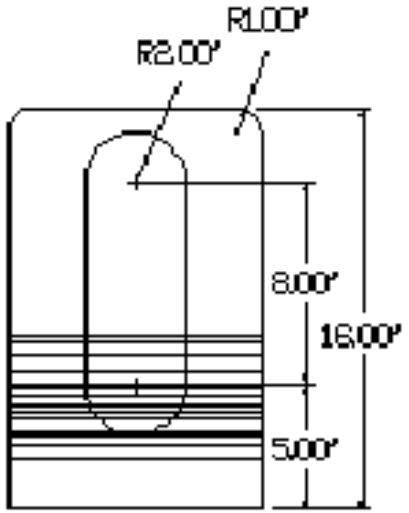



10.00"
3.50"
3.50"
5.50"
11.00"
2.00"






R5.00"
R3.00"
2.00"
2.00"
17.00"


R1.00"
R2.00"
8.00"
16.00"
5.00"

Dimensional, Geometric and Position Tolerances

TOLERANCING

Tolerancing Basics

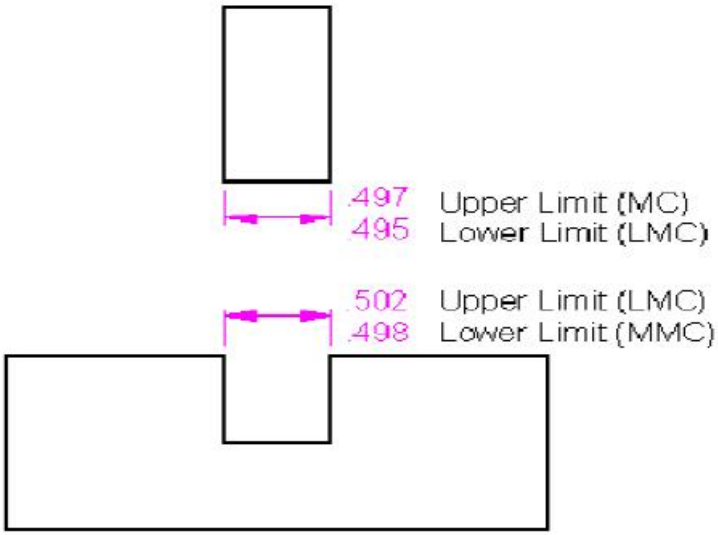
Tolerance: undesired but accepted abatement form a basic size or shape.

There are three types of tolerances:

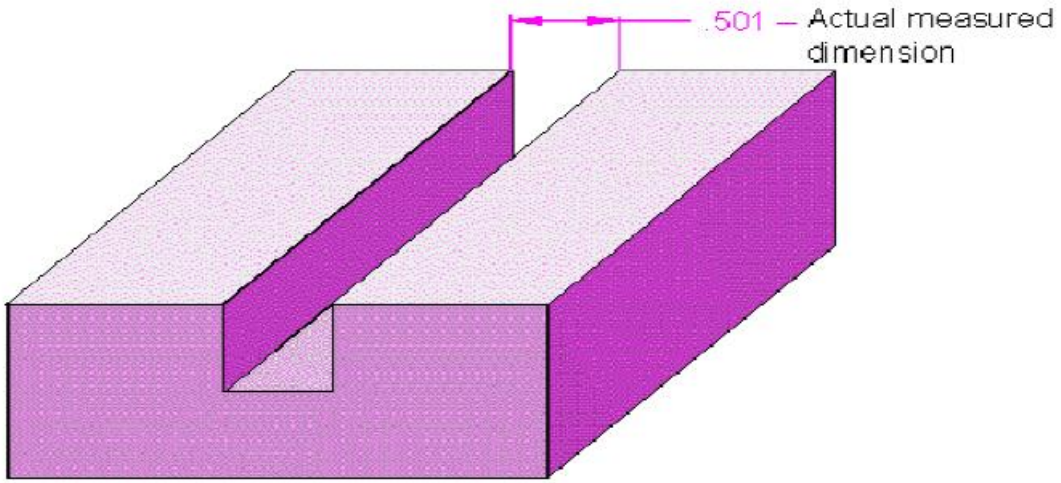
- Dimensional tolerances (limits of the linear or angular dimensions)
- Positional tolerances (limits of linear or angular location of features within a part)
- Geometric tolerances (abatement form shape or position of a specific feature)

Fundamentals

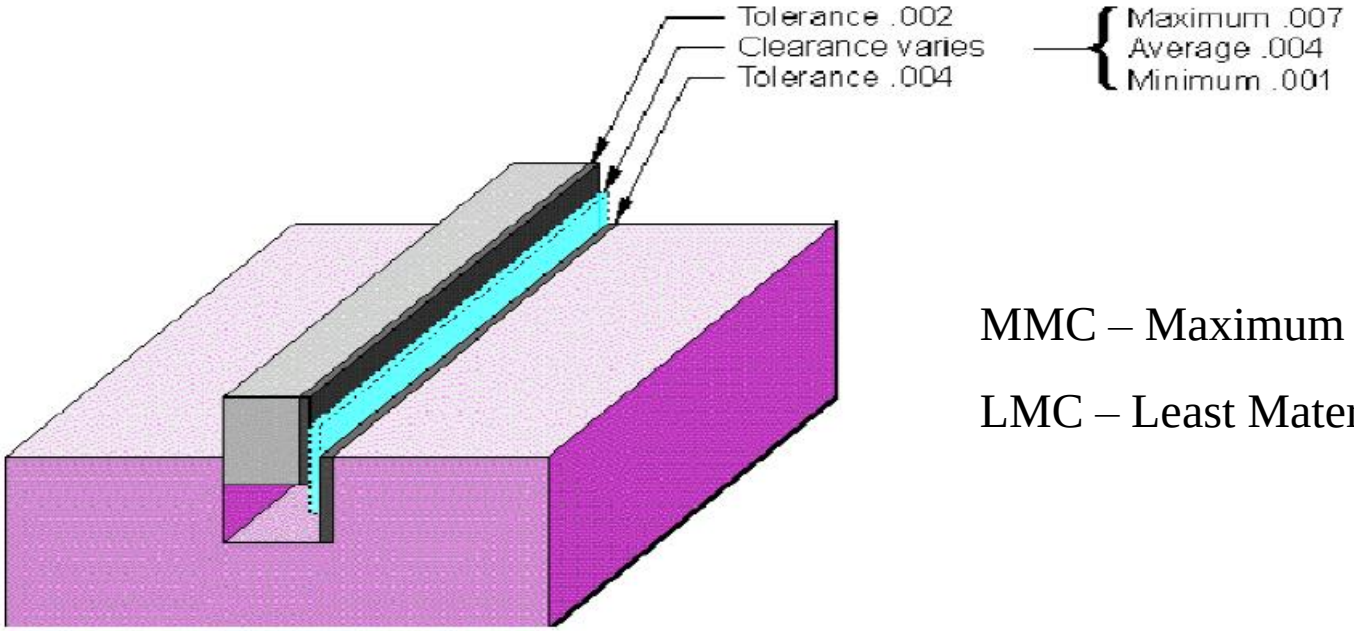
- Undesired but permitted dimensional variations of a certain feature due to economics aspects in manufacturing.
- Tolerances are essential when two or more parts are assembled together (clearance).
- Permitted variations are related to the functions of the parts.



Engineering Dimensioned Drawing



Machined Part



MMC – Maximum Material Condition

LMC – Least Material Condition

What is Important?

- Understanding tolerances,
- Selection and calculation,
- Prescription of tolerances,
- **Tolerance of a size**: the difference between the maximum and the minimum allowed size of the specific dimension.

Nomenclature

- **Nominal size**: the general size
- **Basic size**: theoretical size
- **Actual size**: the measured size
- **Limits**: the max and min sizes shown by the tolerance
- **Allowance**: for mating parts – min clearance or max interference
- **Tolerance**: total allowable variance

Nomenclature

- **Maximum material condition (MMC)** - condition of the part to contain max amount of material
- **Least material conditions (LMC)** - condition of the part to contain the min amount of material
- **Clearance fit** - condition of a fit that enables space between parts
- **Interference fit** - condition of a fit that enables no space between the two mating parts
- **Transition fit** - clearance or interference fit

Tolerance Representation

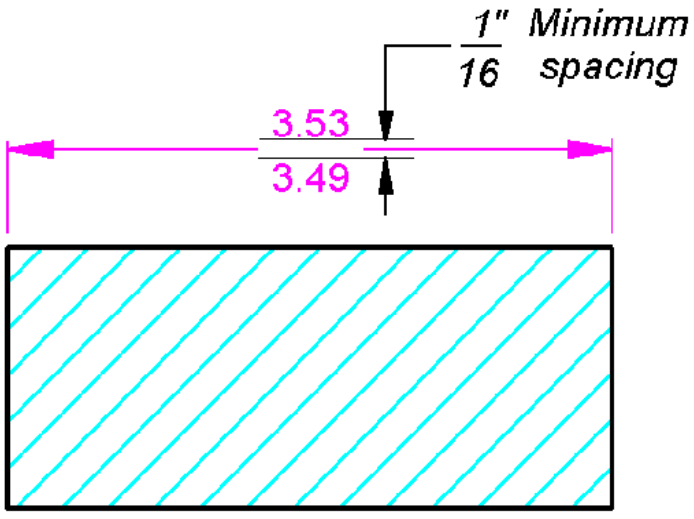
Direct limits
(limit dimensioning)

Tolerance value
(plus or minus dimension)

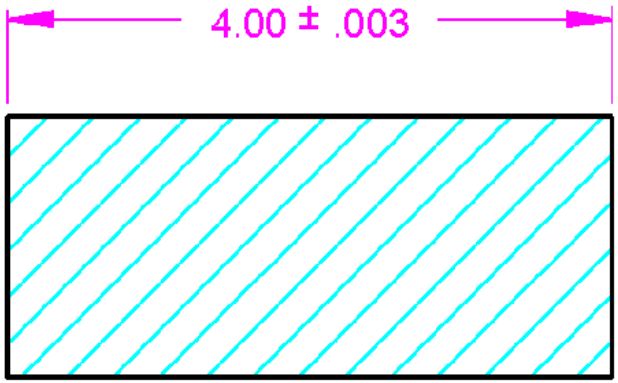
Unilateral Tolerances
(only in one direction from basic size)

Specific note
(* dimension ϕ2±0.001)

General note
(All diameters ϕ2±0.001)

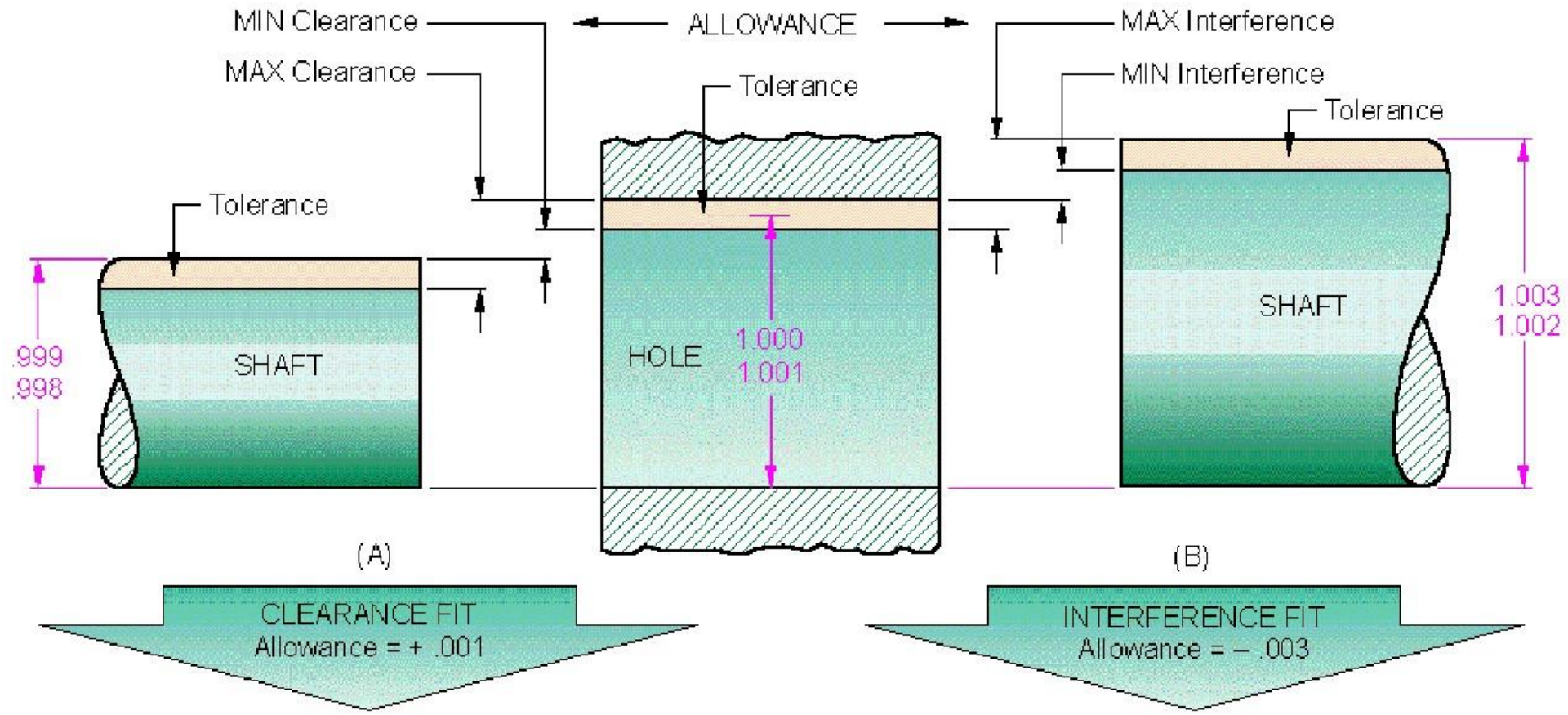


(A) Direct limits



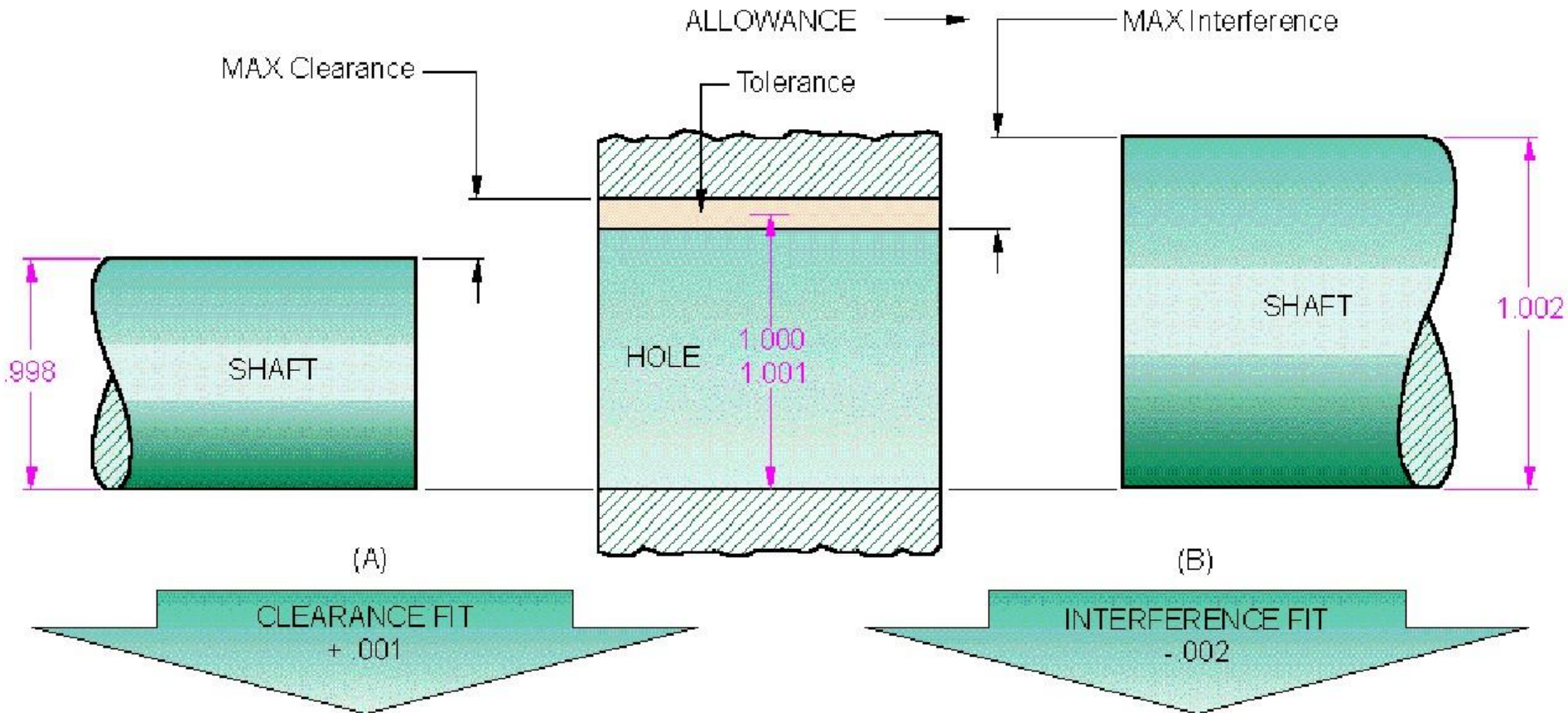
(B) Tolerance values

Clearance and Interference



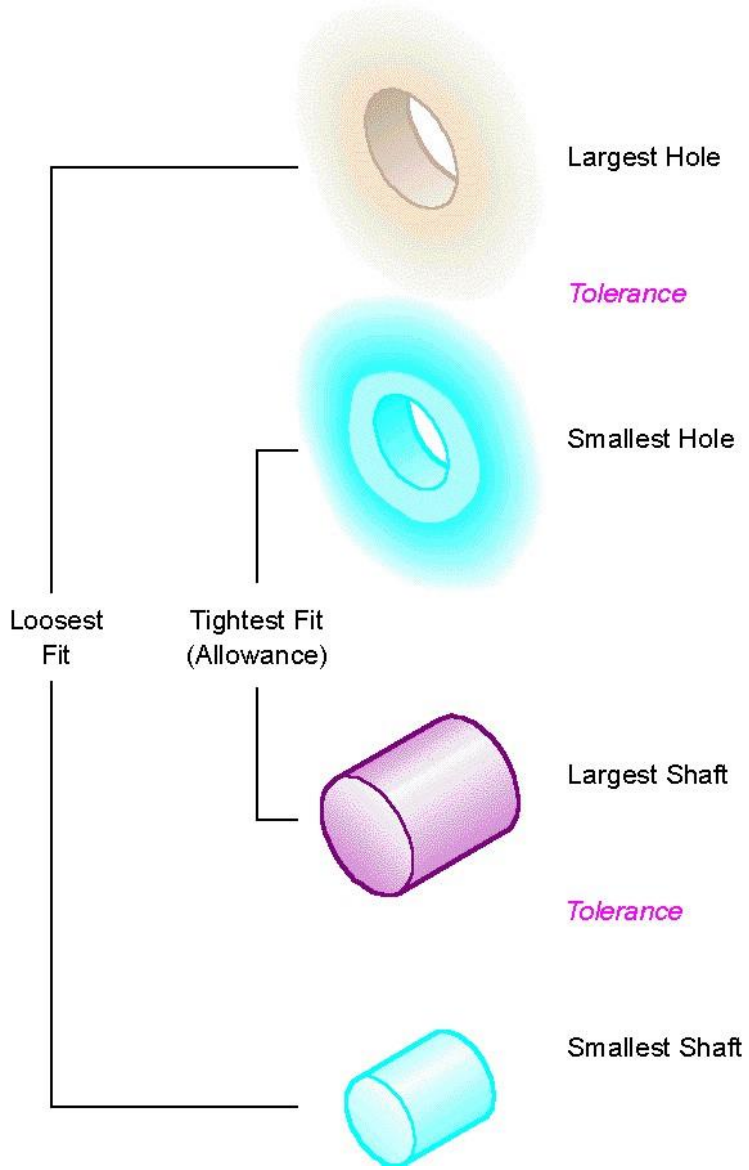
Allowance always equals smallest hole minus largest shaft

Transition Fit



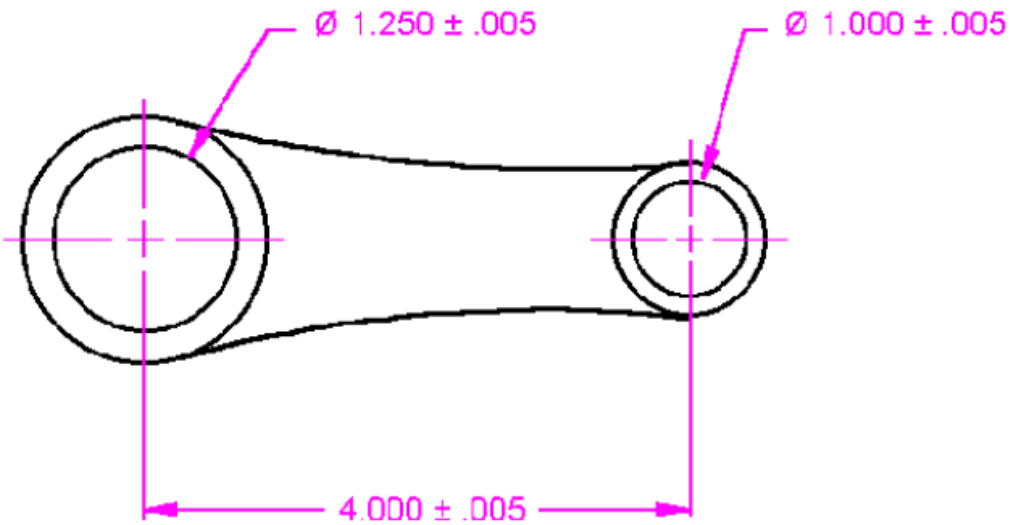
How to Determine the Fits?

Evaluate the allowance and the interference

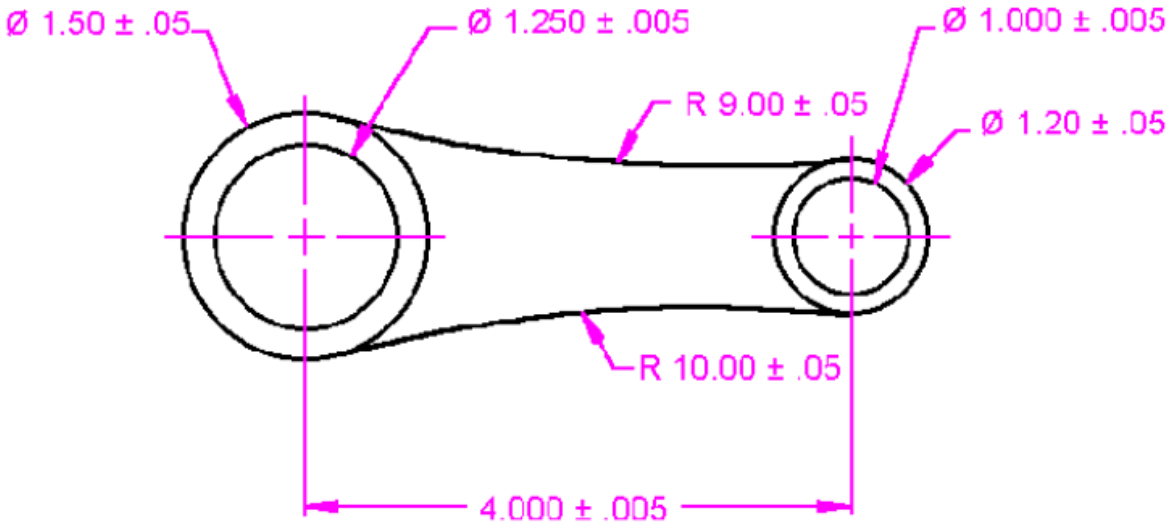


Functional Dimensioning

- Functional dimensioning begins with tolerancing the most important features.
- The functionality of the assembly has to be very clearly established by the designer.
- The assembly procedure as well as the manufacturing processes involved in producing the part must be also clear to the designer.
- Tolerances should be as “coarse” as possible and still permit the satisfactory use of the part – Why?



Functional dimensioning
Step 1



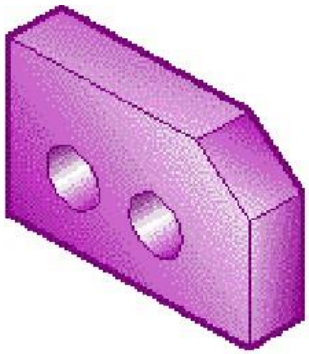
Functional dimensioning
Step 2

Tolerance Stack-up

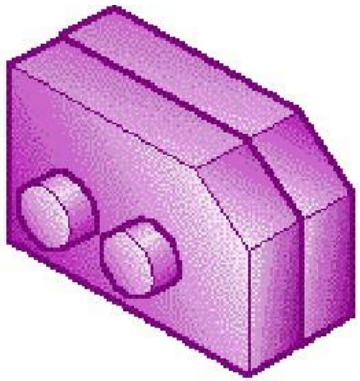
- Tolerances taken in the same direction from one point of reference are additive
  - tolerance stack-up
- Tolerance stack-up can be eliminated with the careful selection and placement of dimensions.

Tolerance Stack-up

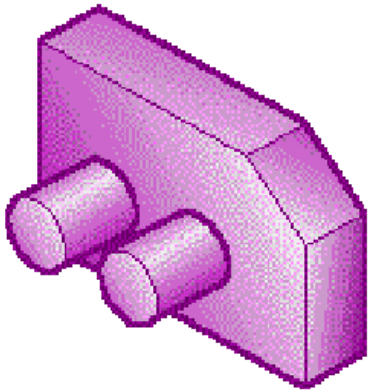
may cause assembly problems



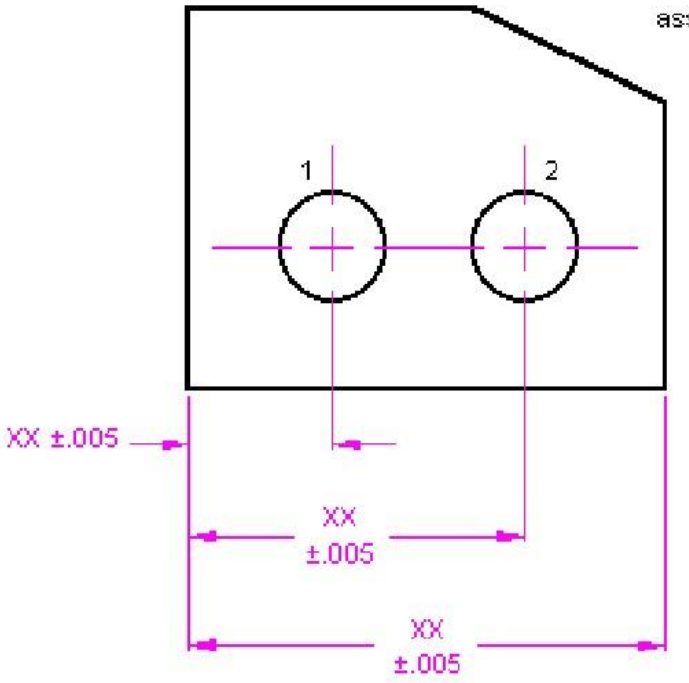
PART A

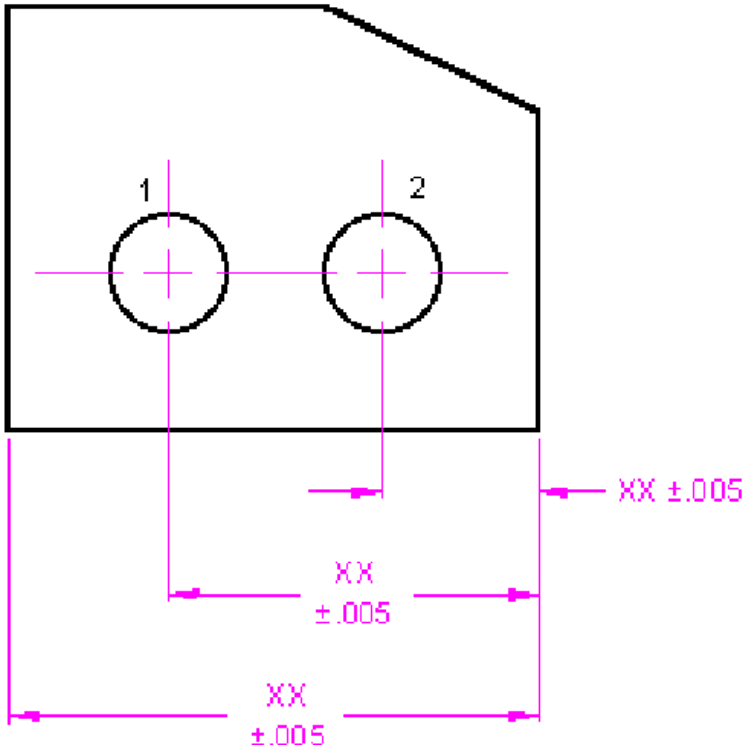


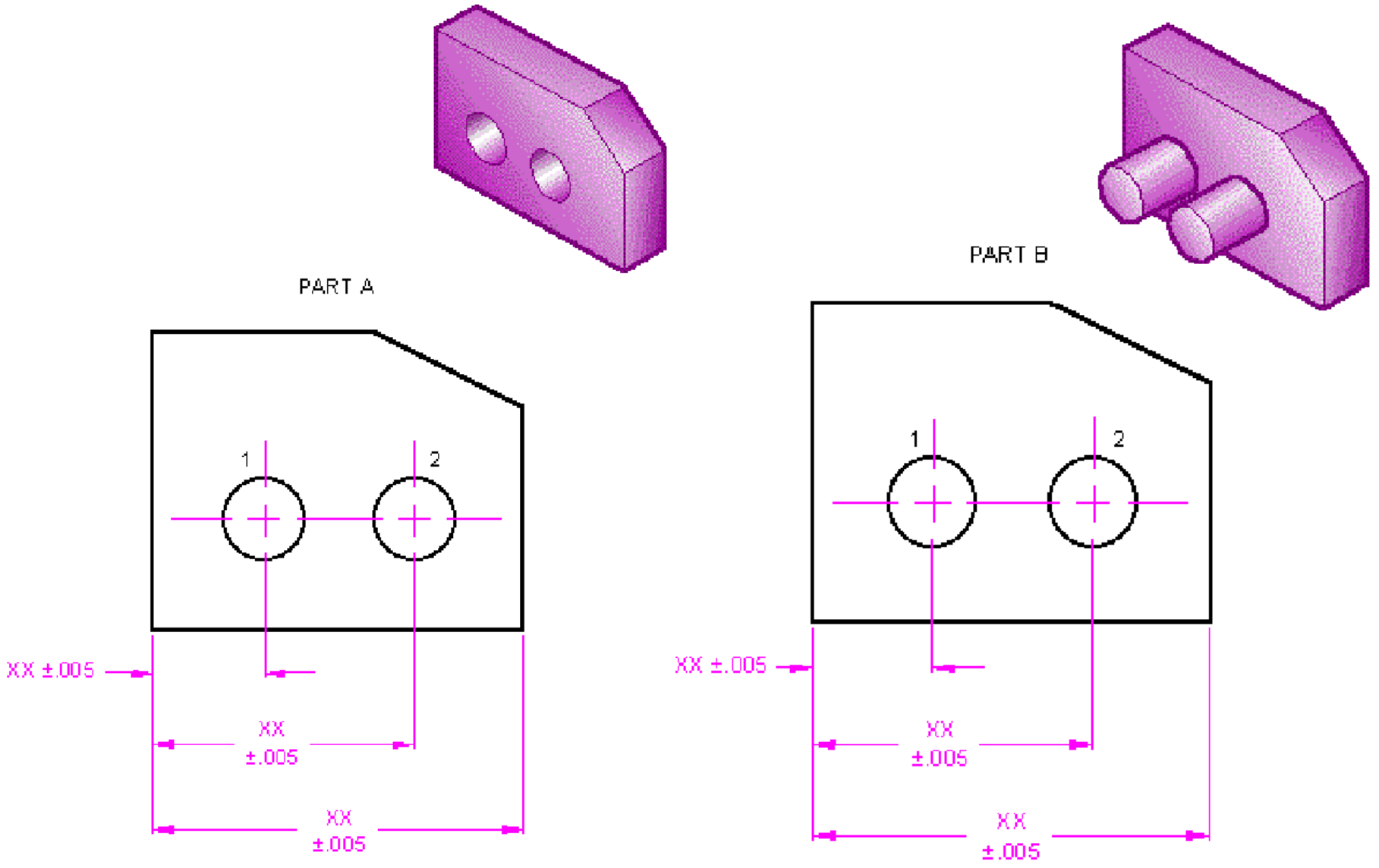
PARTS A and B assembled



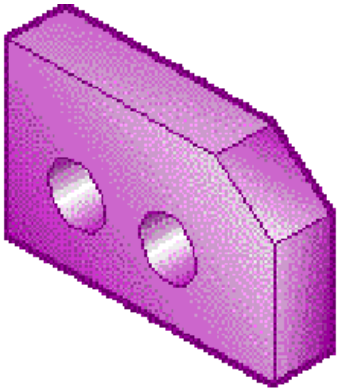
PART B

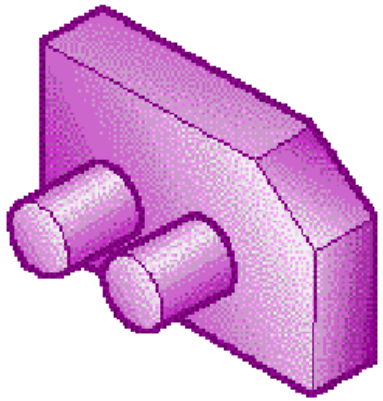




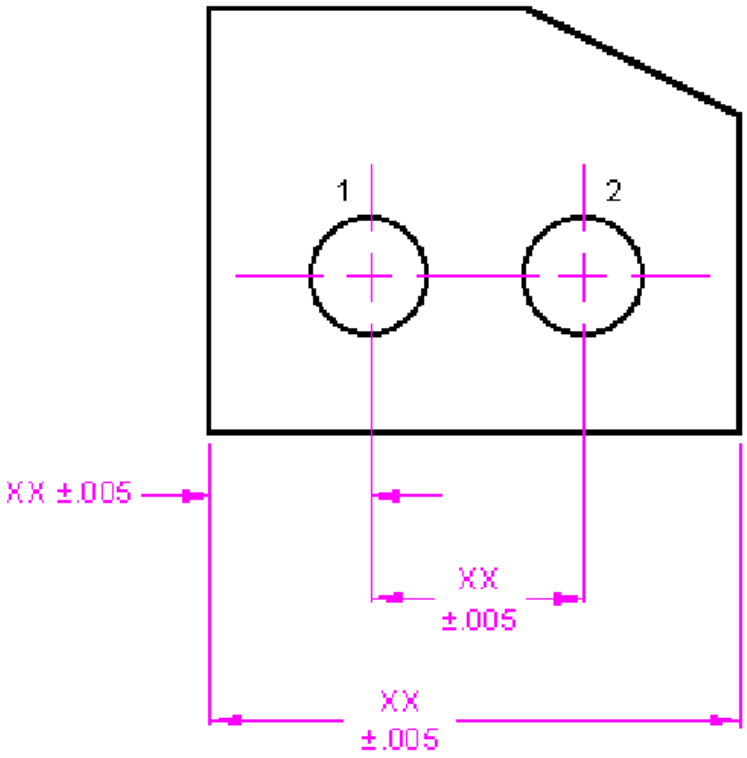


Dimensioning with respect to the base base would help

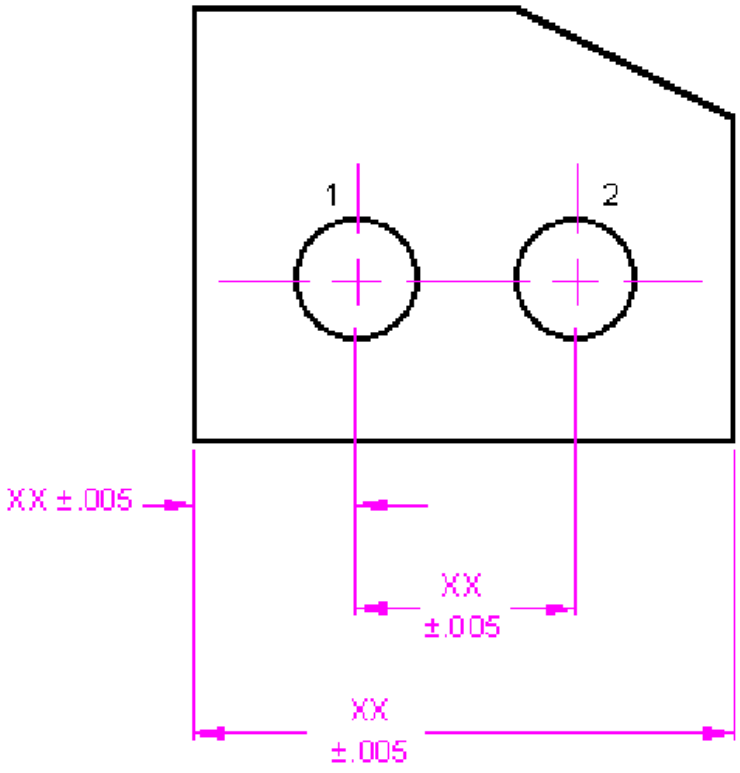




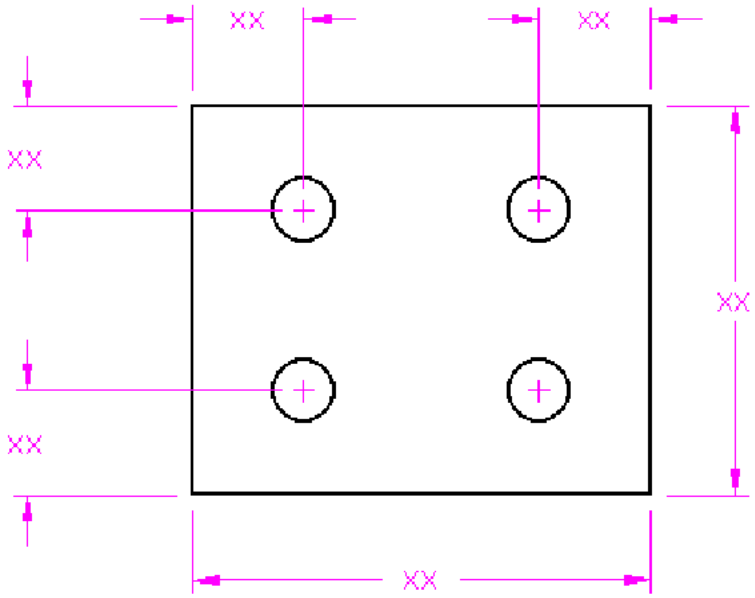
PART A



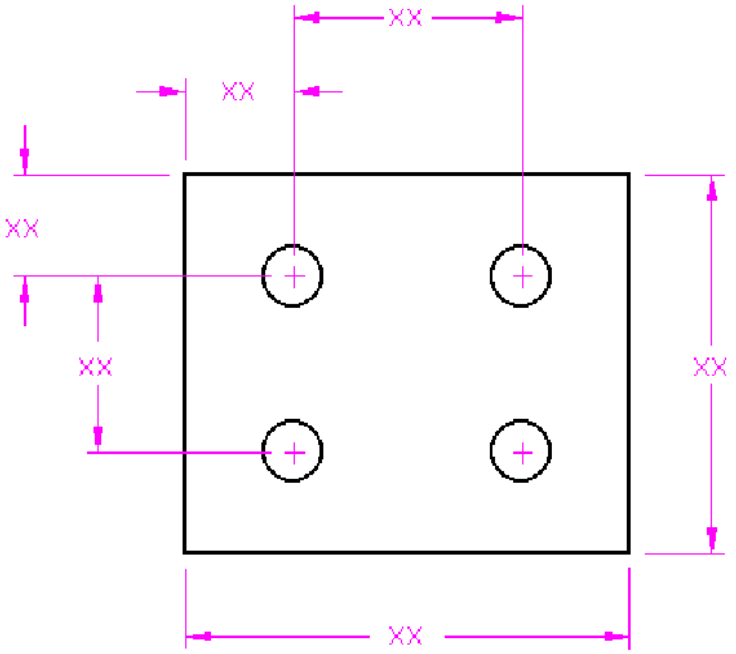
PART B



Providing tolerances for the locating dimensions is a better solution



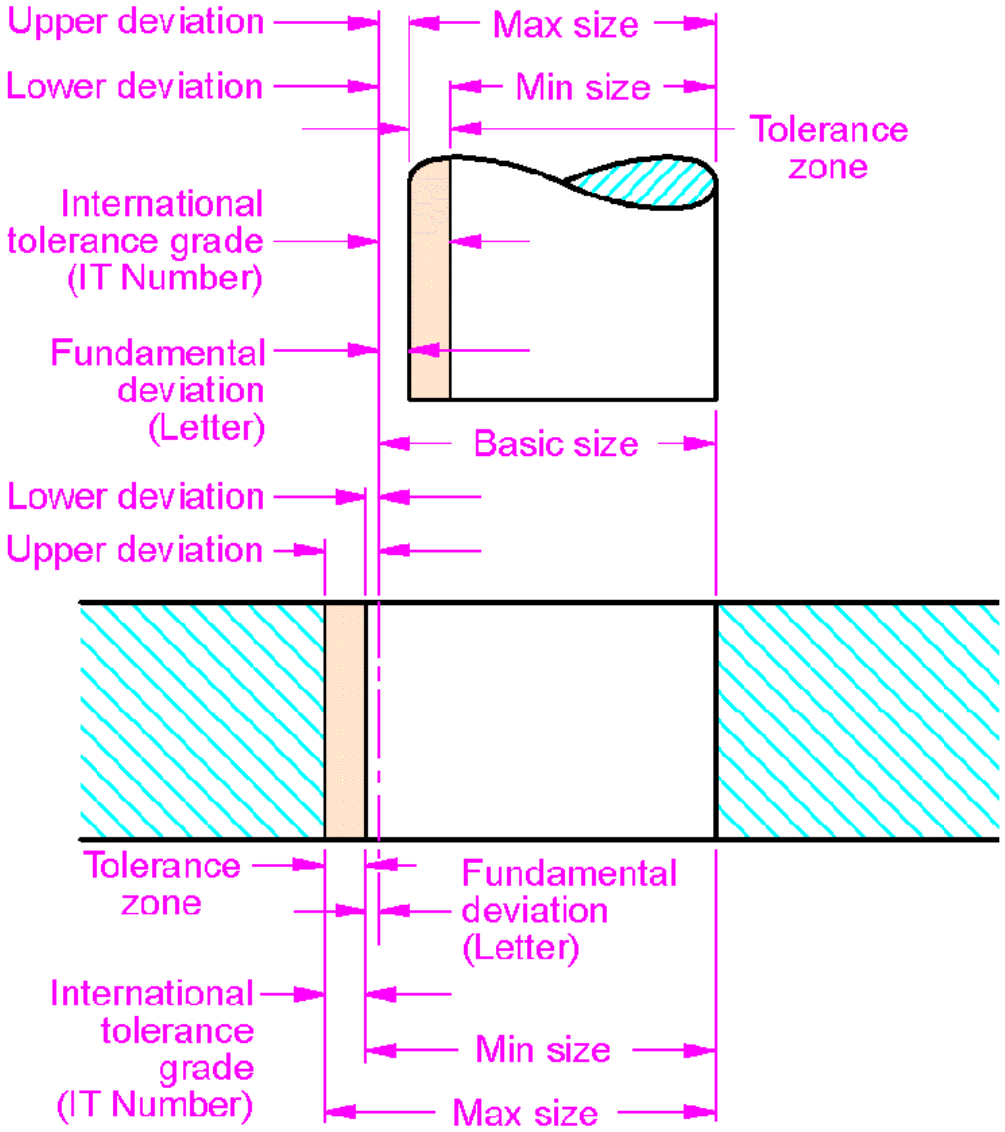
A

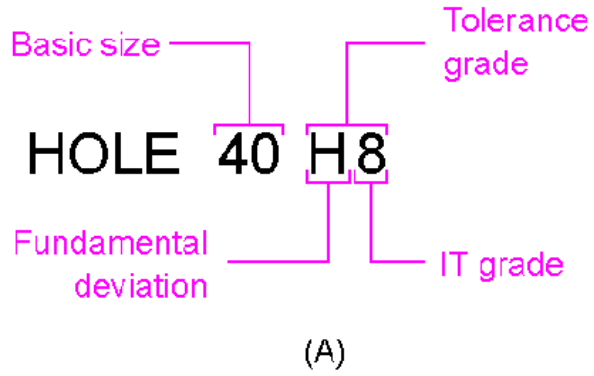


B

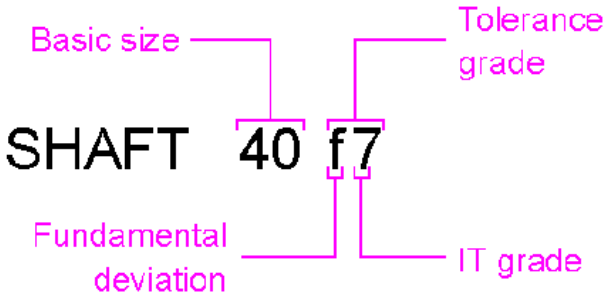
Which of the two dimensioned features is better for assembly purposes?

Tolerancing in ISO

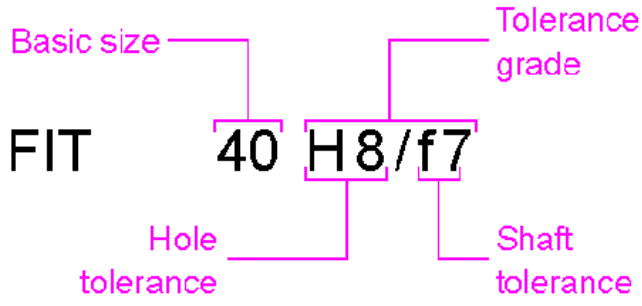




(A)

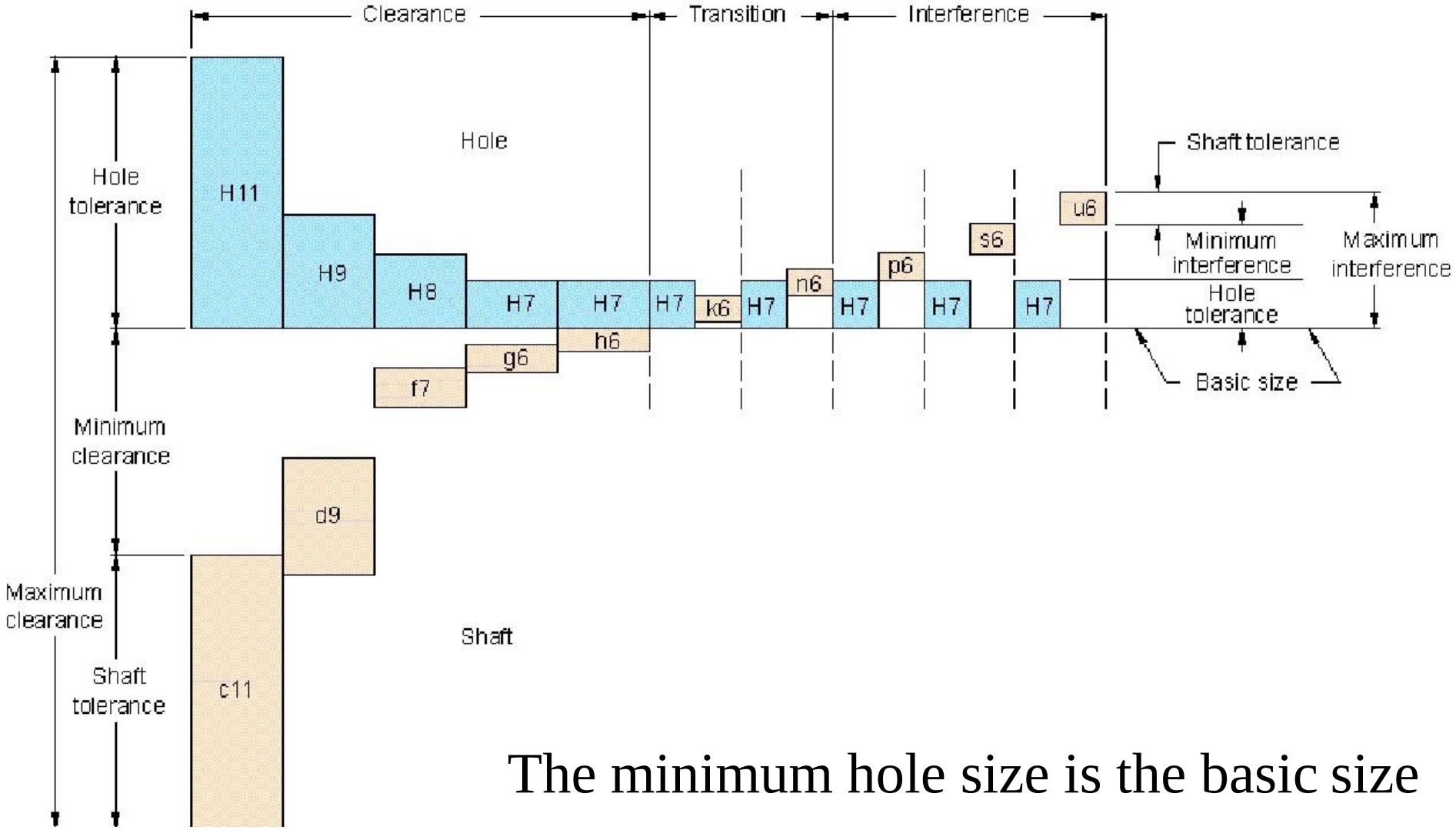


(B)



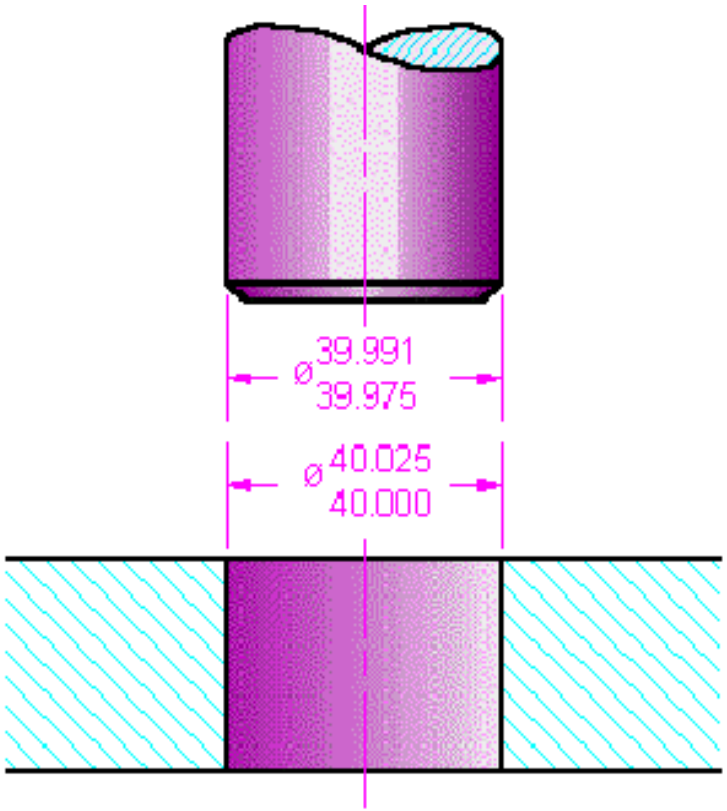
(C)

The Metric Preferred Hole-Based System of Fits



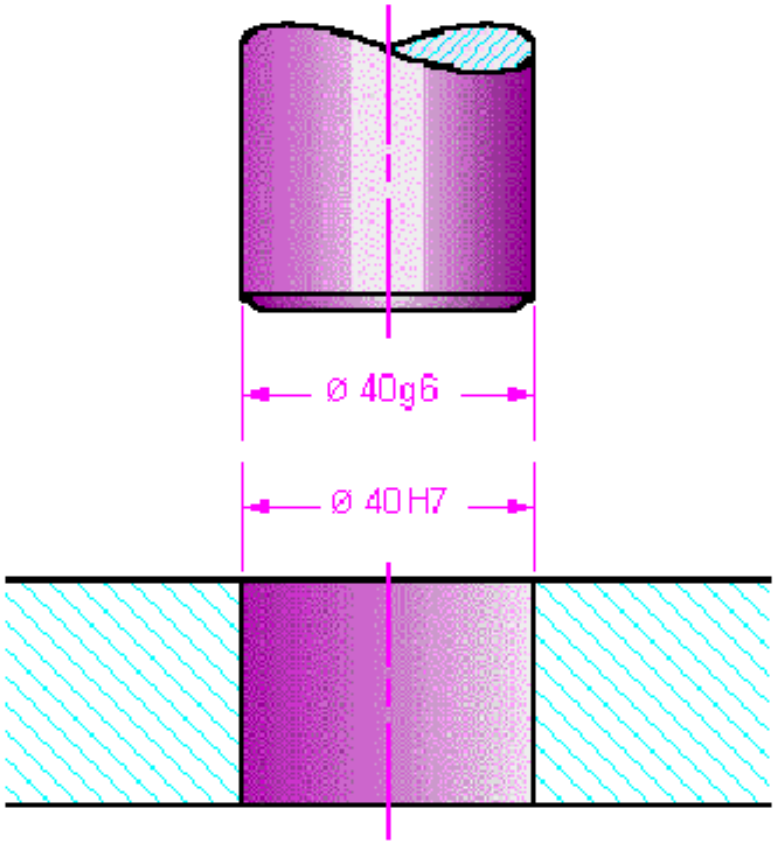
The minimum hole size is the basic size

Line Form vs. Note Form Tolerancing



Method 1
Limit Form

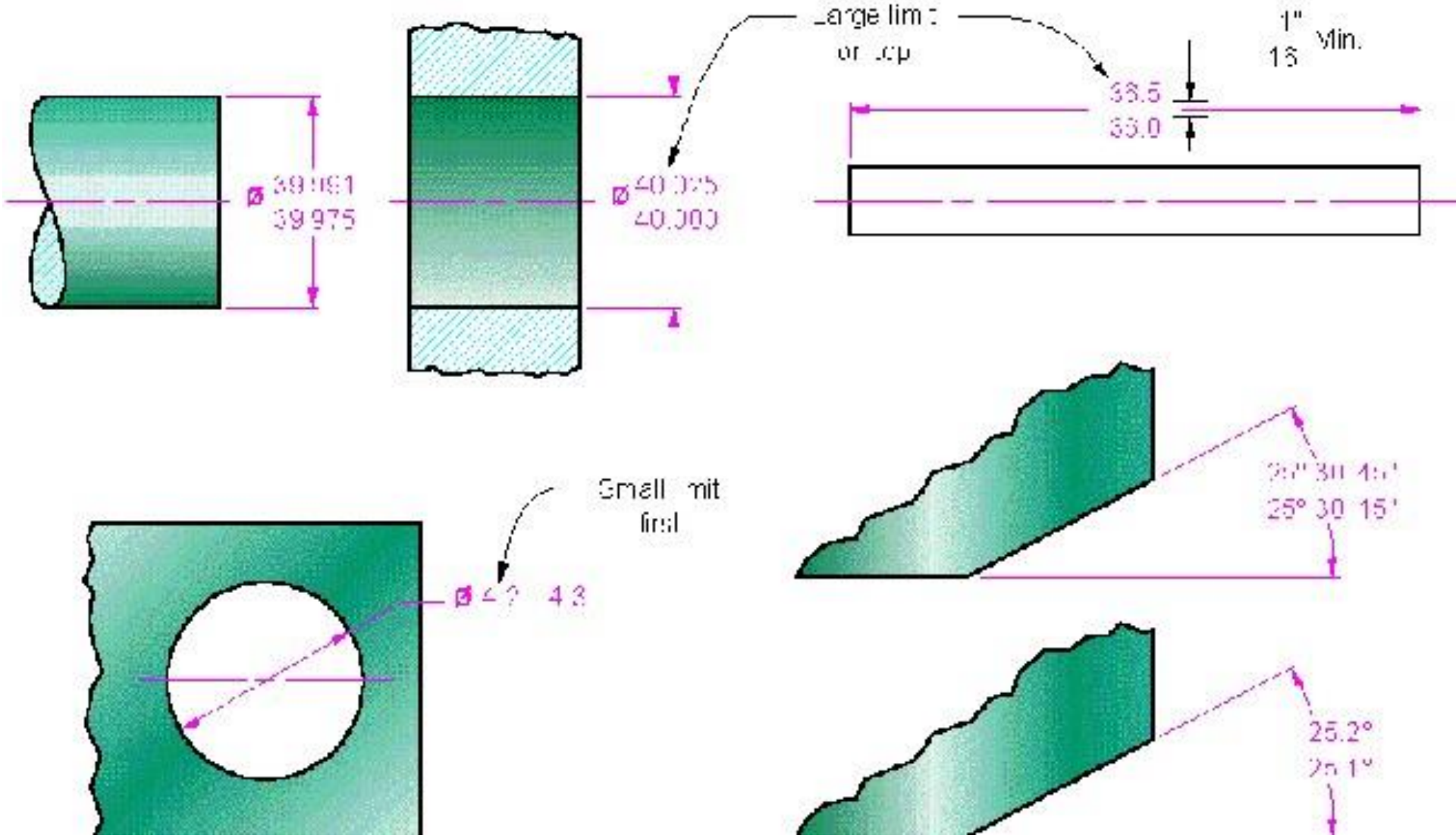
Hole	Shaft
40.025	39.991
40.000	39.975
.025 Hole tolerance	.016 Shaft tolerance



Method 2
Note Form

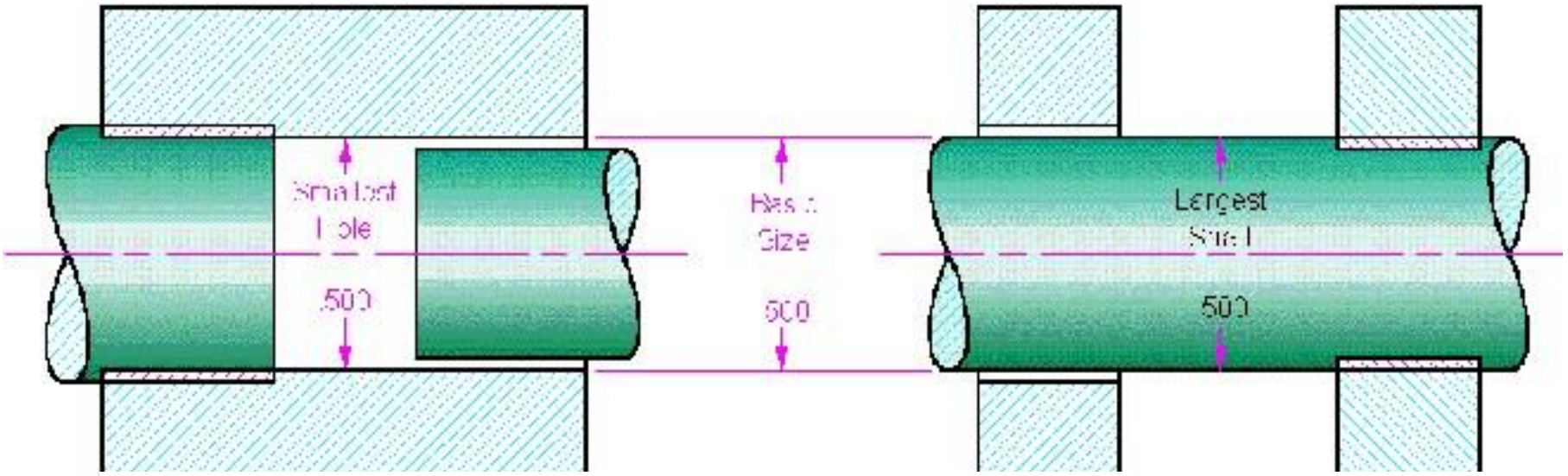
Hole	Shaft
40.025	39.991
40.000	39.975
.025 Tightest fit	.016 Loosest fit

Standard Representation of Metric Tolerances



Basic Hole and Shaft System

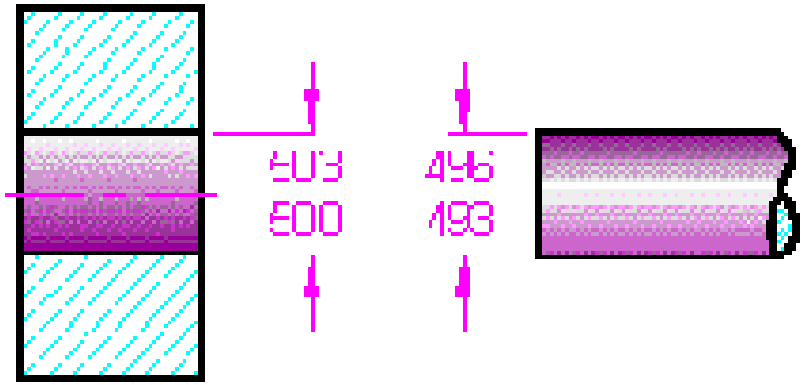
Imperial Size



Interference fit Clearance fit Clearance fit Interference fit

Basic hole system **Basic shaft system**

RUN FIT



0.500 is the smallest hole

0.496 is the largest shaft

0.004 is the tightest fit

0.503 is the largest hole

0.493 is the smallest shaft

0.10 is the loosest fit

0.500 is the lower limit hole

0.496 is the upper limit shaft

0.004 is the ALLOWANCE

0.496 is the upper limit shaft

0.003 is the shaft tolerance

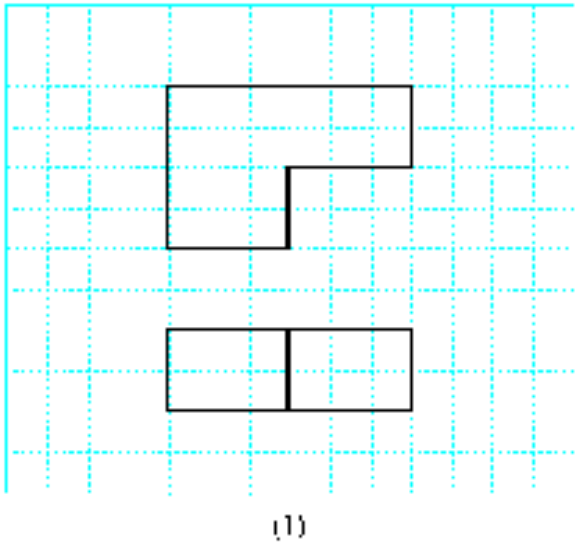
0.493 is the LOWER LIMIT SHAFT

0.500 is the lower limit hole

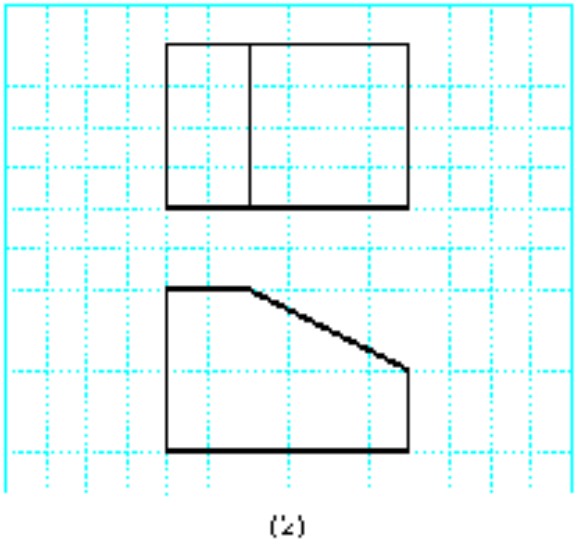
0.003 is the hole tolerance

0.503 is the UPPER LIMIT HOLE

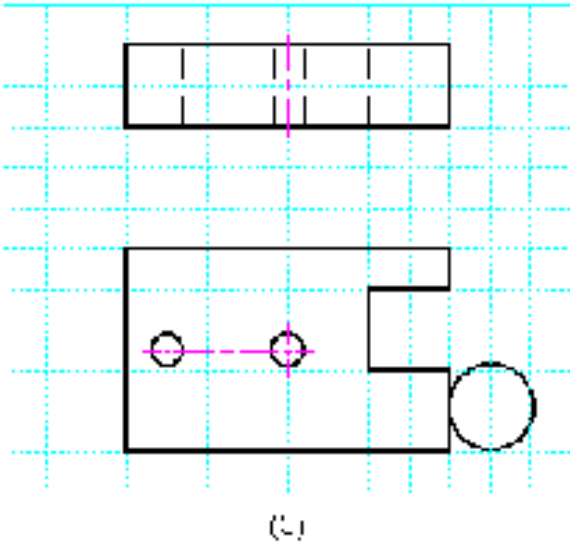
Sample Dimensioning Problems



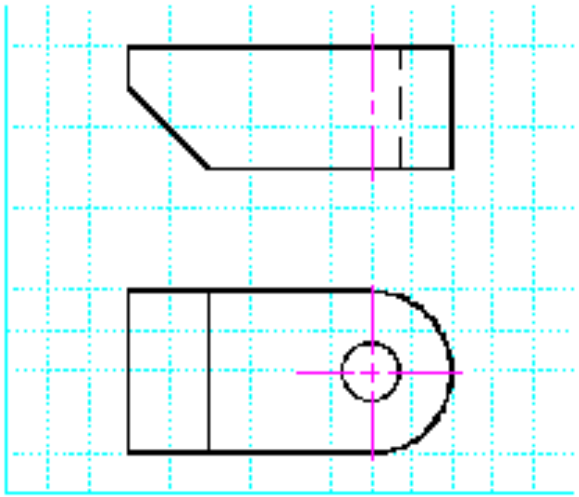
(1)



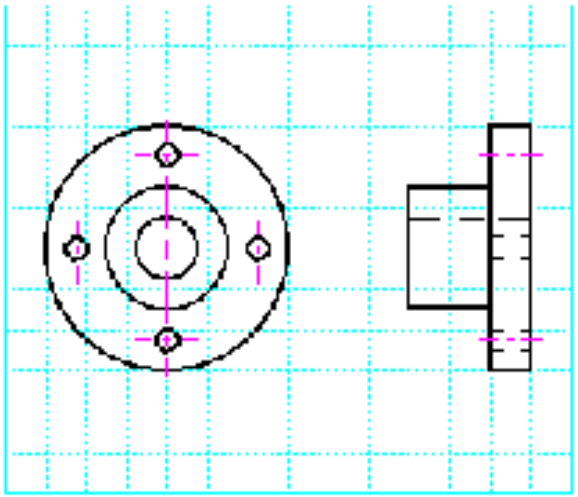
(2)



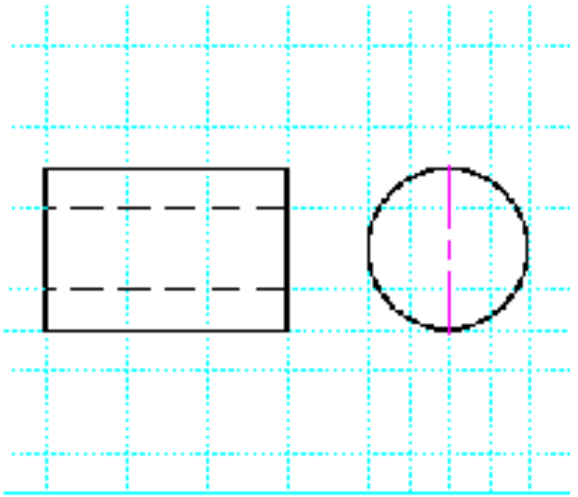
(3)



(4)

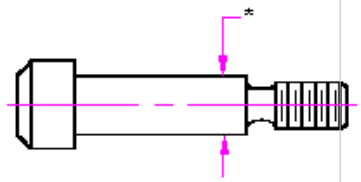


(5)

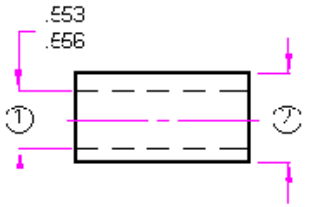


(6)

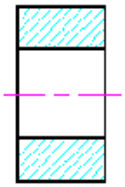
Tolerancing Sample Problem



*See appendix for American Standard Socket-head Shoulder Screw



Bushing

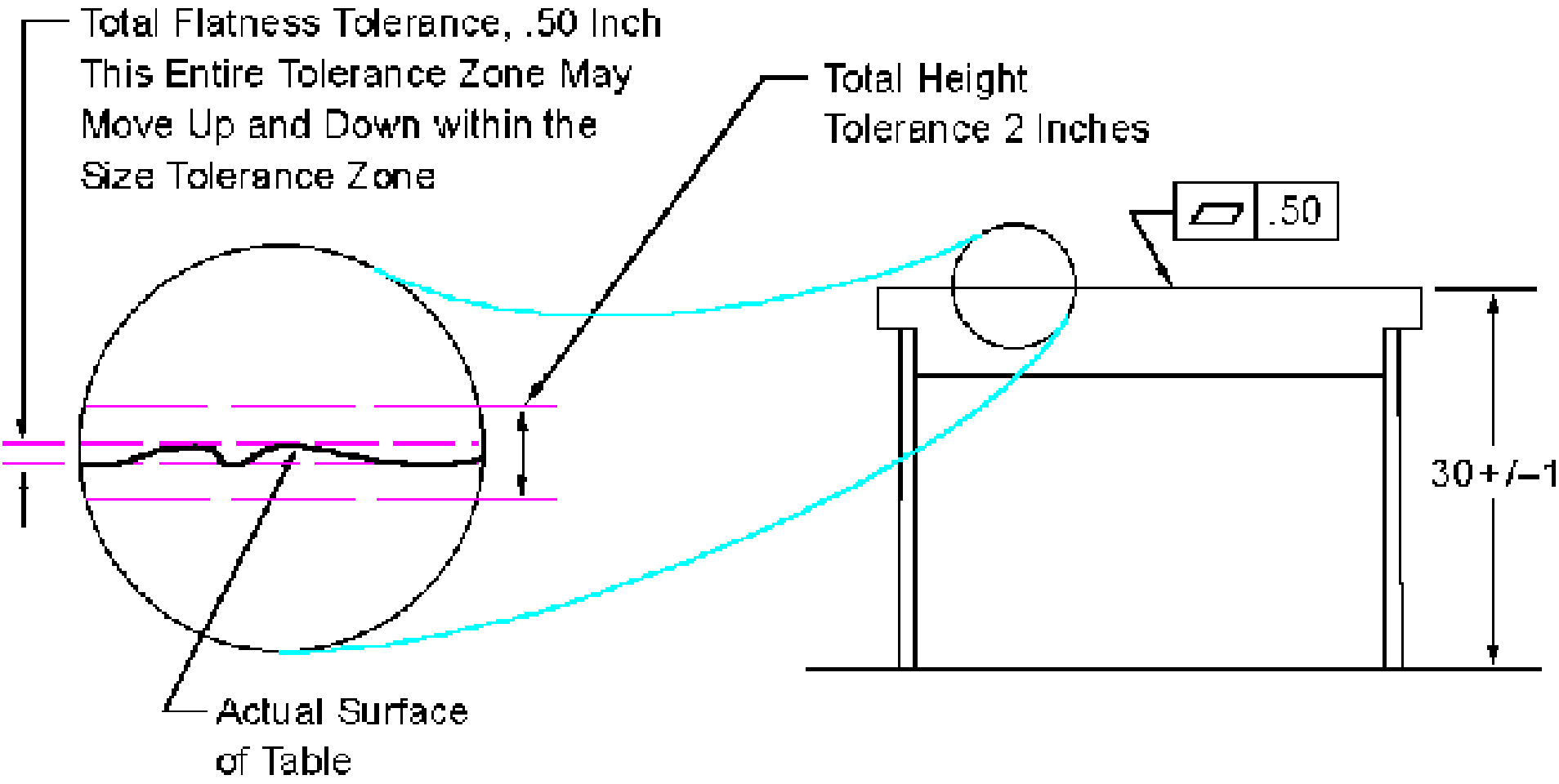


Housing

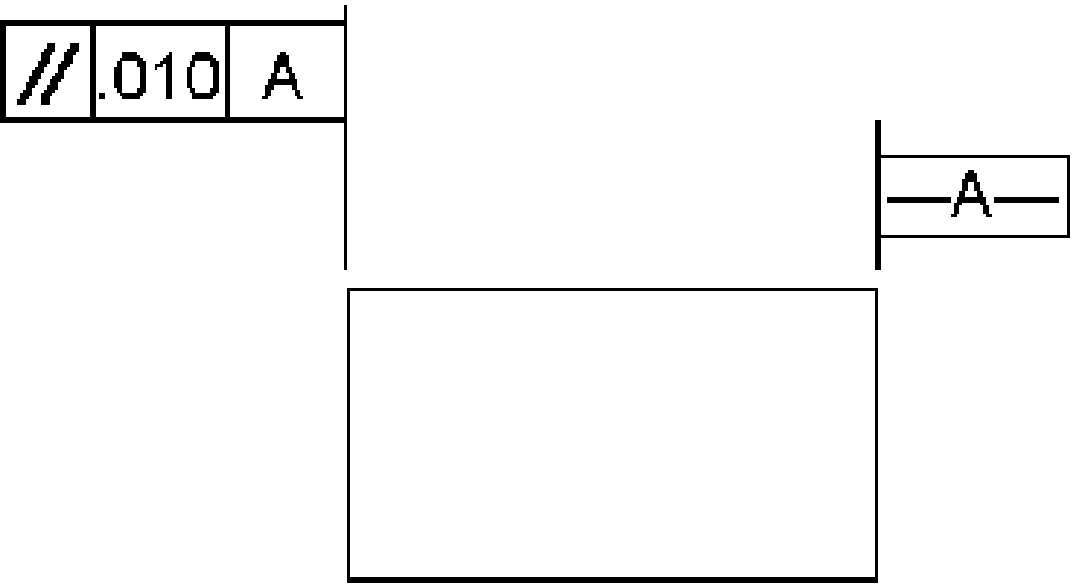
Limits of size			1		2 - FN 4 fit
Hole		Nominal size	.500		.750
	±	Limit	±	±	
	=	Upper limit	=	=	
		Nominal size	.500		.750
	+	Limit	+	+	
	=	Lower limit	=	=	
Shaft		Nominal size	.500		.750
	±	Limit	±	±	
	=	Upper limit	=	=	
		Nominal size	.500		.750
	±	Limit	±	±	
	=	Lower limit	=	=	
Limits of fit		Smallest hole			
	-	Largest shaft	-	-	
	=	Tightest fit	=	=	
		Largest hole			
	-	Smallest shaft	-	-	
	=	Loosest fit	=	=	

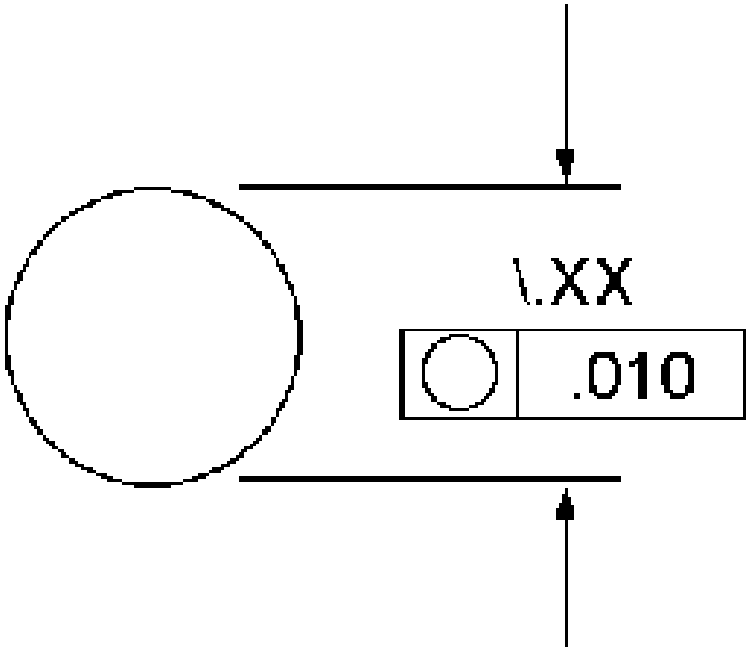
Geometric Tolerancing

Used to limit the abatement in the geometric variation of features.

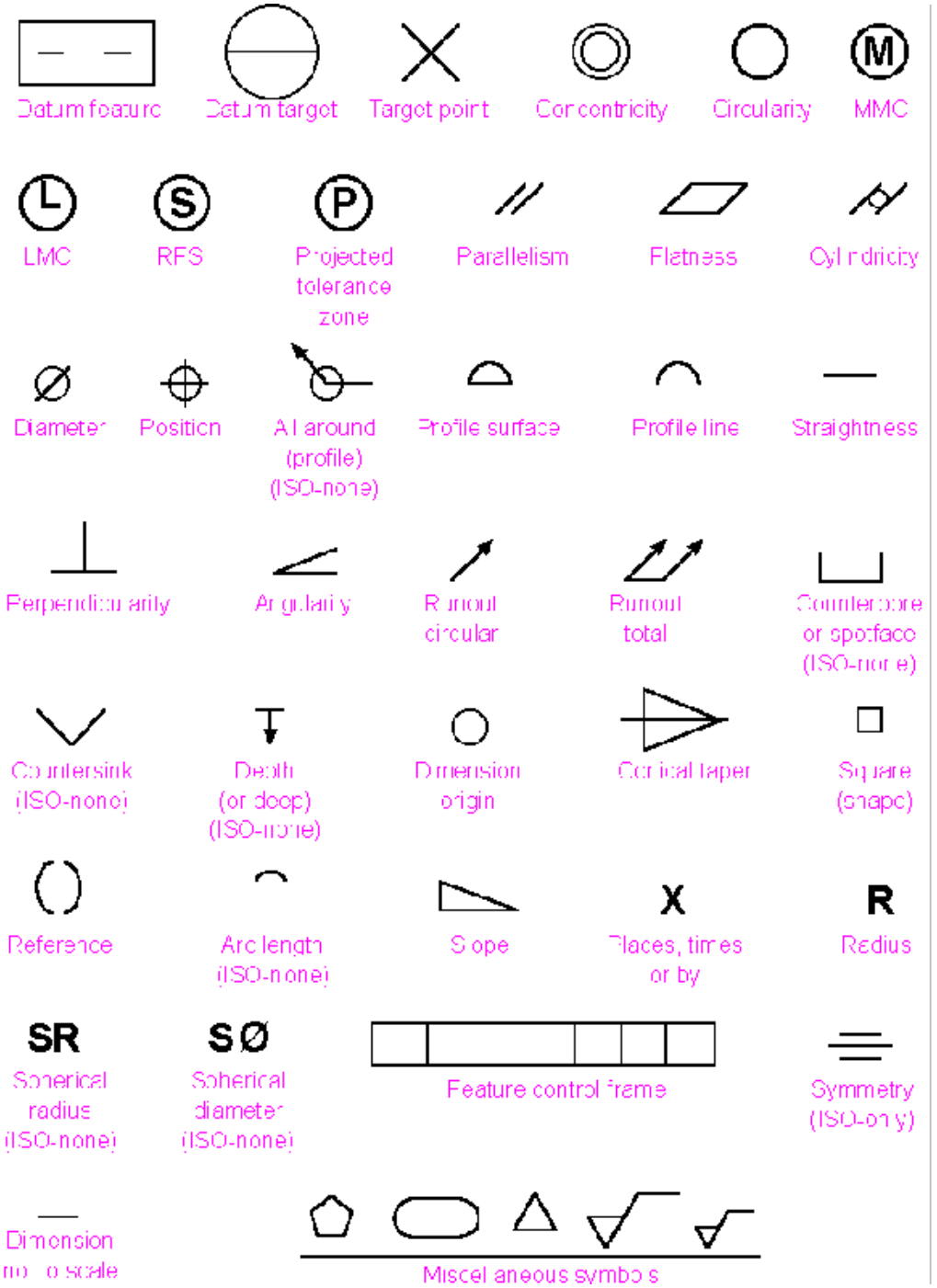


Feature Control Frames Example

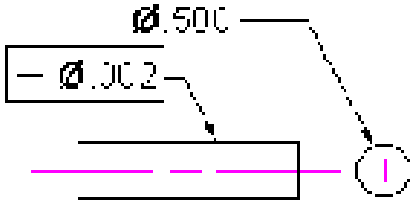




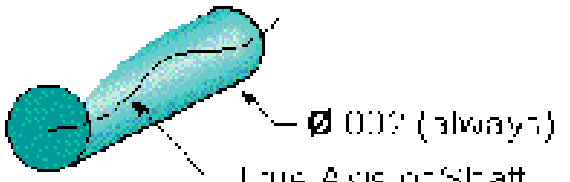
Dimensioning and Tolerancing Symbols

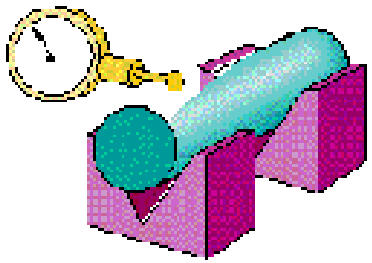


Axis Straightness

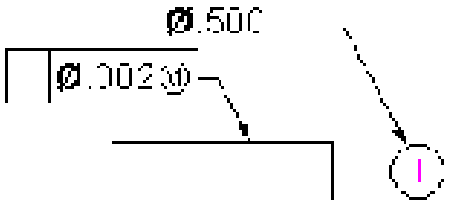


RFS Basis

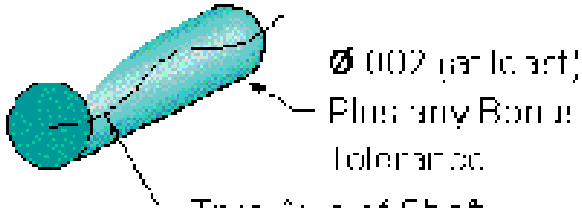




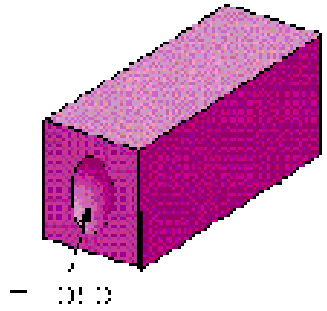
MMC Modified



Drawing

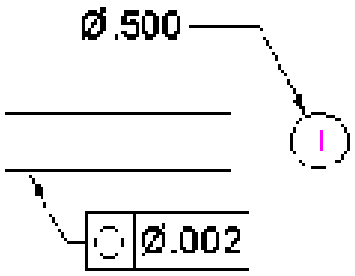


Effect
(scale enlarged)

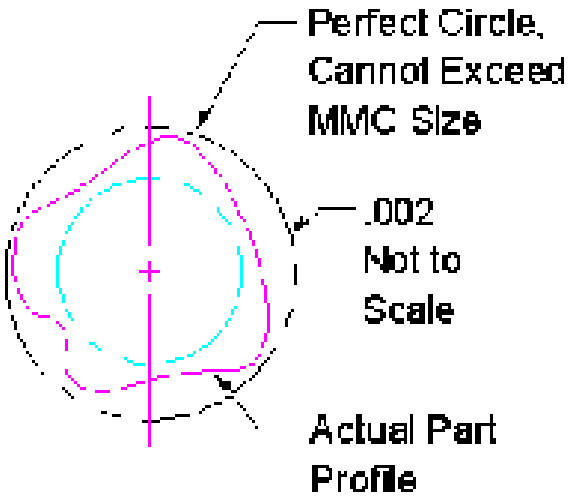


Inspection
Methods

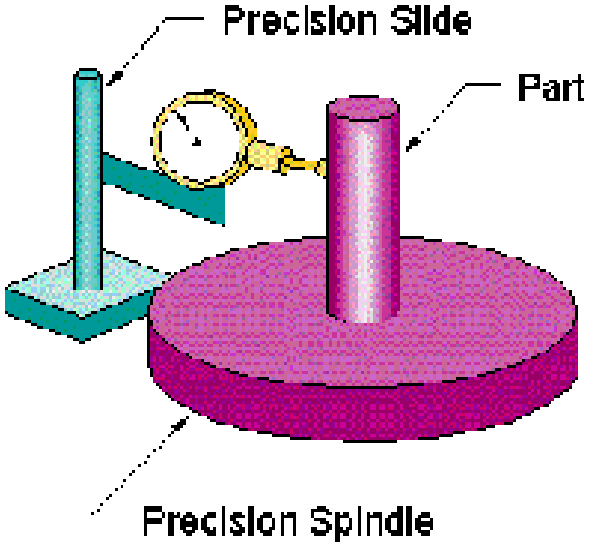
Roundness



Drawing

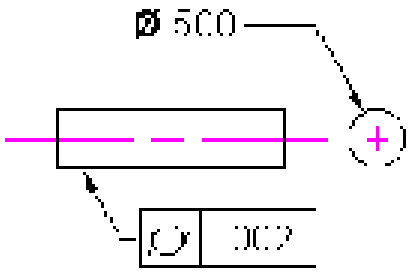


Tolerance Zone

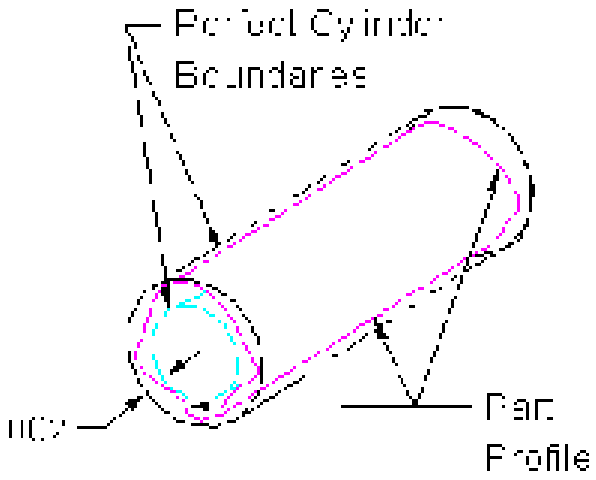


Inspection Method
(check only one slice at a time)

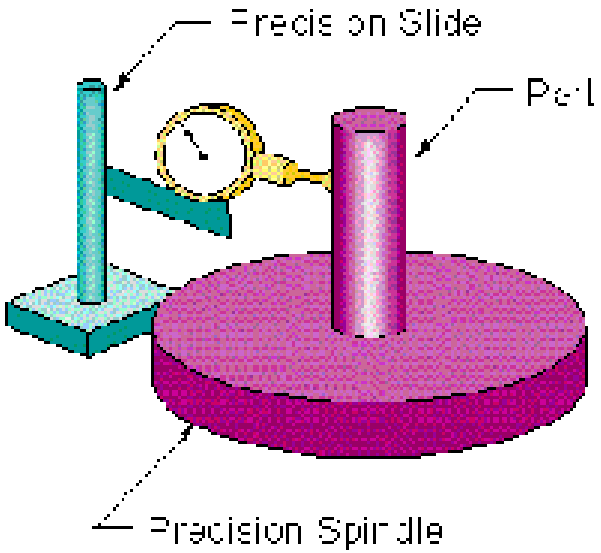
Cylindricity



Drawing

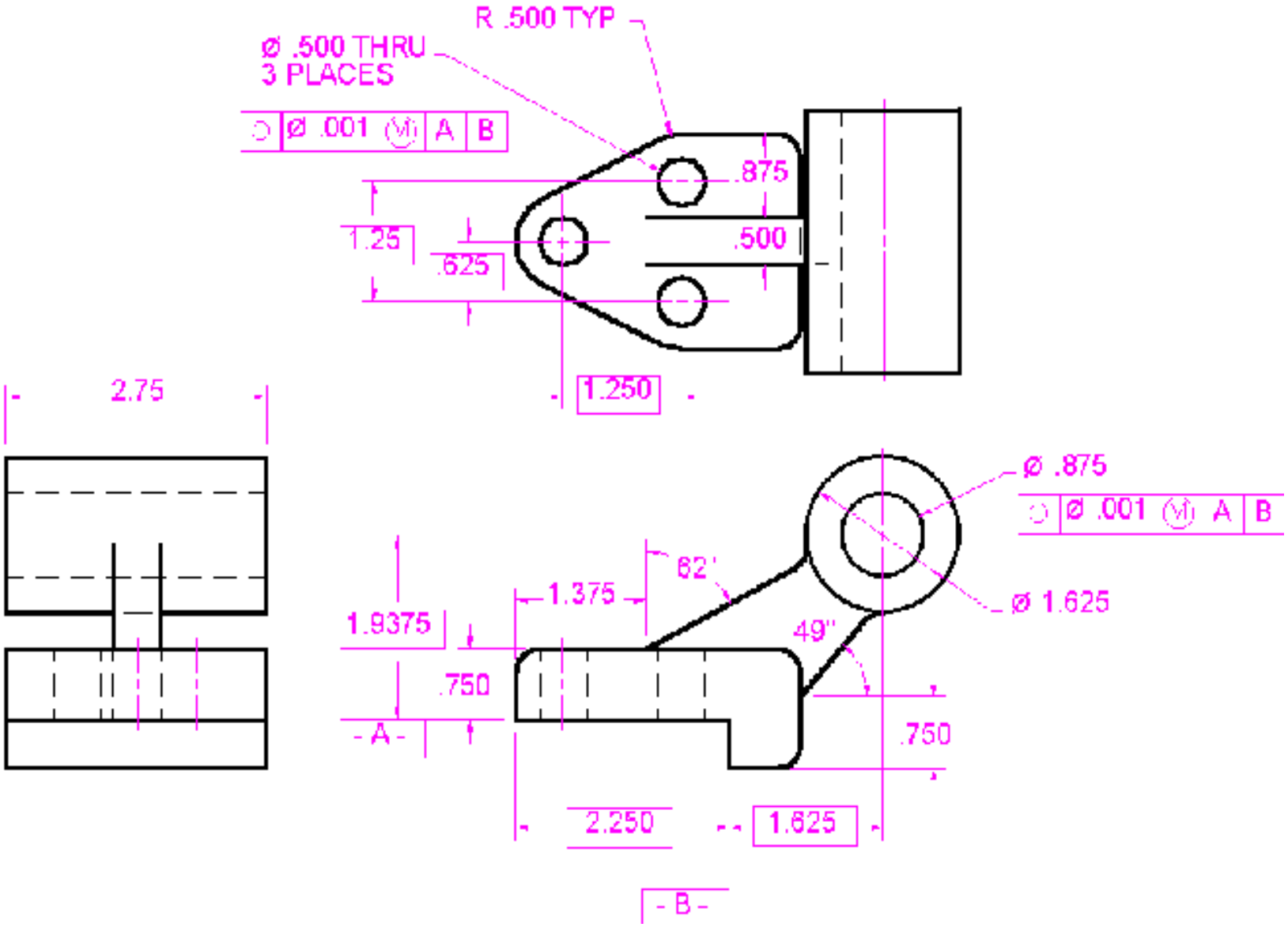


Ideal

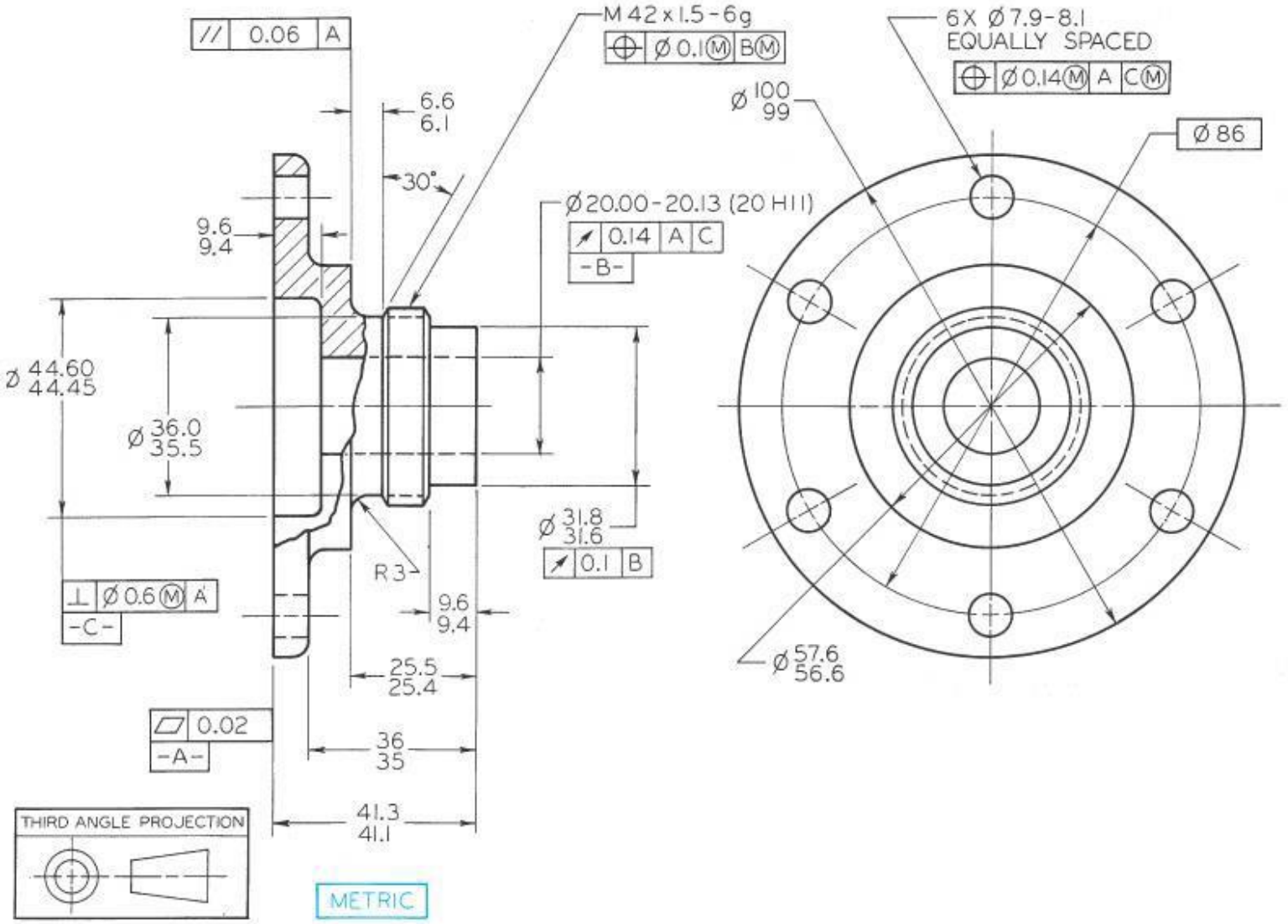


Inspection Method

Bracket with GT Example

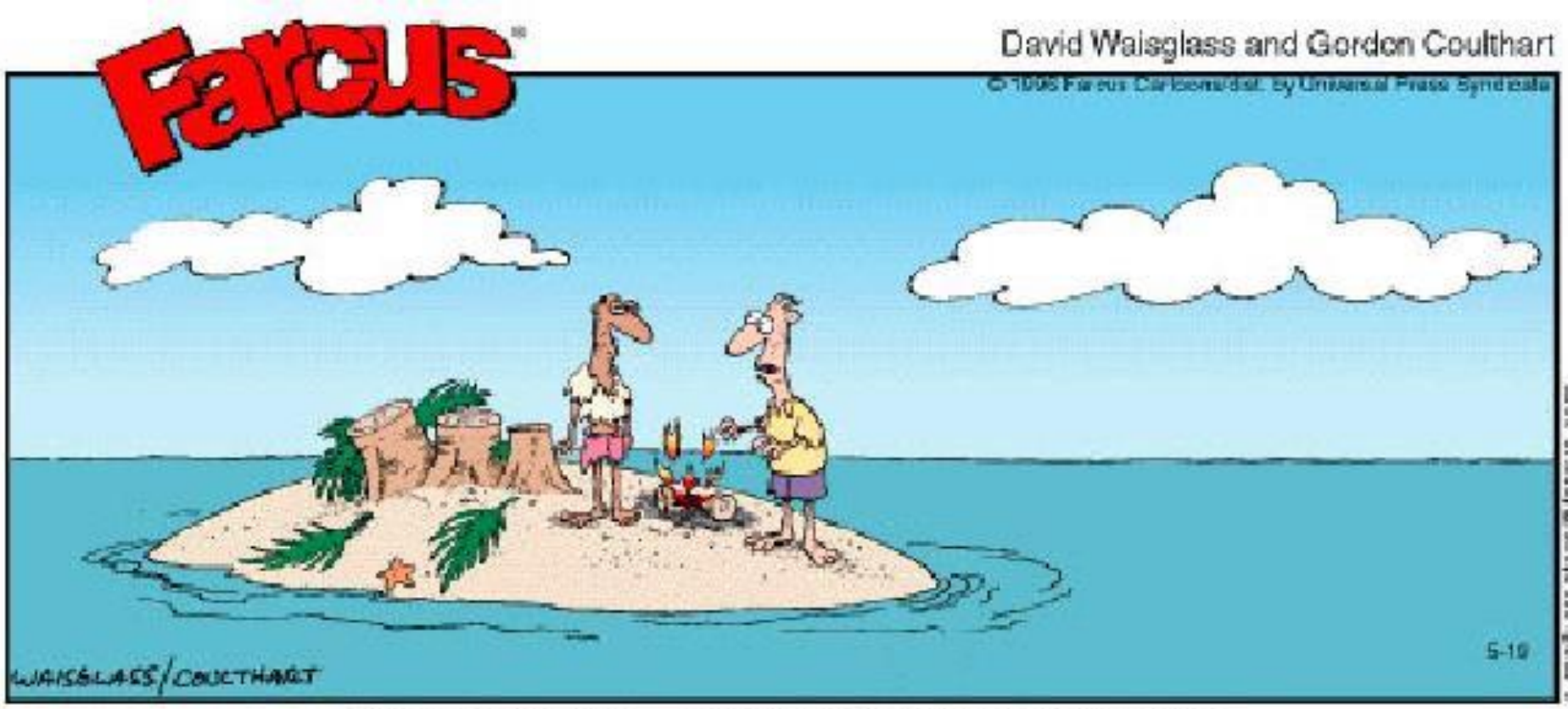


Drawing with GT Example



Think Practical

when approaching a design problem



"You were right ... we should have built a raft."