# Geometric Dimensioning

GD

By AMIT Kr. SRIVASTAVA ( AkS)

# Rules and Practice

- **Accurate dimensioning is one of the most demanding undertakings when designing parts.**
- ***Use the checklist to insure you have followed the basic dimensioning rules.***
- ***Keep in mind there may be a case where the need to break a standard could occur to give clarity to the part and manufacturer.***

- GD&T is a method of defining parts by how they function.
- The method has been developed over the last forty years.
- It allows a designer to define the features of a part without increasing tolerances.
- The current standard is ASME Y14.5M-1994.

# WHY IS GD&T IMPORTANT

- Saves money
  - For example, if large number of parts are being made – GD&T can reduce or eliminate inspection of some features.
- Ensures design, dimension, and tolerance requirements as they relate to the actual function
- Ensures interchangeability of mating parts at the assembly
- Provides uniformity
- It is a universal understanding of the symbols instead of words

# Standards

- In order for the drawings to be dimensioned so that all people can understand them, we need to follow standards that every company in the world must follow. Standards are created by these organizations:
  - -ANSI
  - -ISO
  - -DIN
  - -JIS
  - -MIL
  - -DOD
  - -CEN

# Standards Institutions

- ANSI – American National Standards Institute – This institute creates the engineering standards for North America.

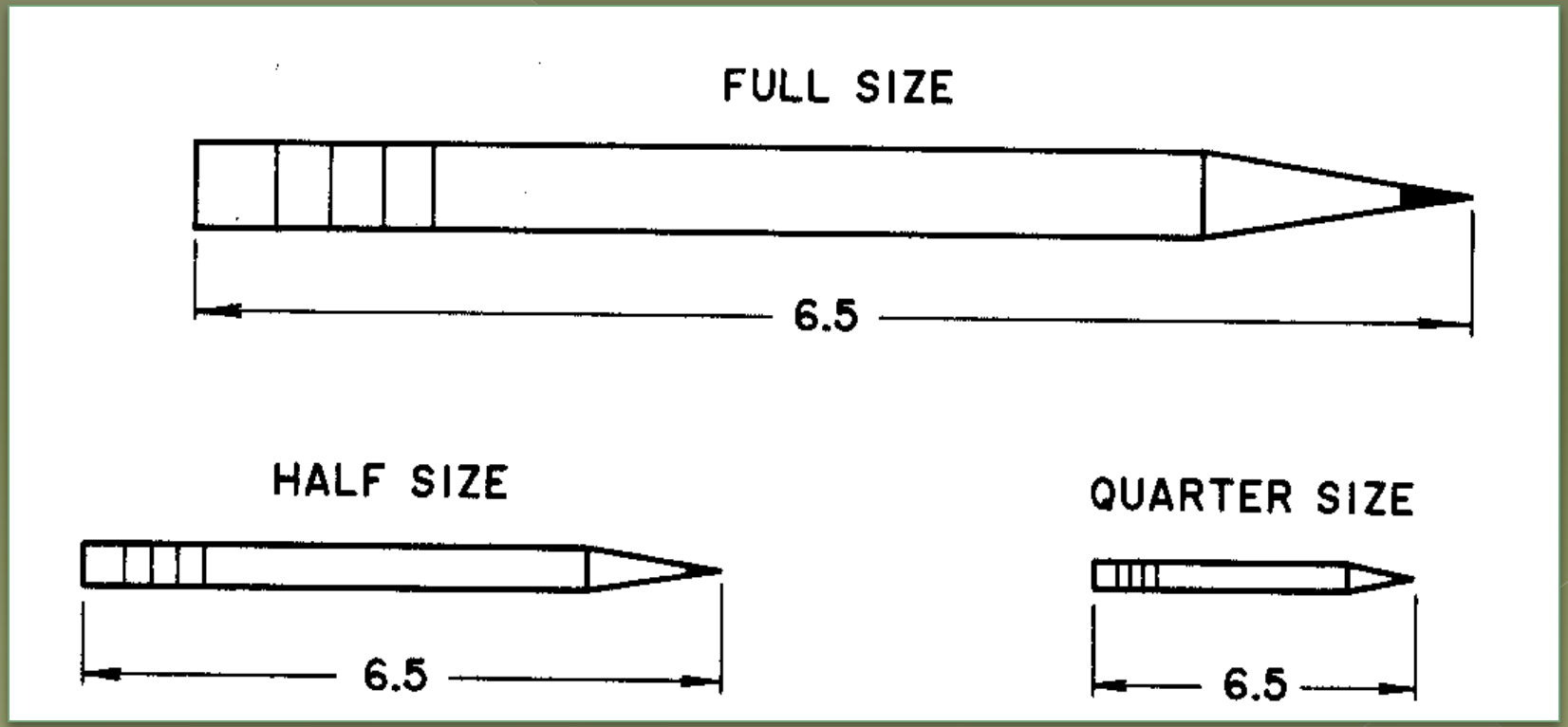
- ISO – International Organization for Standardization – This is a world wide organization that creates engineering standards with approximately 100 participating countries.

# Standards Institutions

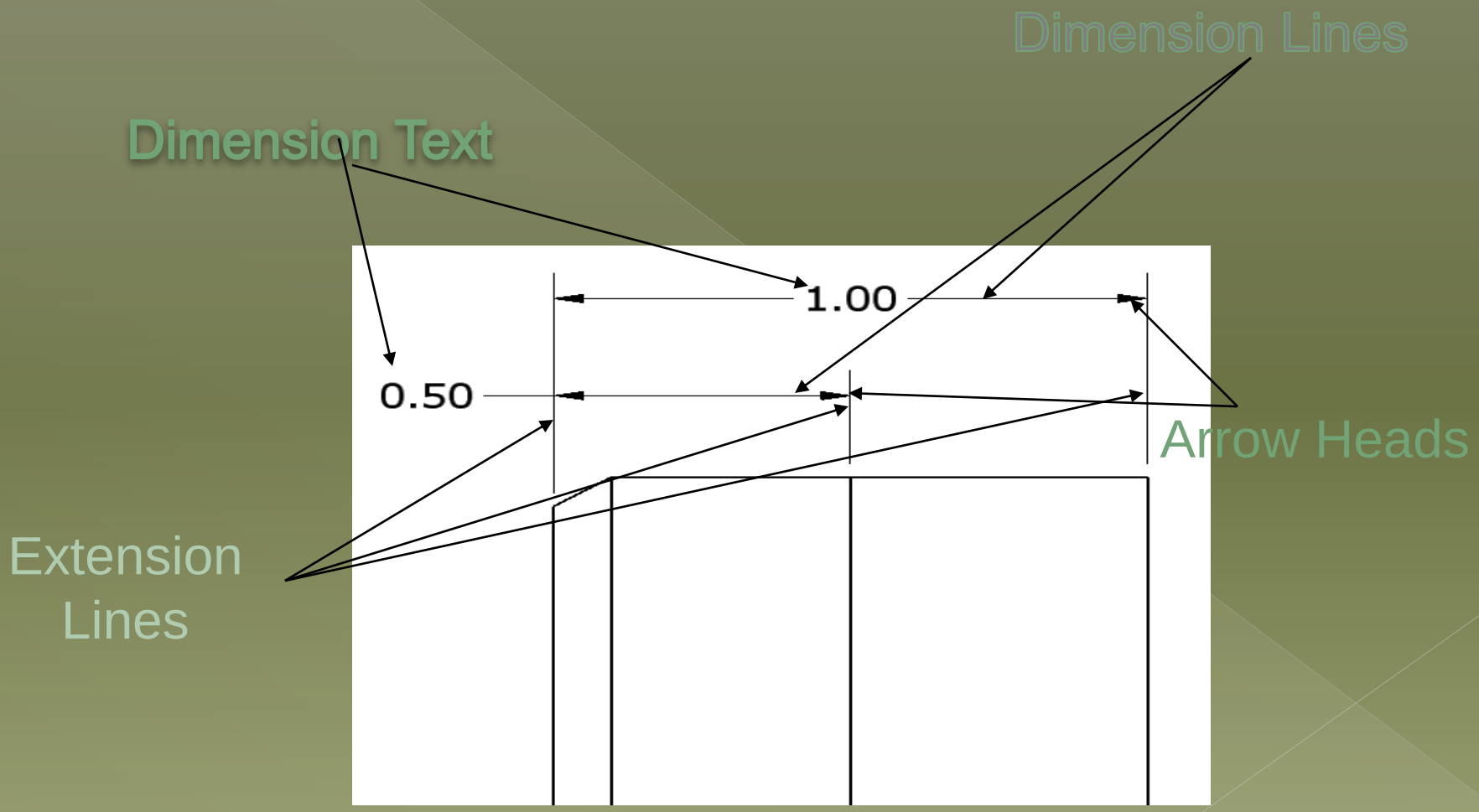
- DIN – Deutsches Institut für Normung – The German Standards Institute created many standards used world wide such as the standards for camera film.
- JIS – Japanese Industrial Standard – Created after WWII for Japanese standards.
- CEN – European Standards Organization

# Scaling vs. Dimensioning

## Drawings can be different scales, but dimensions are ALWAYS at FULL scale



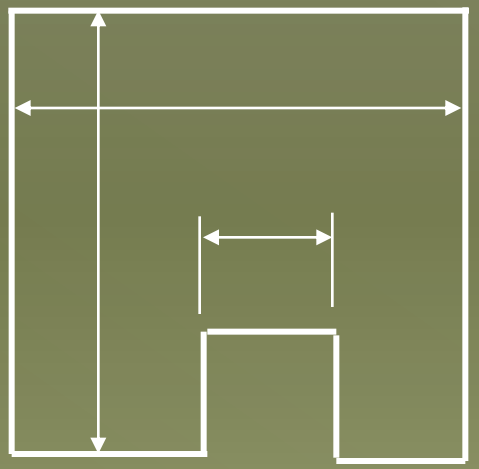
# Linear dimensions are comprised of four components:



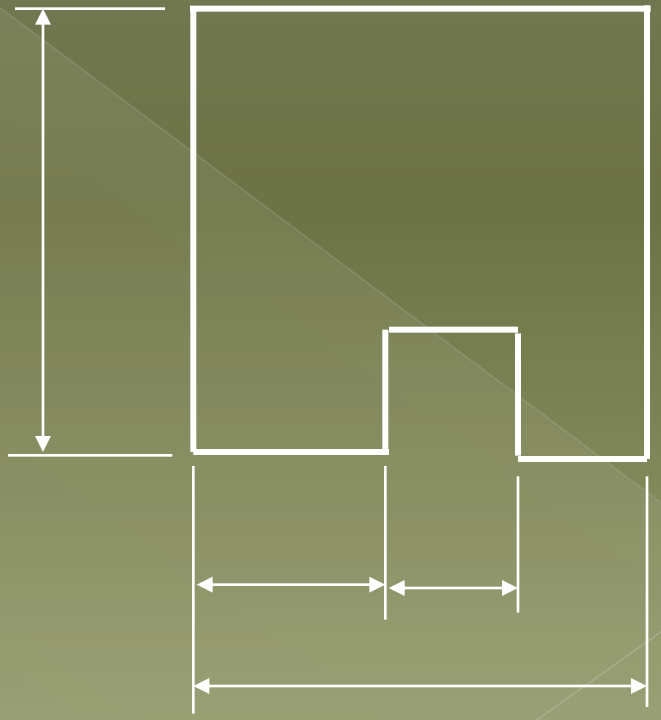
# General Guidelines: Clarity is the Goal

- Dimension Outside of View

Avoid



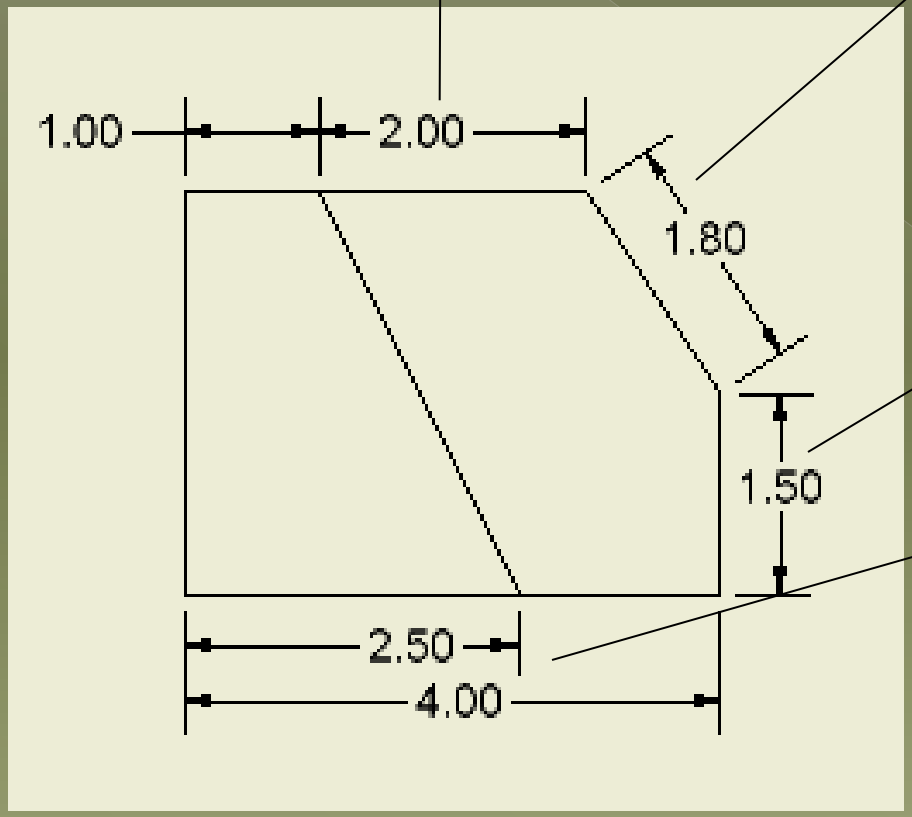
Good Practice



# Dimension Lines

Horizontal

Aligned to a slanted surfaced

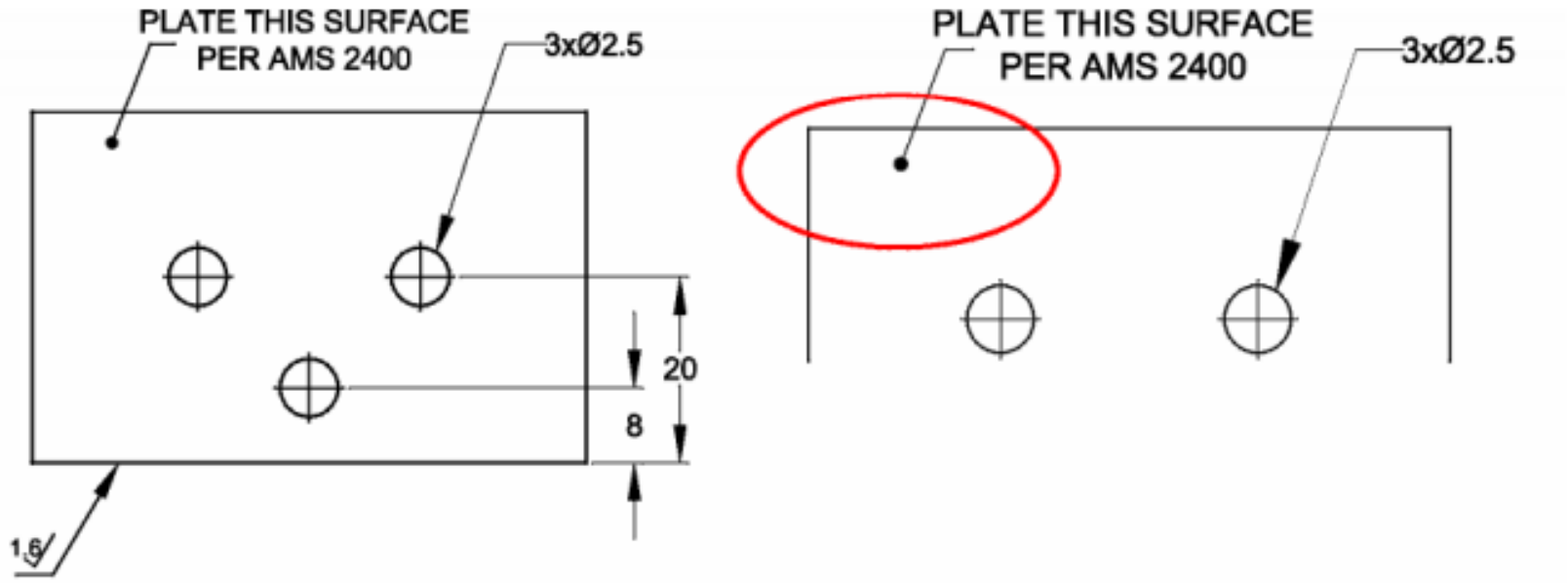



Vertical

When stacked, they are 10mm (.4”) from the view and 6mm(.25”) apart.

# Leader lines

Leader lines are used to direct a dimension, note, or symbol to the intended place on the drawing. Normally a leader terminates in an arrow head. However, where it is intended for a leader to refer to a surface by ending within the outline of that surface, the leader should terminate in a dot.

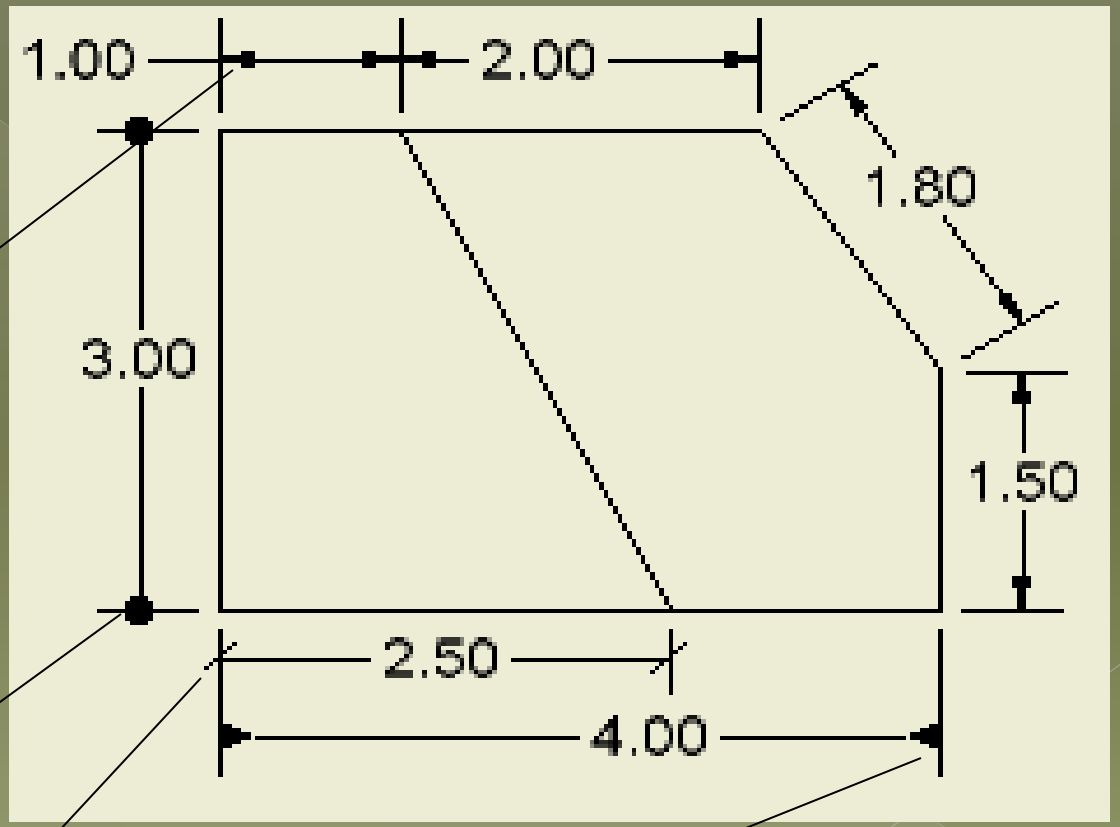


# Arrowheads

## (Dimension Line Terminator)

Arrowheads are typical dimension line terminators. There are other acceptable dimension line terminators.

Arrowheads point directly to the object that is being dimensioned or the extension lines at the end of the dimension. Arrowheads are made three times as long as they are wide.



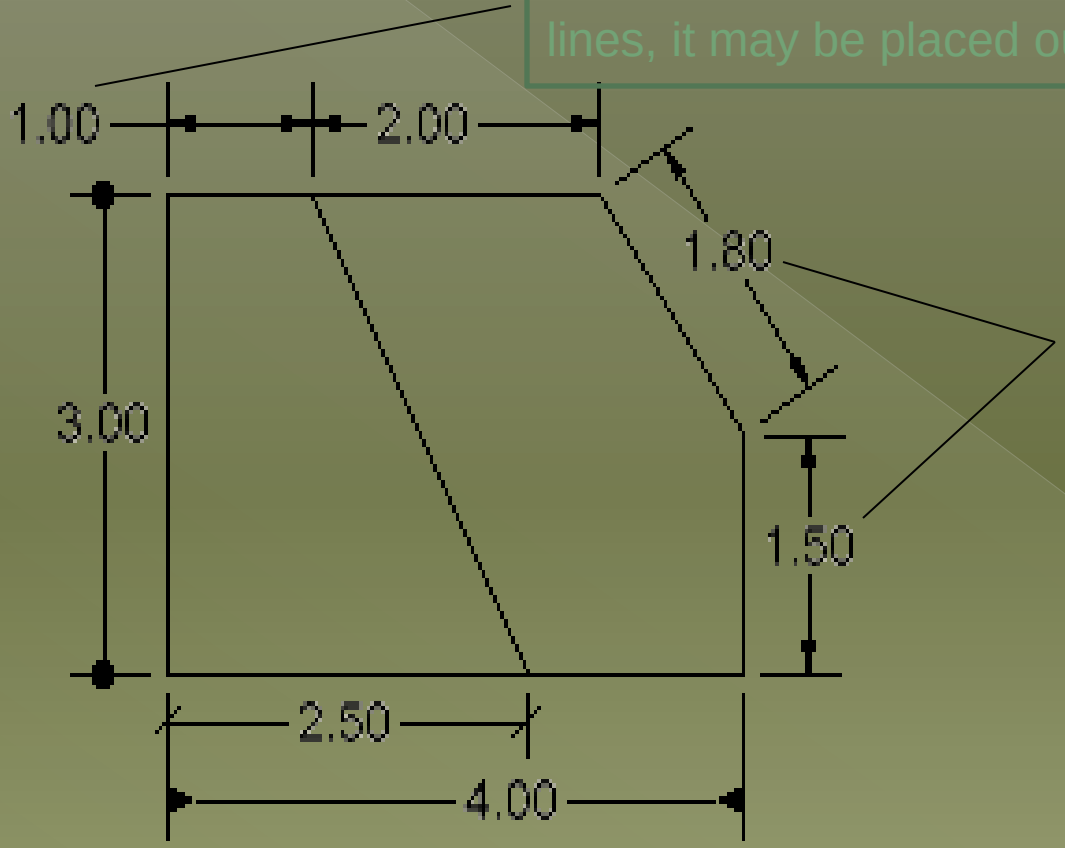

Dot

Oblique or architectural ticks used in architectural drawings

Datum

# Dimension Text

If the dimension text will not fit between the extension lines, it may be placed outside them.

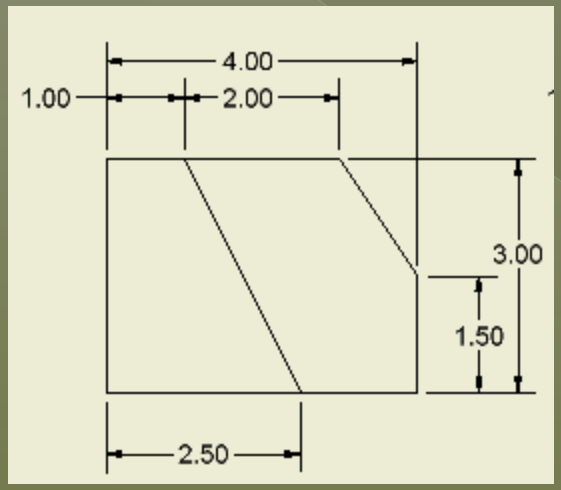


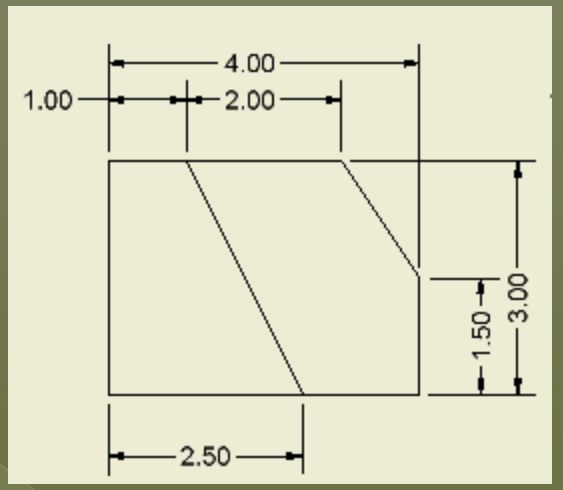

Dimension text is placed in the middle of the line both horizontally and vertically.

# Dimensioning Methods

- Dimensions are represented on a drawing using one of two systems, unidirectional or aligned.
- The *unidirectional* method means all dimensions are read in the same direction.
- The *aligned* method means the dimensions are read in alignment with the dimension lines or side of the part, some read horizontally and others read vertically.

# Dimension Text
# Unidirectional vs. Aligned




Unidirectional dimensions are placed so they can be read from the bottom of the drawing sheet. This method is commonly used in mechanical drafting.

Aligned dimensions are placed so the horizontal dimensions can be read from the bottom of the drawing sheet and the vertical dimensions can be read from the right side of the drawing sheet. This method is commonly used in architectural and structural drafting.

# Types of Dimensions

- There are two classifications of dimensions: size and location.

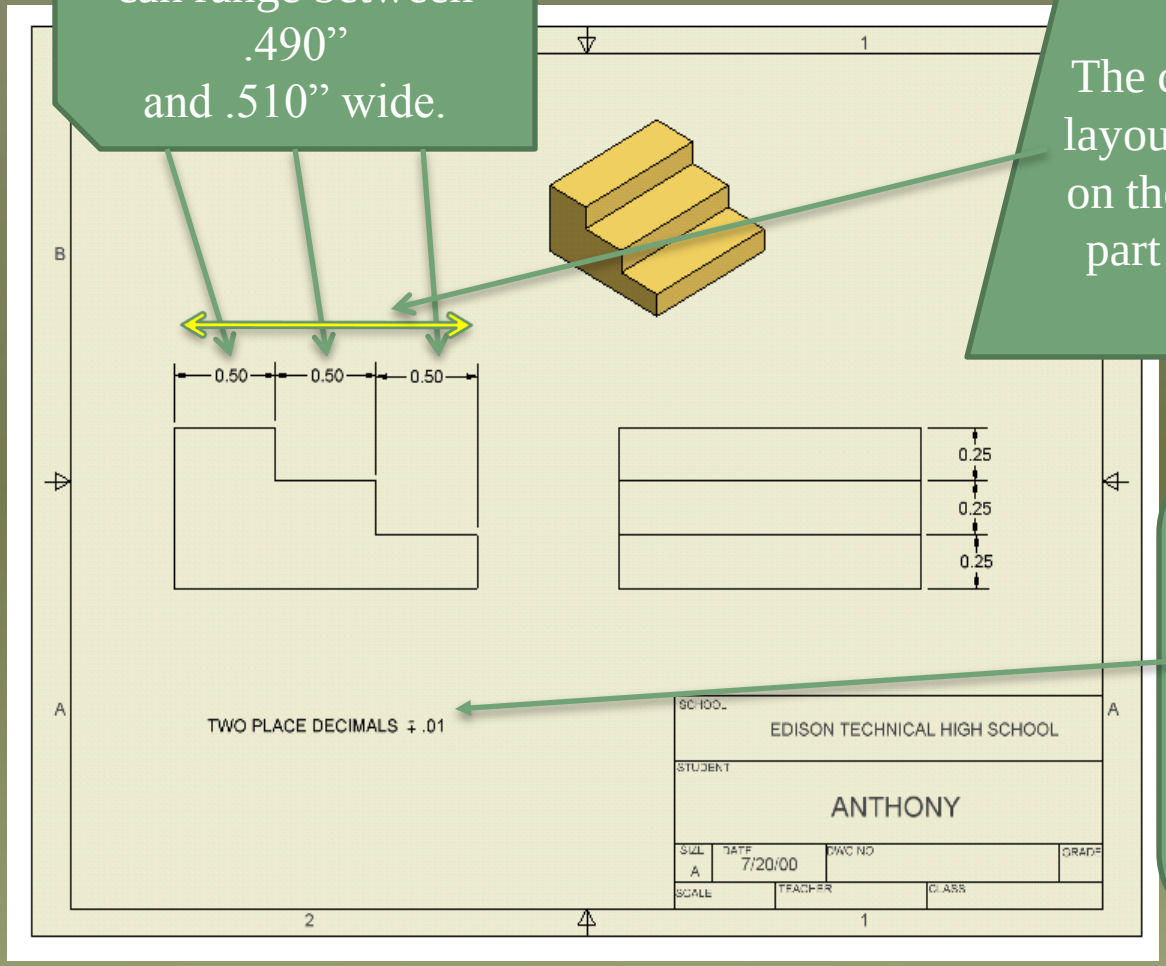
- *Size* dimensions are placed in direct relationship to a feature to identify the specific size.
- *Location* dimensions are used to identify the relationship of a feature to another feature within an object.

# Linear Dimensioning

- Dimensioning from feature to feature is known as *Chain Dimensioning.* It is commonly used and easy to lay out. It does have possible consequences in the manufacturing of a part. Tolerances can accumulate, making the end product larger or smaller than expected.

# Chain Dimensioning

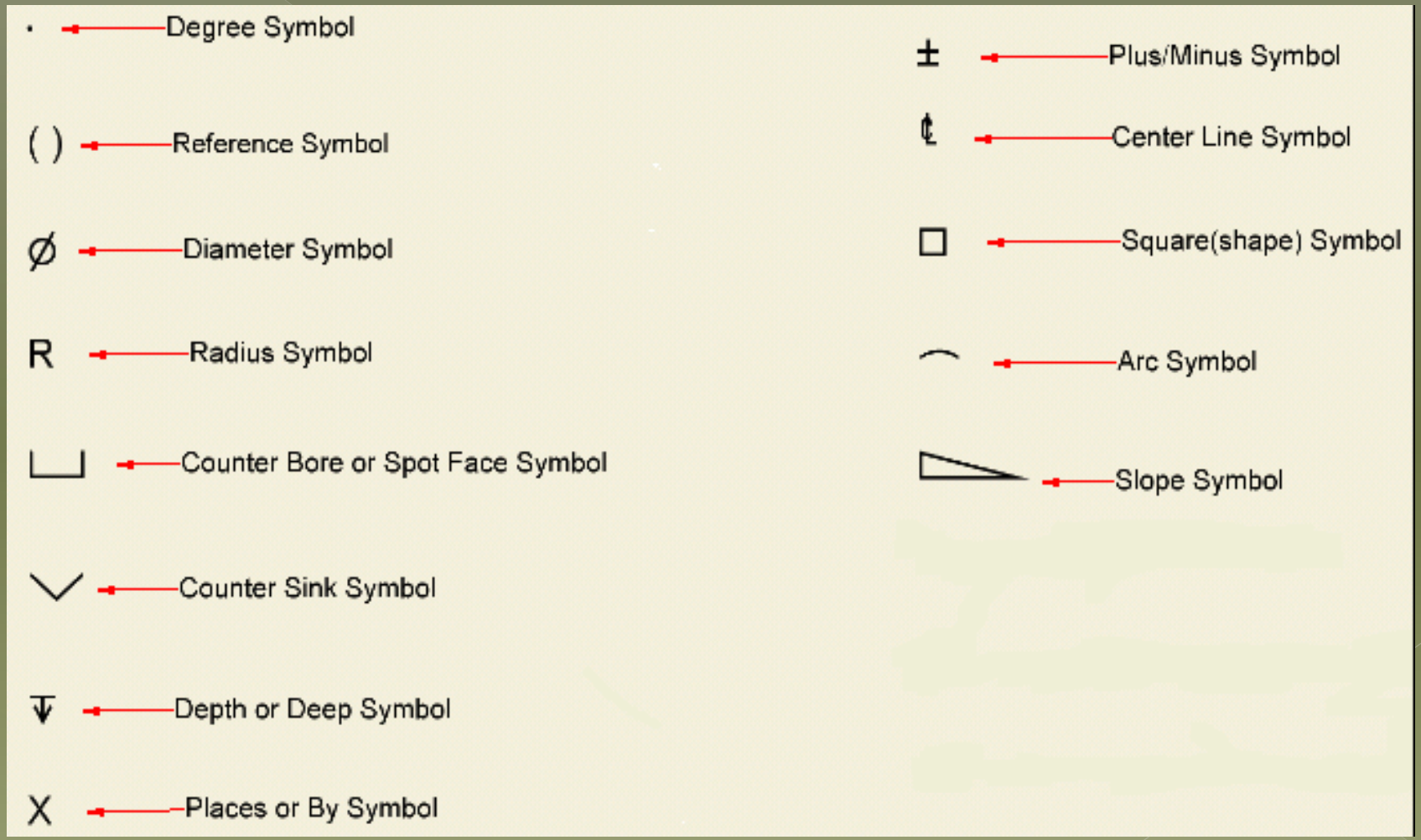
Each of these steps can range between .490” and .510” wide.




The chain dimensioning layout can have an effect on the final length of the part ranging from 1.47 to 1.53.

This is a general note. It indicates that all two place decimal dimensions have a tolerance of plus or minus .01 inch unless otherwise specified

# Dimensioning Symbols

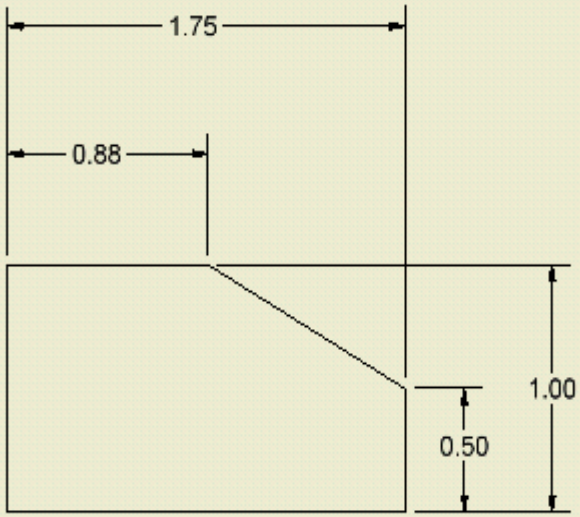


# Dimensioning Angles

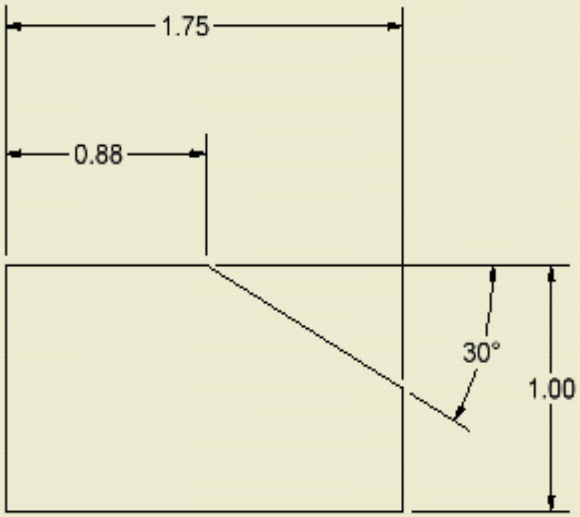
- Angled surface may be dimensioned using *coordinate method* to specify the two location distances of the angle.
- Angled surfaces may also be dimensioned using the *angular method* by specifying one location distance and the angle.

# Dimensioning Angles

Coordinate Method




Angular Method



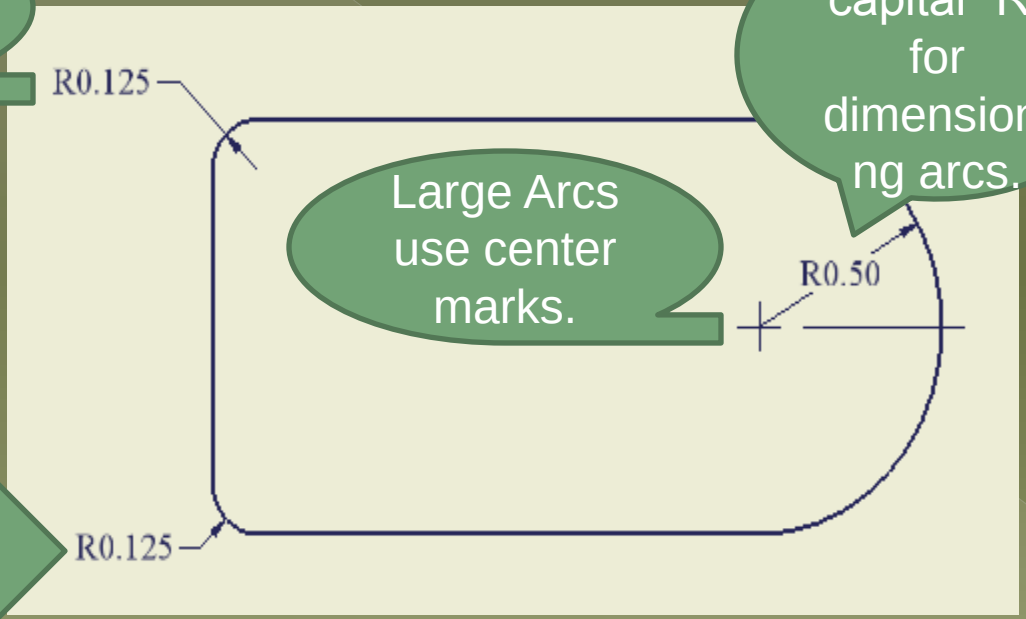
# Dimensioning Arcs and Circles

- Arcs and circles are dimensioned in views that show the arc or circle.
- Arcs are dimensioned with a leader to identify the radius; in some cases, a center mark is included.
- Circles should have a center mark and are dimensioned with a leader to identify the diameter.

# Dimensioning Curved Features and Arcs

The arrow can be inside for small arcs.

Use a capital “R” for dimensioning arcs.

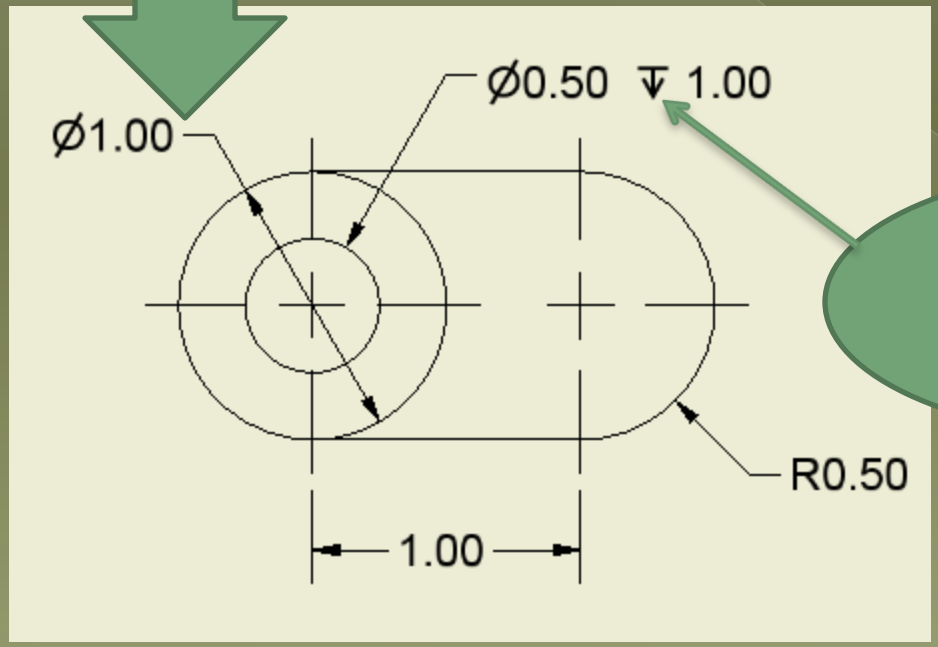



Large Arcs use center marks.

Small arcs do not need center marks. Arrow can be outside.

# Diameters

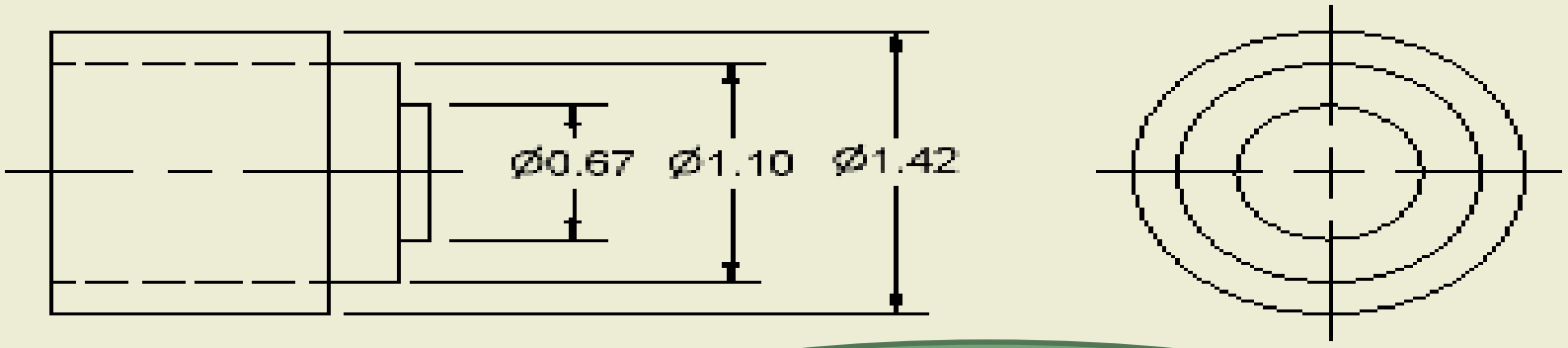
A full circular object should be dimensioned using its diameter. Holes should use hole notes.




This specification calls for a hole with a .5 diameter and 1.00 deep.

# Diameters

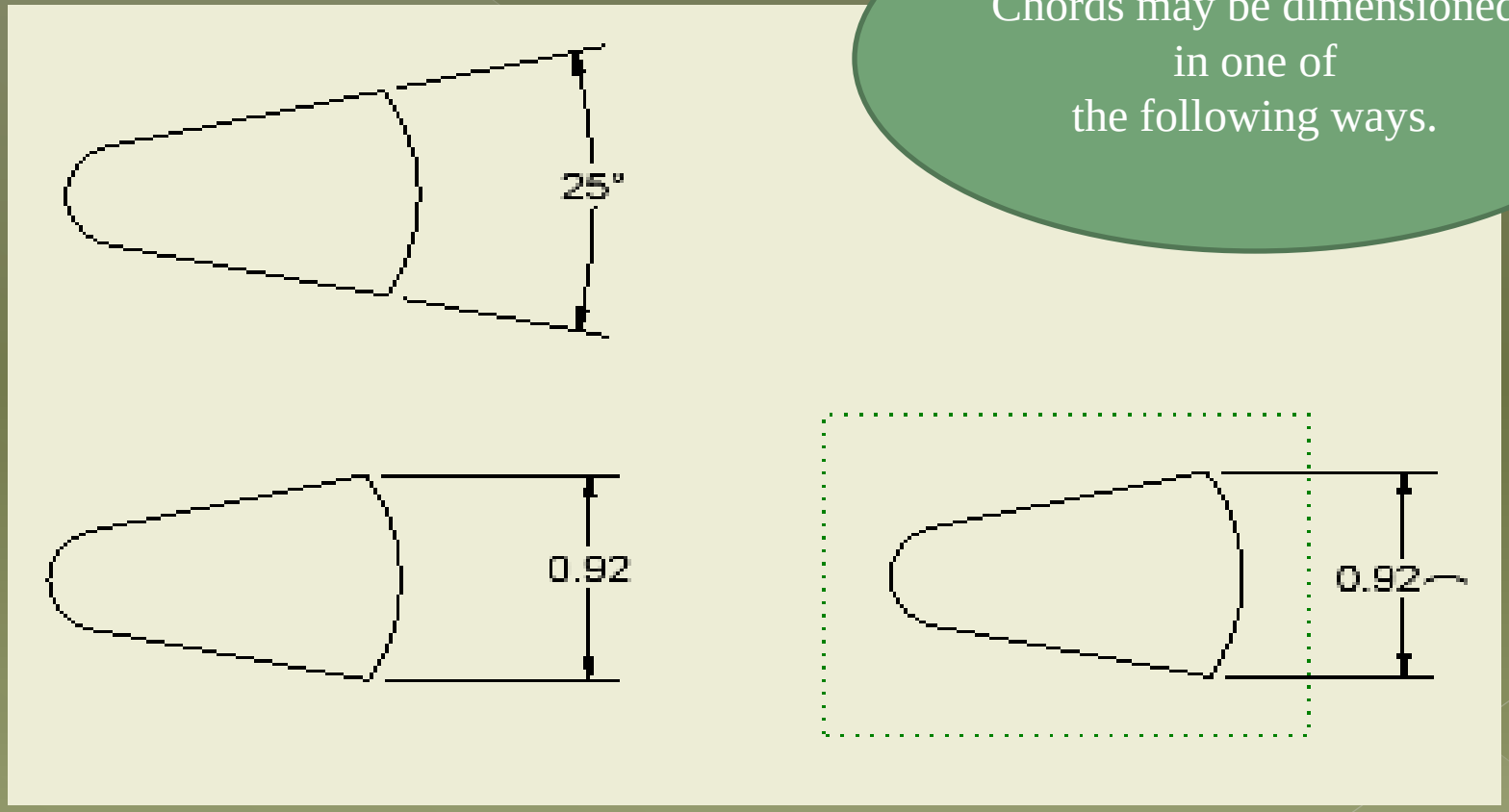
Cylindrical parts may show their diameters in this manner. Dimensioning on the right side view could be too crowded.




Note that the diameter symbol is used so it is not confused with a linear dimension.

# Chords

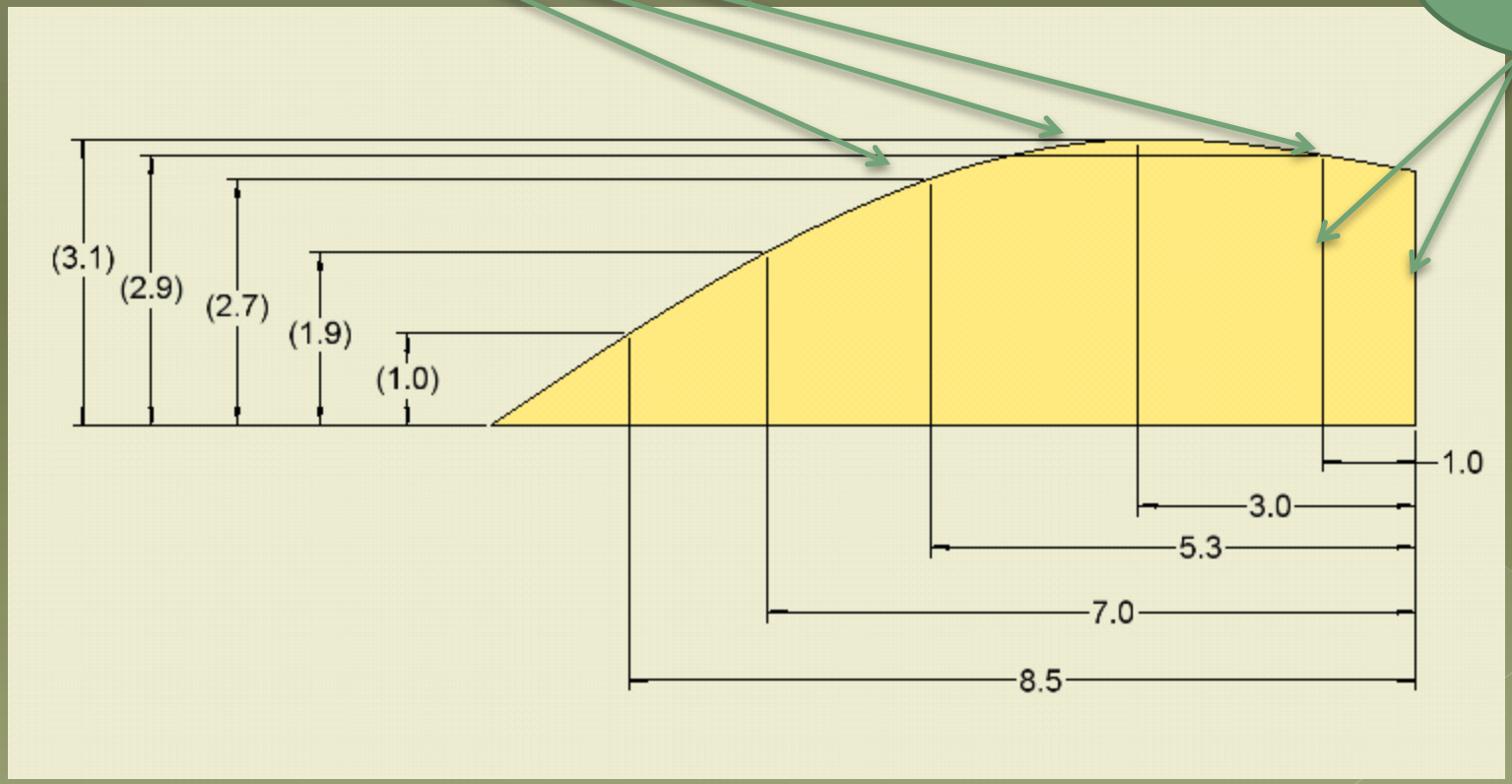
Chords may be dimensioned in one of the following ways.



# Dimensioning Curved Features

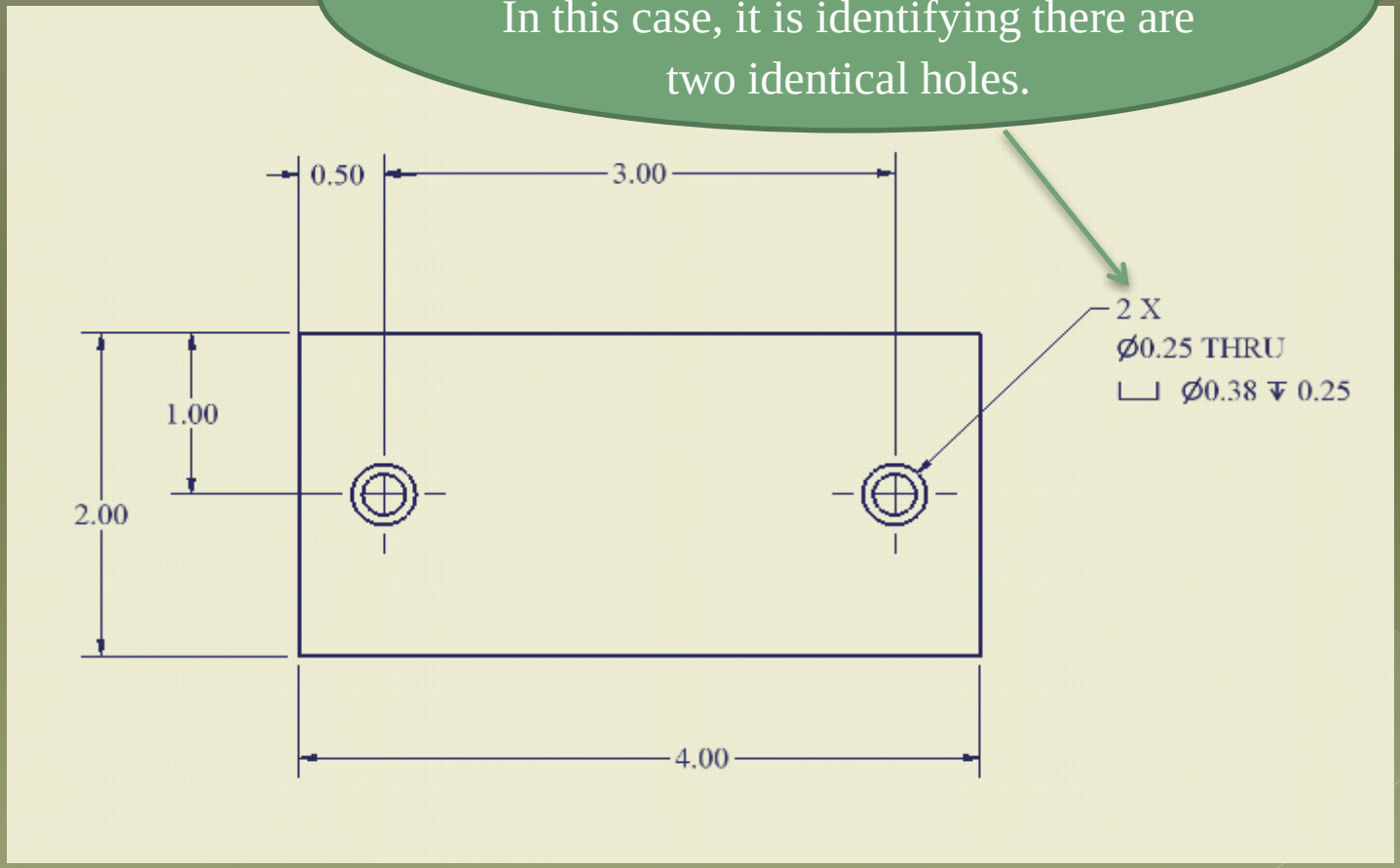
Points are placed along the contour and are dimensioned from the datum.

Datum



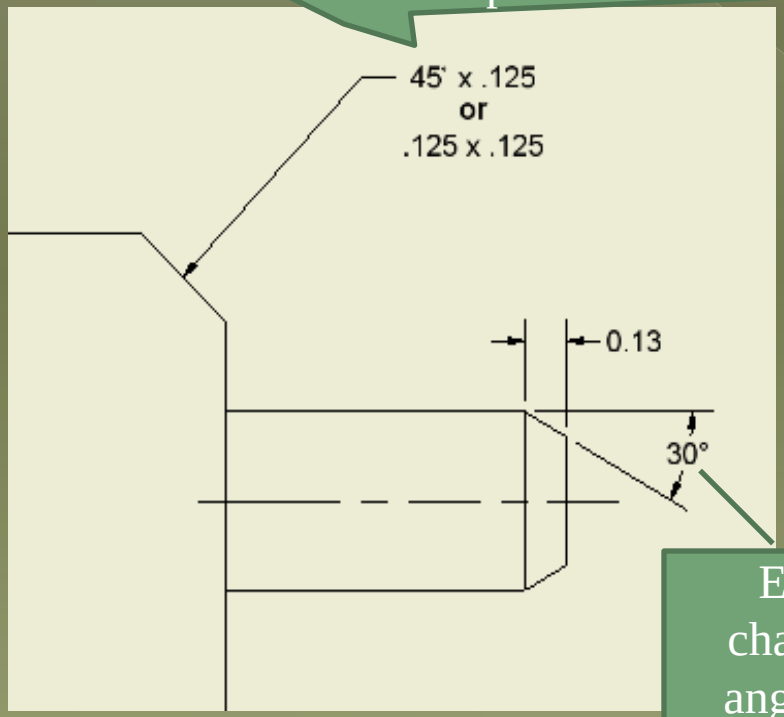
# Reference Dimensions

Designates more than one of the same feature.
In this case, it is identifying there are two identical holes.



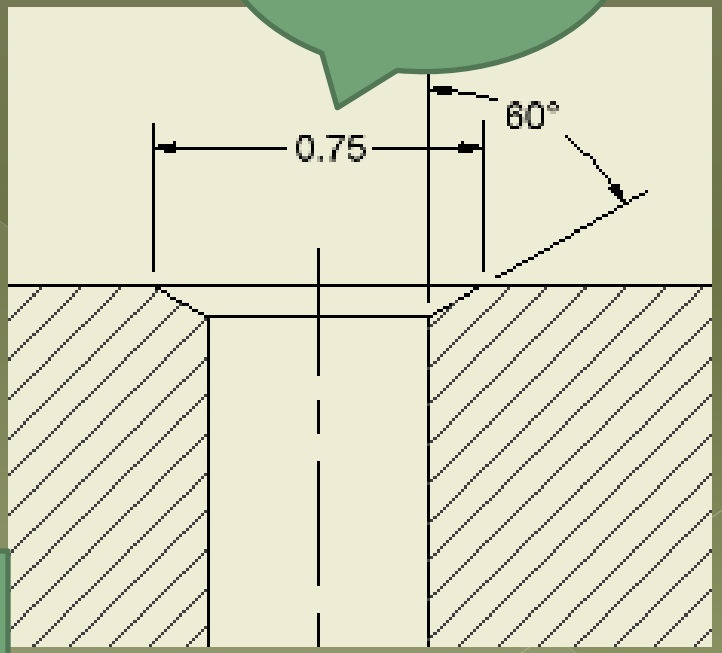
# Chamfers

External chamfer for 45 degree chamfers only.
There are two options.




External chamfer for angles other than 45 degrees.

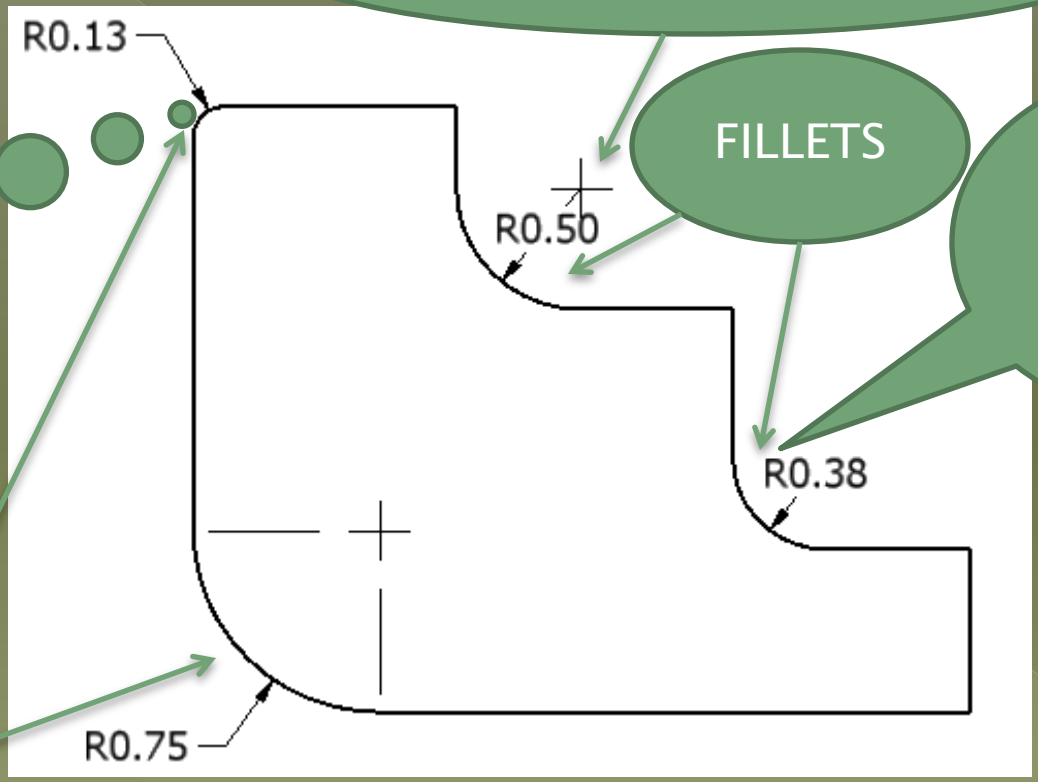
Internal chamfers.



# Fillets and Rounds

Large arcs use center marks

Small arcs do not need center marks. Arrow can be outside the arc.

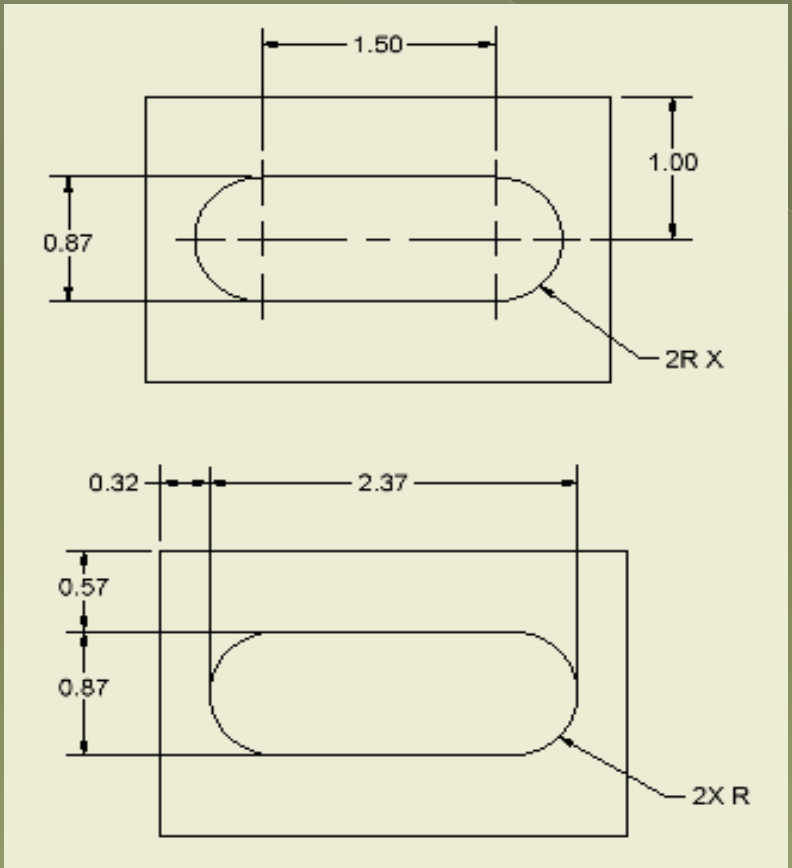



FILLETS

Use a capital "R" for dimensioning the arc.

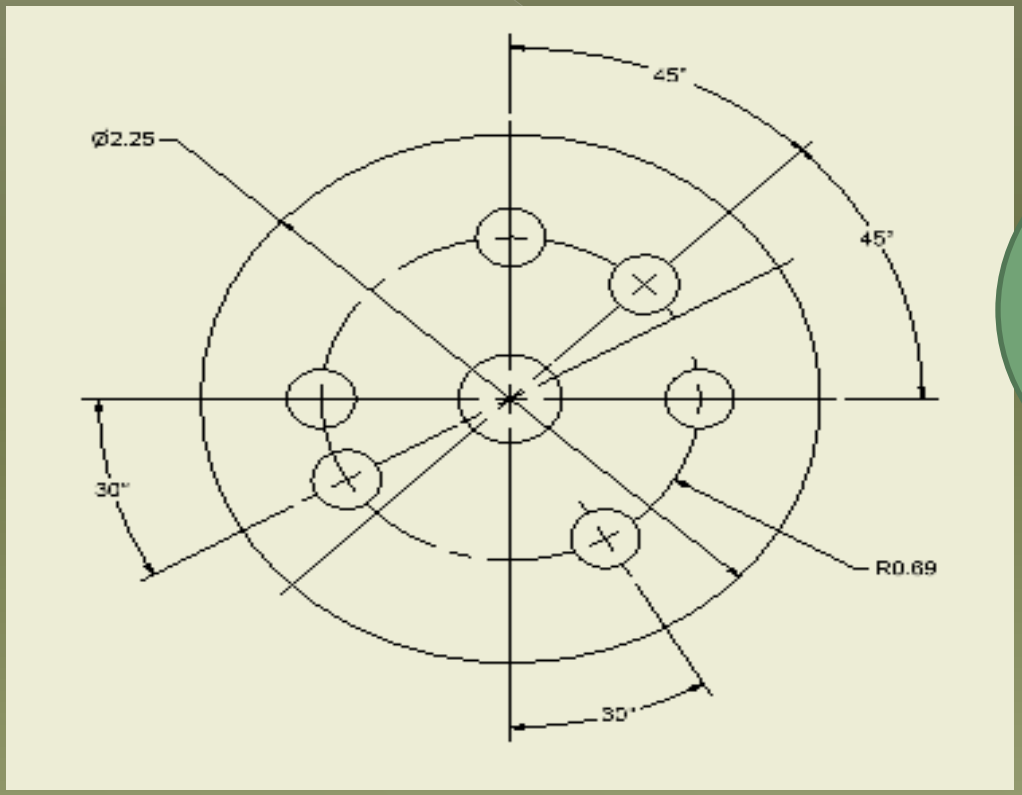
ROUNDS

# Slot Dimensioning



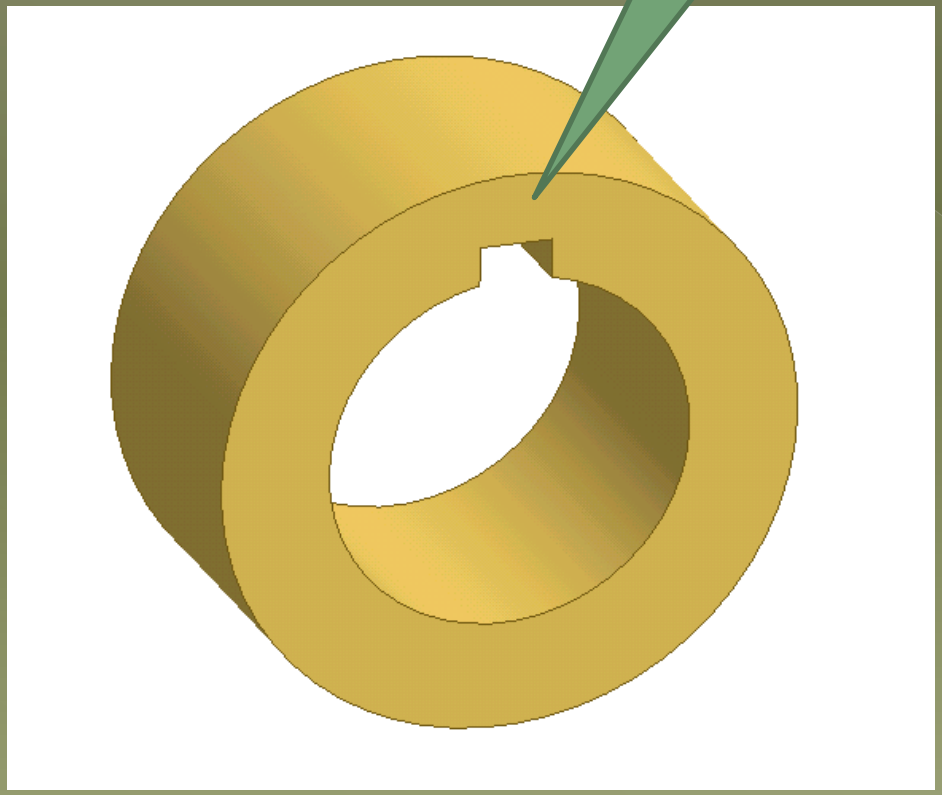

The two methods shown on the left are the acceptable methods for dimensioning slotted holes.

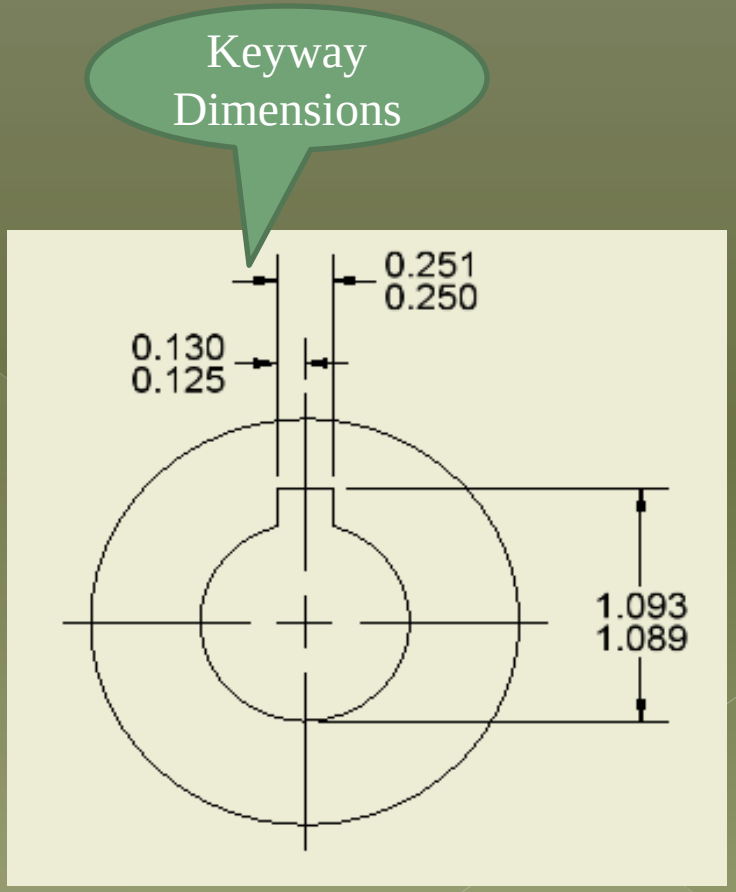
# Dimensioning Radial Patterns




Angles and radius values are used to locate the centers of radial patterned features, such as the holes on this plate.

# Keyway



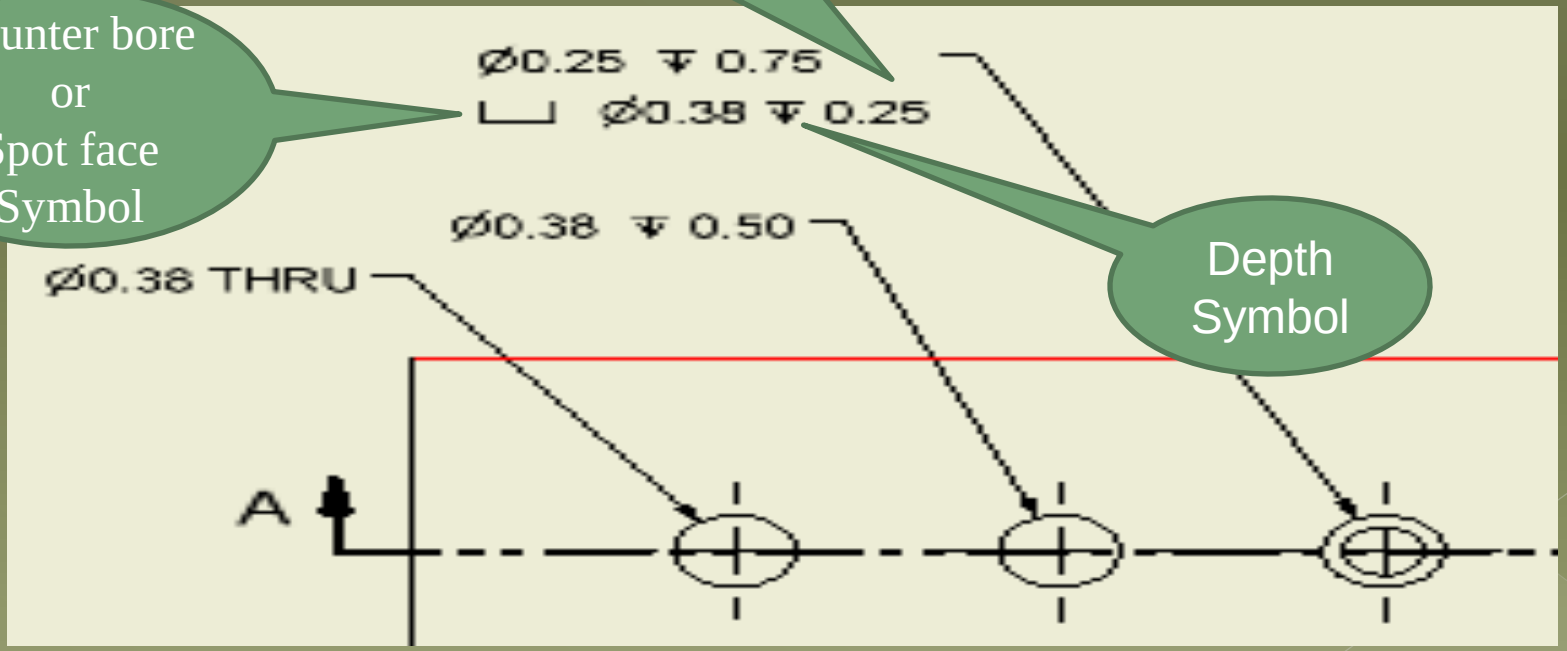



# Hole Dimensioning

The Hole Diameter is .25” and will be drilled .75” deep.
The Hole will be Counter bored to a .38”diameter and to a depth of .25”

Holes are specified with numbers and symbols

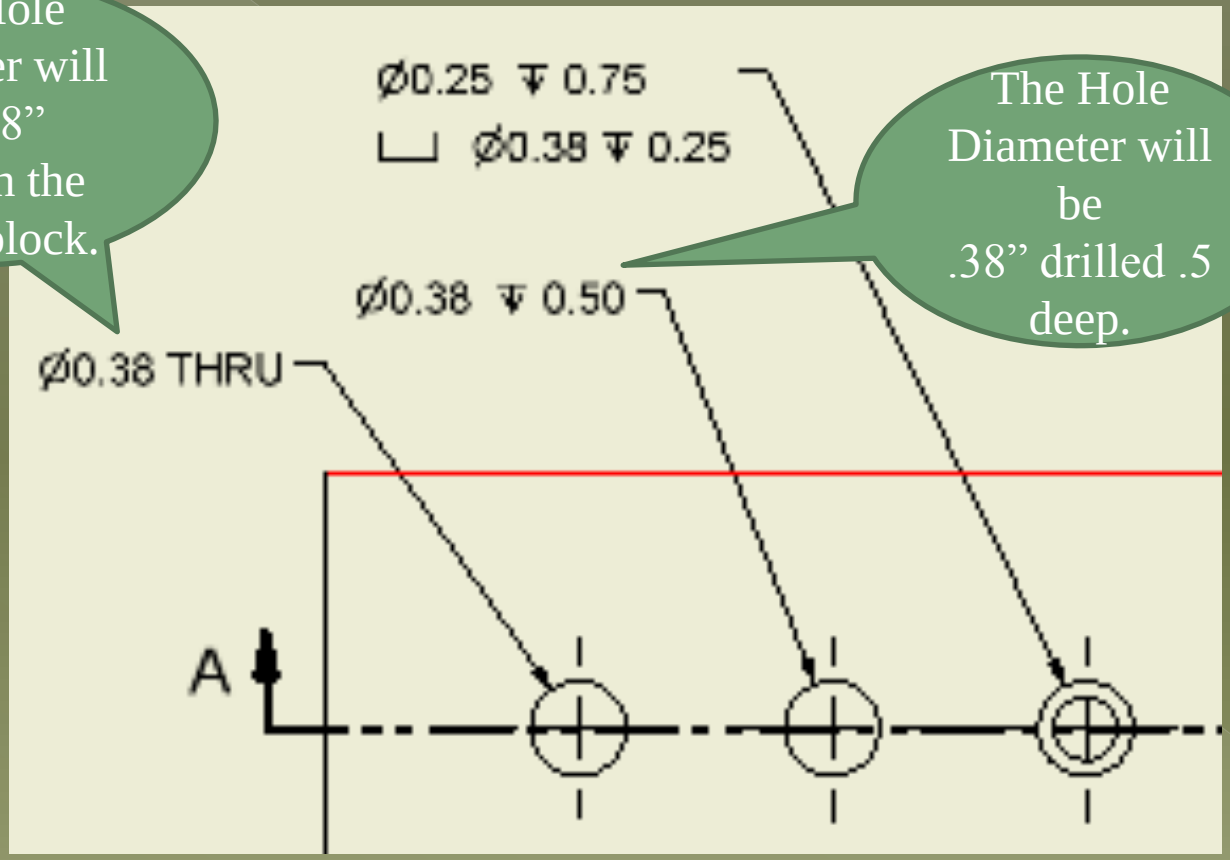
Counter bore or Spot face Symbol




Depth Symbol

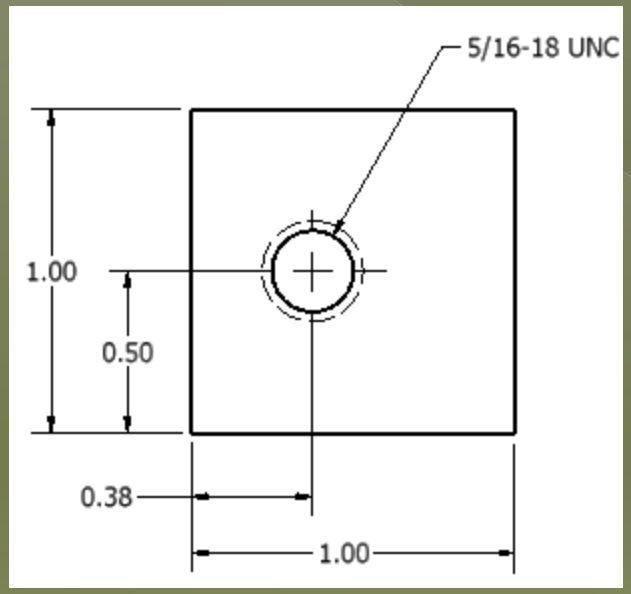
# Reading a Hole Note

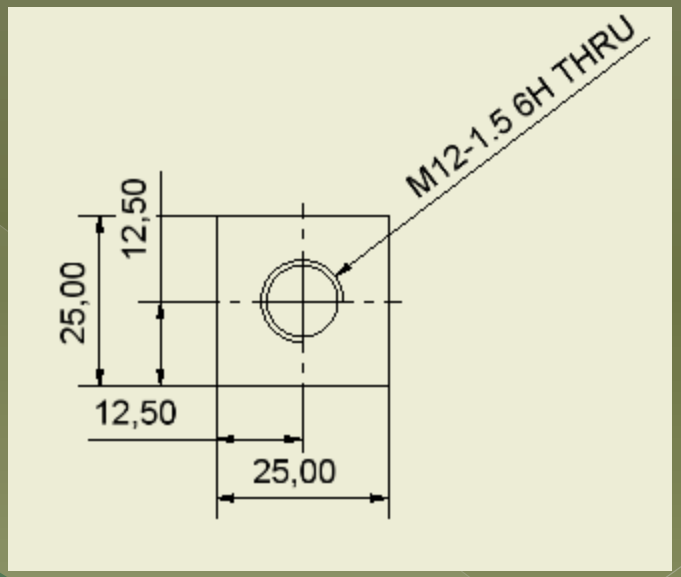
The Hole Diameter will be .38” through the whole block.




The Hole Diameter will be .38” drilled .5 deep.

# Reading Thread Notes



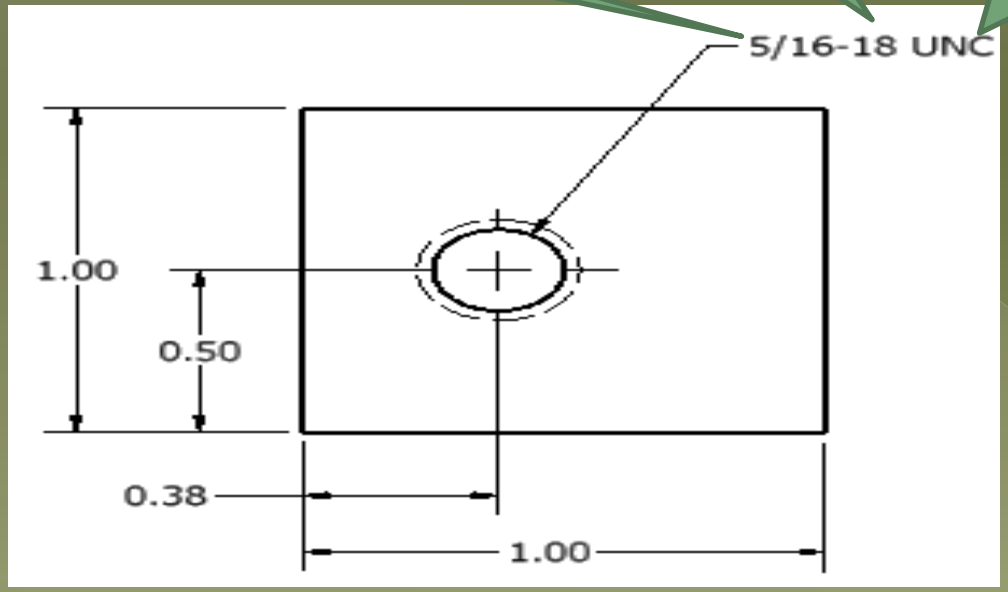

Threads are dimensioned with the use of local notes. We will discuss two methods: the ISO and the Unified National Thread method.

# Reading a Unified National Thread Note

Major Diameter

Threads per Inch

Identifies coarse or fine thread. In this case, C for coarse. F is for fine.



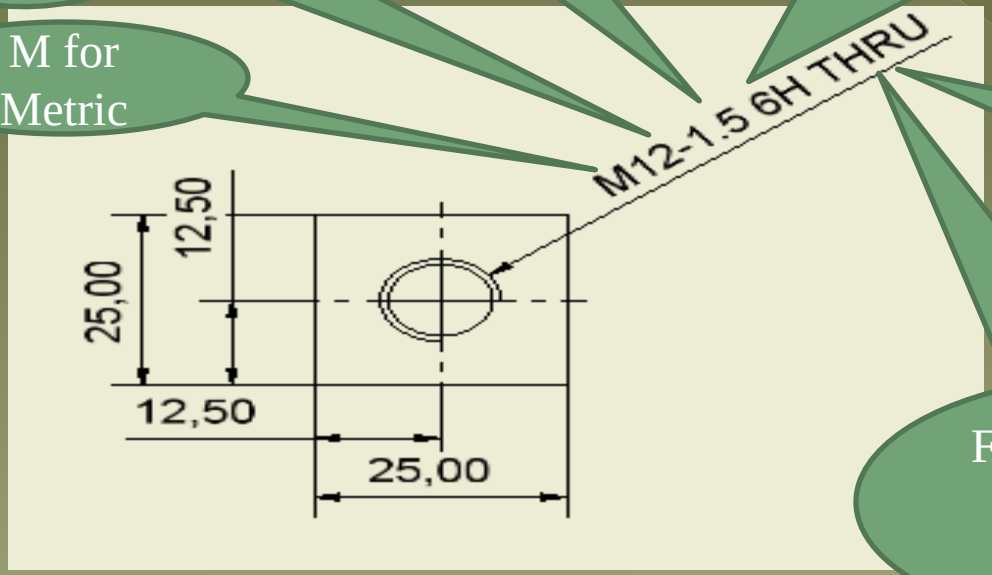
# Reading a ISO Thread Notes

This number can be 3,4,5,6,7,8,9. It is the grade of tolerance in the threads from fine to coarse. The H is for allowance: G would be a tight allowance and H is no allowance.

Nominal Diameter In Millimeters

Pitch of the threads

M for Metric




Prior to THRU, you may have an LH for left hand thread.

Finally THRU or a depth may be specified.