

# UNIT D: Dimensioning and Conventional Tolerancing

## Competency: D404.00

*Demonstrate intermediate dimensioning and conventional tolerancing techniques.*

## Objective: D404.01

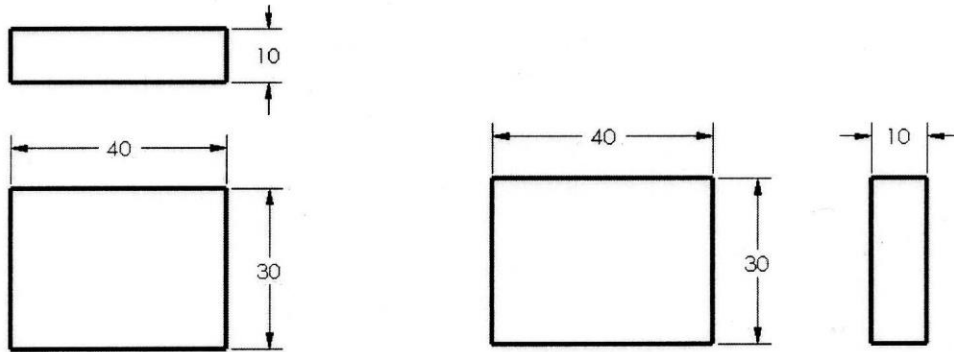
*Explain intermediate dimensioning techniques.*

**Introduction:** The purpose of this unit is to cover intermediate dimensioning techniques and the area of tolerance dimensioning. When students begin dimensioning parts, especially parts that interact within an assembly, it is critical that they understand the differences between clearance and interference fits. Modern industry relies on interchangeable manufacturing so that parts can be manufactured in widely separate localities and then brought together for assembly without further machining. After this unit, students should be able to define terminology related to tolerance dimensioning, identify different types of fits given two dimensioned parts, and correctly apply limit dimensions to a part when given the type of fit.

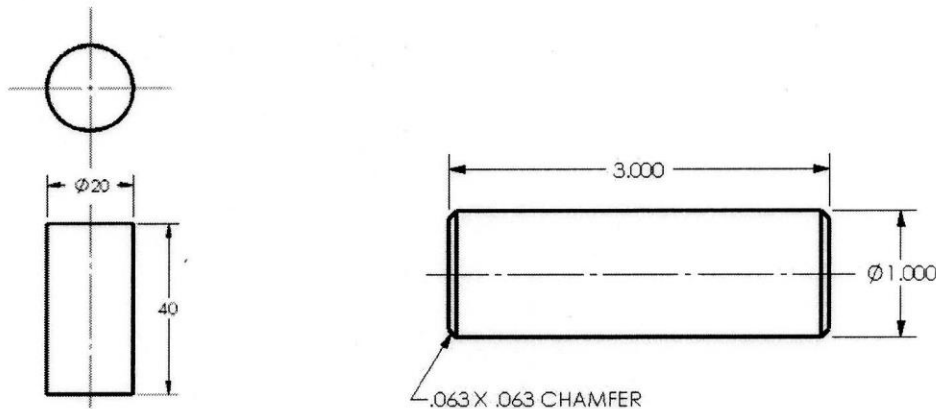
### Explain the following:

#### A. Dimensioning prisms, cylinders, cones, and spheres. R1(305-308), R2(237-241), R3(200-204).

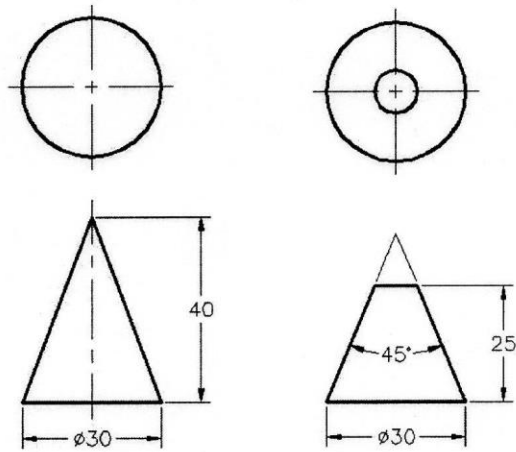
##### 1. Rectangular prisms



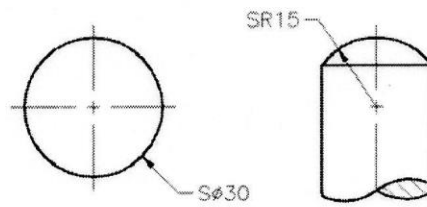
##### 2. Cylinders – The diameters of cylinders should be dimensioned in the rectangular view. Unless a hole is present in the cylinder, only one view is necessary.



##### 3. Cones

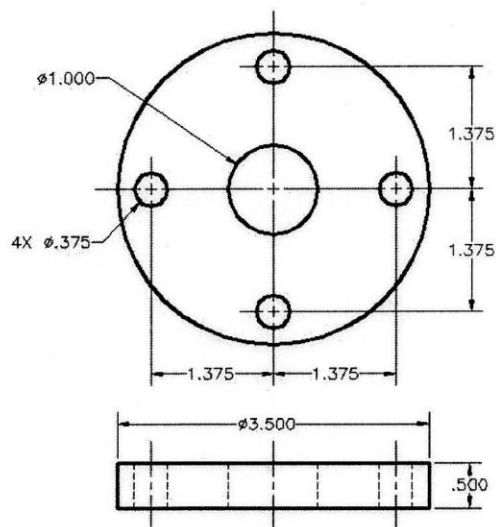


4. Spheres



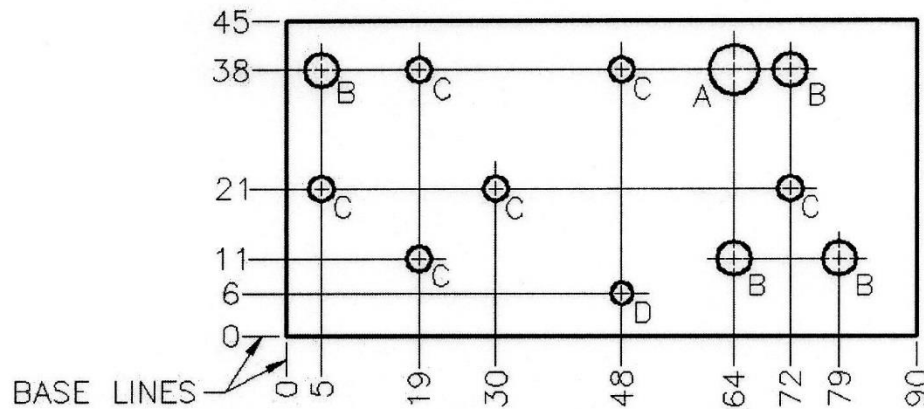
**B. Rectangular Coordinate Dimensioning** – This type of dimensioning system is specifically used when computer-controlled production machines are used to manufacture parts. Notice in the examples that all dimensions are referenced from an origin or 0,0 location. The designer should consult with personnel in manufacturing to ensure that the origin is located in an appropriate position. *R1(321-322), R2(246).*

1. Coordinate Dimensioning with Dimension Lines

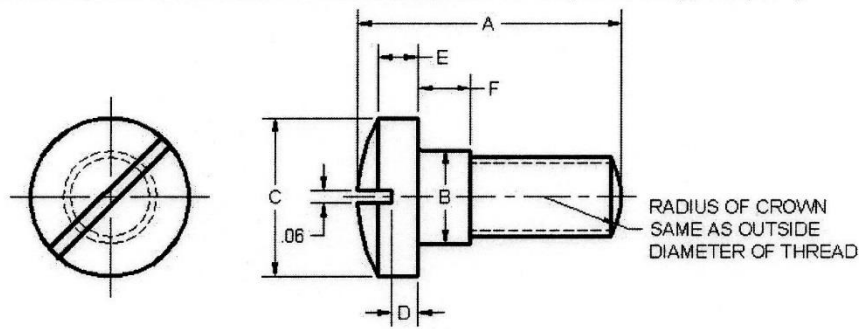


2. Coordinate Dimensioning Without Dimension Lines.

SIZE SYMBOL	A	B	C	D
HOLE $\phi$	7	4.8	3.6	3.1



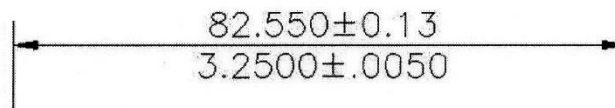
C. **Tabular Dimensioning.** Tabular dimensioning is used when a series of parts consists of the same features or geometry but vary in dimension. Letters are used in place of dimension values, and the values are then placed in a table. Most standard parts are dimensioned this way in catalogs, the machinery handbook, and in the back of most textbooks. *R1(320-321), R2(471).*

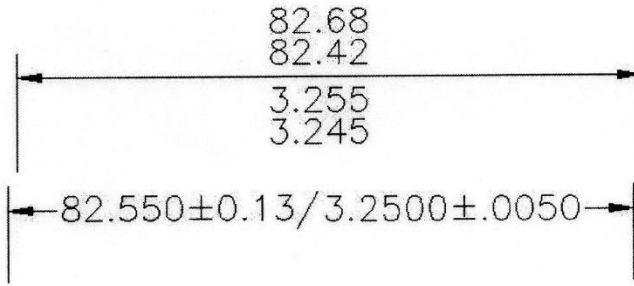


DETAIL	A	B	C	D	E	F	UNC THREAD
1	1.000	.380	.625	.091	.188	.250	.312 - 18
2	1.250	.440	.750	.125	.188	.250	.375 - 16
3	1.500	.500	.875	.125	.250	.375	.437 - 14

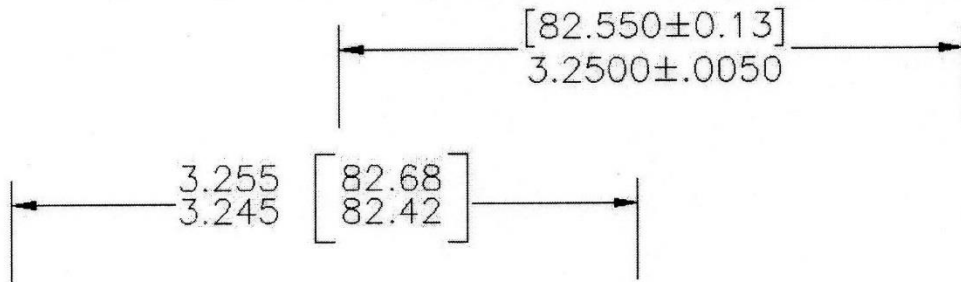
D. **Dual Dimensioning** – used to show both metric and decimal inch dimensioning on the same drawing. *R1(299-300), R2(234), R3(219).*

1. **Position Method** – millimeter value is placed above (or below) the inch value or separated by a dash.





2. *Bracket Method* – millimeter value is enclosed in square brackets. A note should be placed on the drawing such as: **DIMENSIONS IN [ ] ARE MILLIMETERS.**



## UNIT D: Dimensioning and Conventional Tolerancing

### Competency: D404.00

*Demonstrate intermediate dimensioning and conventional tolerancing techniques.*

### Objective: D404.02

*Explain procedures for determining tolerance dimensions.*

#### Explain the following:

- A. Define the following terms: *R1(330-339)*, *R2(246-249)*, *R3(213-217)*.
1. **Interchangeable** - Parts that are made to easily fit mating parts without additional machining at the time of assembly.
  2. **Actual size** - The actual size is the measured size.
  3. **Basic size** - The basic size is the size to which allowances and tolerances are added to get the limits of size.
  4. **Nominal size** - The nominal size is a designation used for general identification. Typically this is a fraction when working in inches.
  5. **Design size** - The design size is the size from which the limits of size are derived by the application of tolerances.
  6. **Tolerance** - The total amount a single dimension can vary. The difference between the upper and lower limits.
  7. **Allowance** - Allowance is the minimum clearance or maximum interference intended between the maximum material condition (MMC) of mating parts. Example: Smallest hole with the largest shaft. The tightest permissible fit.
  8. **Limit dimensions** - A tolerancing method showing the maximum and minimum size values. The maximum dimension is placed above the minimum dimension. When expressed in a single line, the lower limit precedes the upper limit.
  9. **Unilateral tolerance** - A tolerance that allows variation in one direction.
  10. **Bilateral tolerance** - A tolerance that allows variation in both directions from the design size.
  11. **Clearance fit** - A fit between mating parts having limits of size so prescribed that a clearance always results in assembly.
  12. **Interference fit** - A fit between mating parts having limits of size so prescribed that interference always results in assembly.
  13. **Transition fit** - A fit between mating parts having limits of size so prescribed as to partially or wholly overlap, so that either a clearance or interference may result in assembly.
- B. Tolerance dimensioning.
1. Expressing tolerances on a drawing. Tolerance dimensions can be expressed in the following ways:
    - a. **Limit dimensions** – This is the preferred method for expressing tolerances. Show both the upper and lower limits. The high or upper limit is placed above the low or lower limit. In a single-line note the lower limit precedes the upper limit and they are separated by a dash.

1.5016	1.5000-1.5016
1.5000	
<i>Limit Dimension</i>	<i>Single-Line Limit Dimension</i>

- b. **Plus-or-minus dimensions** – For this method, the basic dimension is followed by a plus-or-minus expression.

$$.5000^{+.0050}_{-.0000}$$

*Unilateral*

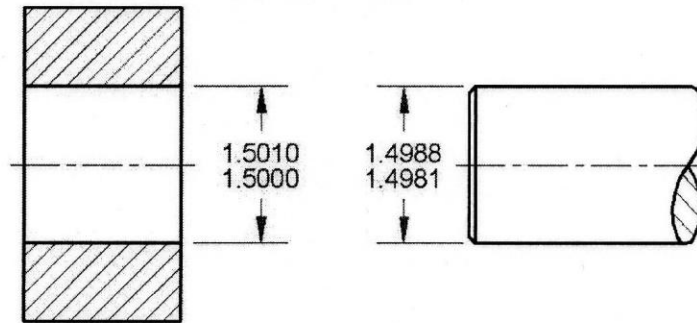
$$.5000 \pm .0050$$

*Bilateral-Equal*

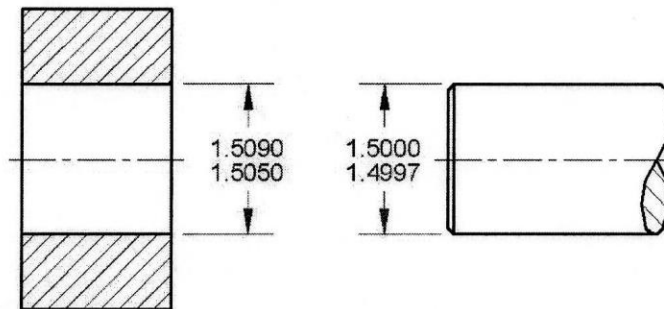
$$.5000^{+.0050}_{-.0025}$$

*Bilateral-Unequal*

2. Describe the following types of systems as they relate to tolerance dimensioning:
- a. **Basic hole system** - In a basic hole system, the design size of the hole is the basic size and the allowance is applied to the shaft. The basic hole system can be used to keep the costs of tooling down since standard tools such as a ream or broach can be used for machining. Notice that the lower limit of the hole is the basic size 1.5000.



- b. **Basic shaft system** - In a basic shaft system, the design size of the shaft is the basic size and the allowance is applied to the hole. The basic shaft system should only be used when a standard size shaft is needed. Notice that the upper limit of the shaft is the basic size 1.5000.

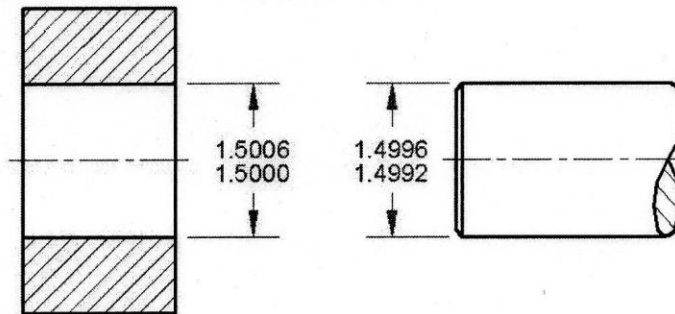


3. Describe the differences between the following types of fits: Be able to find/specify the following types of fits on a standard table.
- Running and Sliding Fits** - These fits provide a similar running performance, with suitable lubrication allowance, throughout the range of sizes.
  - Locational Clearance Fits** - These fits are intended for parts that are normally stationary but can be freely assembled or disassembled.
  - Locational Transition Fits** - These fits are intended where accuracy of location is important, but either a small amount of clearance or interference is permissible.
  - Locational Interference Fits** - These fits are intended where accuracy of location is of prime importance, and for parts requiring rigidity and alignment with no special requirement for bore pressure.
  - Force and Shrink Fits** - These types of interference fits are normally characterized by maintenance of constant bore pressures throughout the range of sizes.

C. Looking up specified fits on tables.

1. **Looking up fits in inches.** Below is a portion of a table listing *American National Standard Running and Sliding Fits*. One of the first things to notice about the table is that all of the values are in **thousandths of an inch** (so a value of +1 is really + .001).

- a. Determine the type of fit necessary (eg. close sliding fit, sliding fit, precision running fit, etc.). Find the fit at the top of one of the columns labeled as RC1, RC2, etc.
- b. Find the row on the left that contains the range for the nominal size of the shaft or machined hole.
- c. Find where the column and the row intersect to find the correct values.
- d. Example: Sliding fit with a system that has a nominal size of 1.500.



### Running and Sliding Fits – American National Standard

- RC 1 *Close Sliding Fits* are intended for the accurate location of parts which must assemble without permissible play.
- RC 2 *Sliding Fits* are intended for accurate location, but with greater maximum clearance than class RC1. Parts made to this fit move and turn easily but are not intended to run freely, and in the larger sizes may seize with small temperature changes.
- RC 3 *Precision Running Fits* are about the closest fits which can be expected to run freely and are intended for precision work at slow speeds and light journal pressures, but they are not suitable where appreciable temperature differences are likely to be encountered.
- RC 4 *Close Running Fits* are intended chiefly for running fits on accurate machinery with moderate surface speeds and journal pressures, where accurate location and minimum play are desired.

Basic hole system. **Limits are in thousandths of an inch.**

Limits for hole and shaft are applied algebraically to the basic size to obtain the limits of size for the parts.

Data in **boldface** are in accordance with ABC agreements.

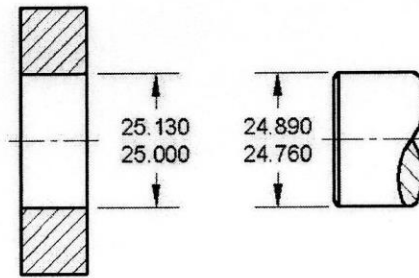
Symbols H5, g5, etc., are hole and shaft designations used in ABC System.

Nominal Size Range, Inches Over To	Class RC 1			Class RC 2		
	Limits of Clearance	Standard Limits		Limits of Clearance	Standard Limits	
		Hole H5	Shaft g4		Hole H6	Shaft g5
0-0.12	0.1	+0.2	-0.1	0.1	+0.25	-0.1
	0.45	-0	-0.25	0.55	-0	-0.3
0.12-0.24	0.15	+0.2	-0.15	0.15	+0.3	-0.15
	0.5	-0	-0.3	0.65	-0	-0.35
0.24-0.40	0.2	+0.25	-0.2	0.2	+0.4	-0.2
	0.6	-0	-0.35	0.85	-0	-0.45
0.40-0.71	0.25	+0.3	-0.25	0.25	+0.4	-0.25
	0.75	-0	-0.45	0.95	-0	-0.55
0.71-1.19	0.3	+0.4	-0.3	0.3	+0.5	-0.3
	0.95	-0	-0.55	1.2	-0	-0.7
1.19-1.97	0.4	+0.4	-0.4	0.4	+0.6	-0.4
	1.1	-0	-0.7	1.4	-0	-0.8

\* From ANSI B4.1 – 1967 (R1994). For larger diameters, see the standard.

2. **Looking up fits in millimeters.** Below is a portion of a table listing *American National Standard Preferred Metric Hole Basis Clearance Fits*.
  - a. Determine the type of fit necessary (eg. loose running, free running, etc.). Find the fit at the top of one of the columns.
  - b. Find the row on the left that contains the nominal size of the shaft or machined hole.
  - c. Find where the column and the row intersect to find the correct values.
  - d. Example: Loose Running fit with a system that has a nominal size of 25.





**Preferred Metric Hole Basis Clearance Fits – American National Standard**

Dimensions are in millimeters.

Basic Size	Loose Running			Free Running		
	Hole H11	Shaft c11	Fit	Hole H9	Shaft d9	Fit
1 Max	1.060	0.940	0.180	1.025	0.980	0.070
1 Min	1.060	0.880	0.060	1.000	0.955	0.020
20 Max	20.130	19.890	0.370	20.052	19.935	0.169
20 Min	20.000	19.760	0.110	20.000	19.883	0.065
25 Max	25.130	24.890	0.370	25.052	24.935	0.169
25 Min	25.000	24.760	0.110	25.000	24.833	0.065

From ANSI B4.2 – 1978 (R1984). For larger diameters, see the standard.

Appendix – Reading Dimensions

- .1 one tenth of an inch
- .01 one hundredth of an inch
- .001 one thousandth of an inch
- .0001 one ten-thousandth of an inch
- .00001 one millionth of an inch