UNIT D: Dimensioning and Conventional Tolerancing

Competency: D404.00

Demonstrate intermediate dimensioning and conventional tolerancing techniques.

Objective: D404.01

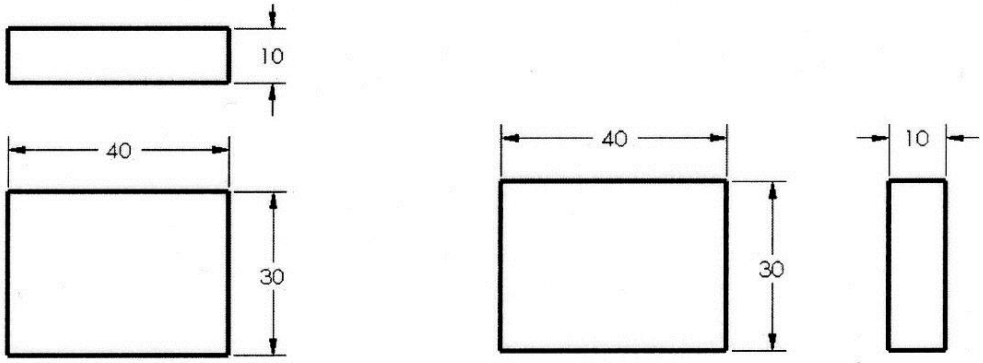
Explain intermediate dimensioning techniques.

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.

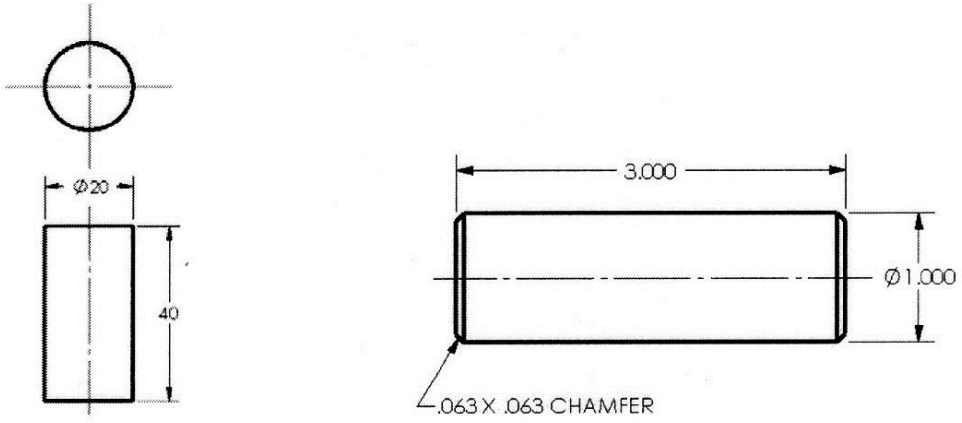
Explain the following:

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

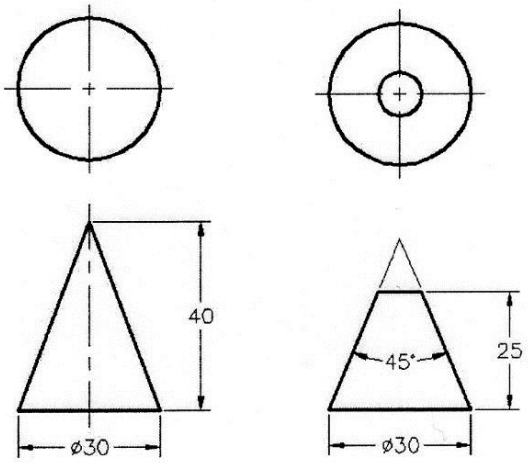
1. Rectangular prisms



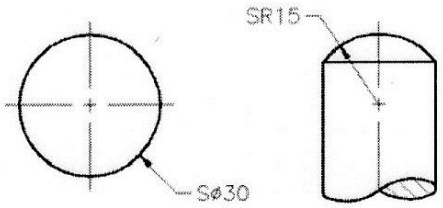
2. Cylinders – The diameters of cylinders should be dimensioned in the rectangular view. Unless a hole is present in the cylinder, only one view is necessary.



3. Cones

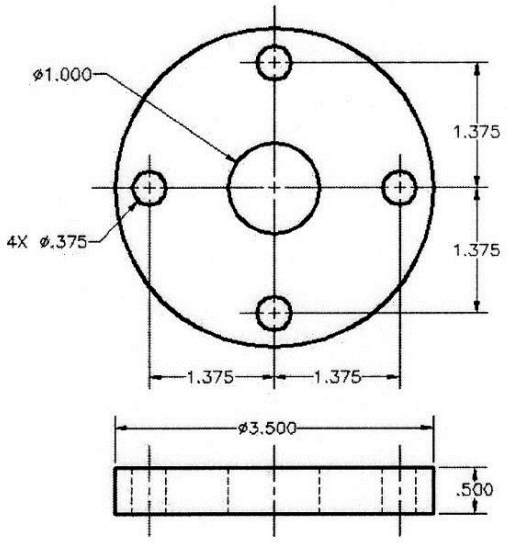


4. Spheres



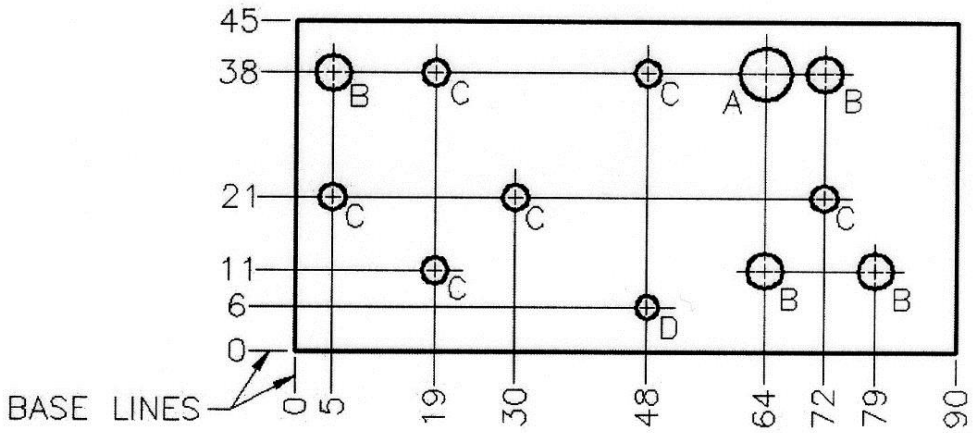
B. **Rectangular Coordinate Dimensioning** – This type of dimensioning system is specifically used when computer-controlled production machines are used to manufacture parts. Notice in the examples that all dimensions are referenced from an origin or 0,0 location. The designer should consult with personnel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

1. Coordinate Dimensioning with Dimension Lines

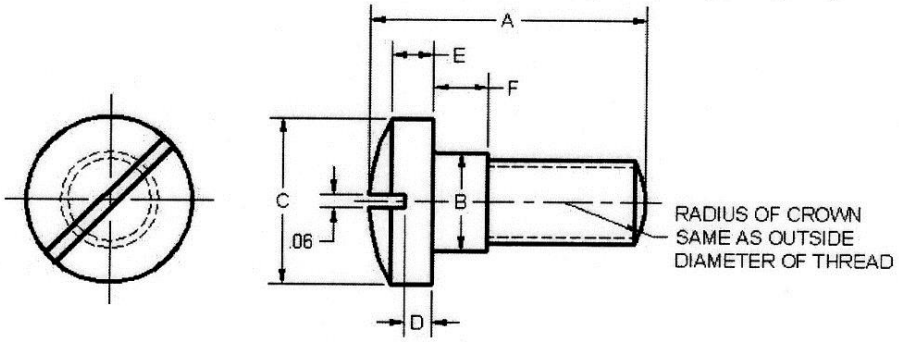


2. Coordinate Dimensioning Without Dimension Lines.

SIZE SYMBOL	A	B	C	D
HOLE ø	7	4.8	3.6	3.1



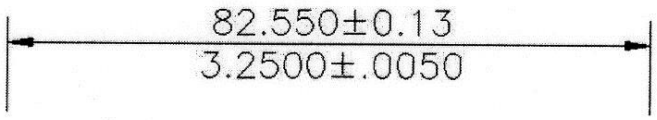
C. **Tabular Dimensioning**. Tabular dimensioning is used when a series of parts consists of the same features or geometry but vary in dimension. Letters are used in place of dimension values, and the values are then placed in a table. Most standard parts are dimensioned this way in catalogs, the machinery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

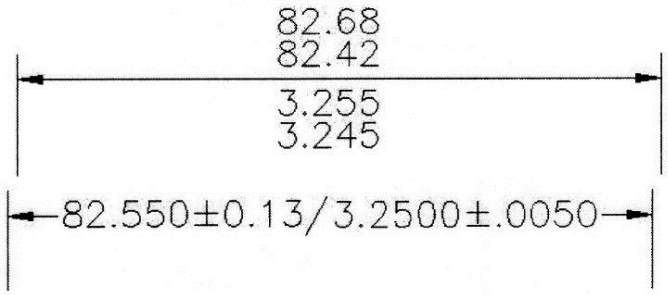


DETAIL	A	B	C	D	E	F	UNC THREAD
1	1.000	.380	.625	.091	.188	.250	.312 - 18
2	1.250	.440	.750	.125	.188	.250	.375 - 16
3	1.500	.500	.875	.125	.250	.375	.437 - 14

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.**