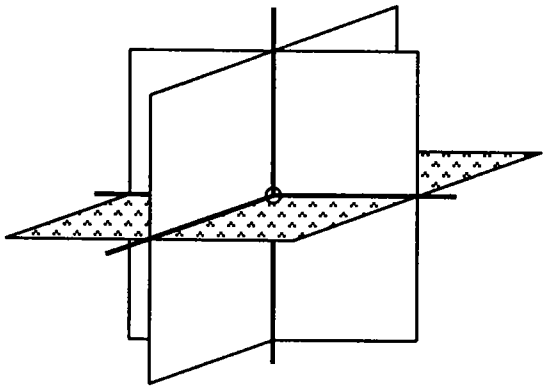
WORKSHOP EXERCISES 5.1

1. The datum reference frame (DRF) consists of points, lines (axes) and planes. Label these components in the illustration below.



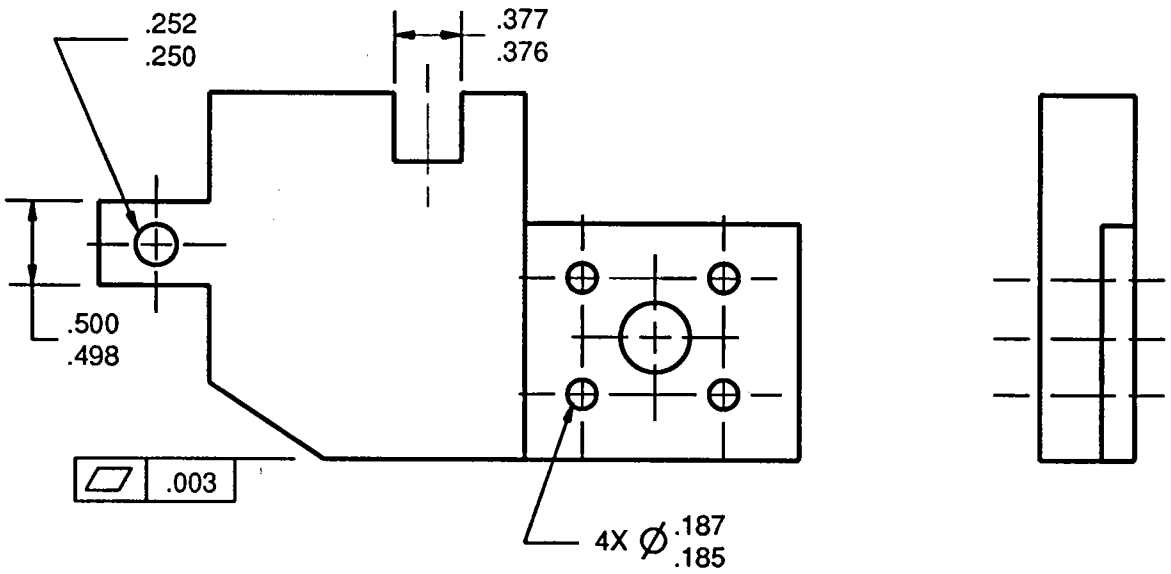
2. The datum reference frame is three mutually perpendicular planes. In engineering, it represents the surface of a drafting table, the straight edge and the 90 degree triangle or the coordinate system in CAD. What equipment does it represent in manufacturing?

3. What equipment does the datum reference frame represent in inspection?

4. What does the datum reference frame represent in mathematics? Hint: It is named after a French mathematician.

5. The two symbols below are called _____

WORKSHOP EXERCISE 5.1



On the drawing above, apply datum feature symbols as directed below.

13. Establish the right hand face in the side view as datum feature A.

14. Establish the bottom surface in the plan view as datum B.

15. Establish the top of the tab as datum feature C.

16. Establish the tab as datum feature D. (Median plane of the feature)

17. Establish the bottom surface of the slot as datum feature E.

18. Establish the slot as datum feature F. (Median plane of the feature)

19. Establish a partial datum G on the right hand surface of the part in the plan view. The length is 1.750 from the bottom, up.

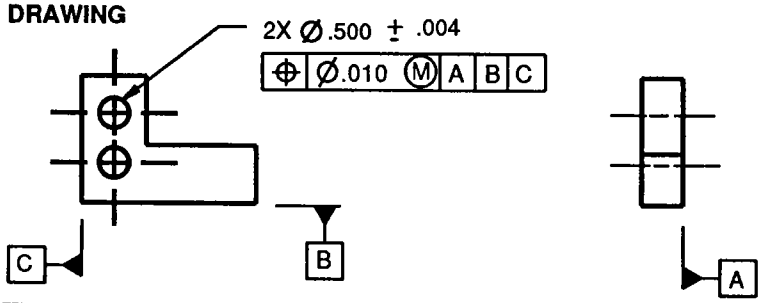
20. Establish the 4 hole pattern as datum feature H. (Axis and orientation planes of the features.

21. Identify the .250 diameter hole as datum feature J. (Axis of the feature)

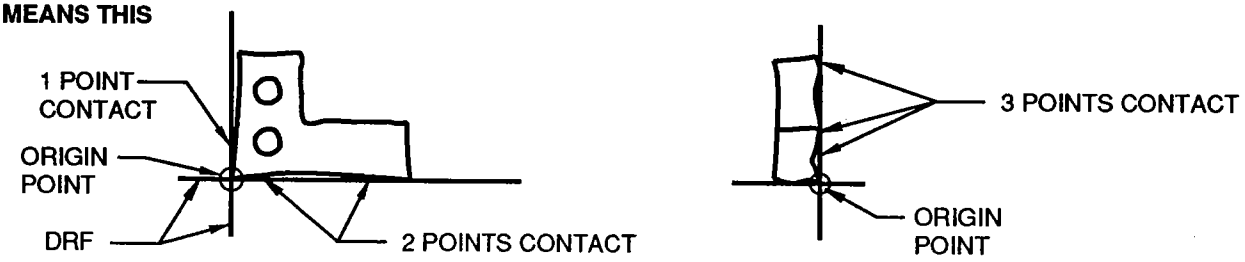
22. Datum features can be placed in two categories - datum features that have size and datum features that have no size. Place an "X" next to all the datums that have size.

WORKSHOP EXERCISE 5.1

THIS ON THE DRAWING



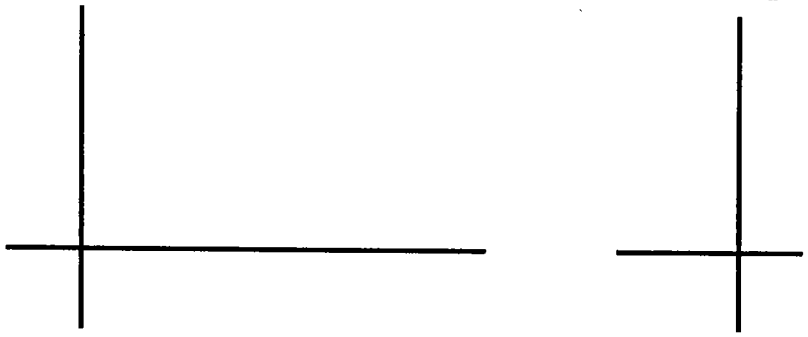
MEANS THIS



In the figure above, an imperfect part is shown loaded on the datum reference frame according to the order designated in the feature control frame. In the following examples, sketch an imperfect part loaded on the datum reference frame according to the applicable feature control frame. Also, specify the points of contact on each plane and identify the origin point.

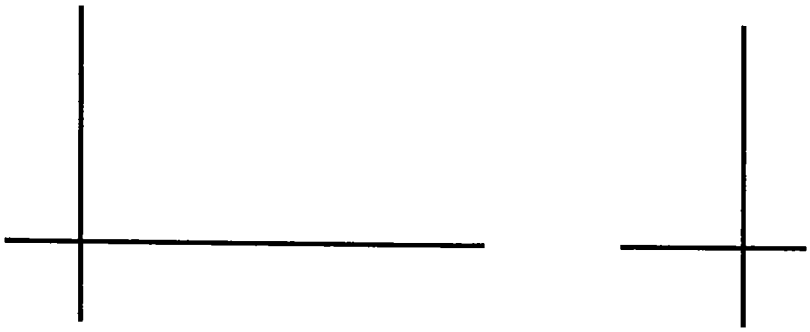
23. This order in the feature control frame. → 2X Ø.500 ± .004 | Ø .010 M | A | C | B |

MEANS THIS



24. This order in the feature control frame. → 2X Ø.500 ± .004 | Ø .010 M | B | C | A |

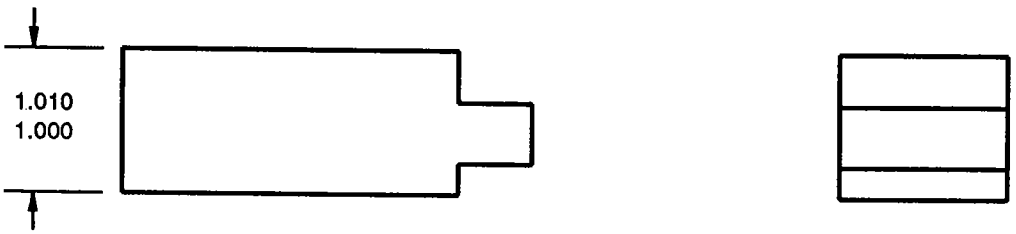
MEANS THIS



WORKSHOP EXERCISE 11.1

1. Name the three orientation tolerances.

2. Are datums always required with orientation tolerances?



3. On the part above, what is the maximum variation in parallelism before any geometric tolerancing is applied?

4. On the part above, specify a parallelism tolerance to make the top surface parallel to the bottom surface within a total of .006. Be sure to properly identify the datum feature.

5. Can the MMC modifier be applied to the feature tolerance or the datum feature? Why or why not?

6. What can we expect the flatness tolerance on the top surface to be?

7. On the produced part below show a simple sketch on how the part would be checked. Make sure to dimension and show the tolerance zones.

WORKSHOP EXERCISE 14.1

6. The example below illustrates all the coaxial controls. Next to each feature control frame, identify the best description that identifies the control. Select from the descriptions below. In addition, state whether the control is a 2D or 3D specification.

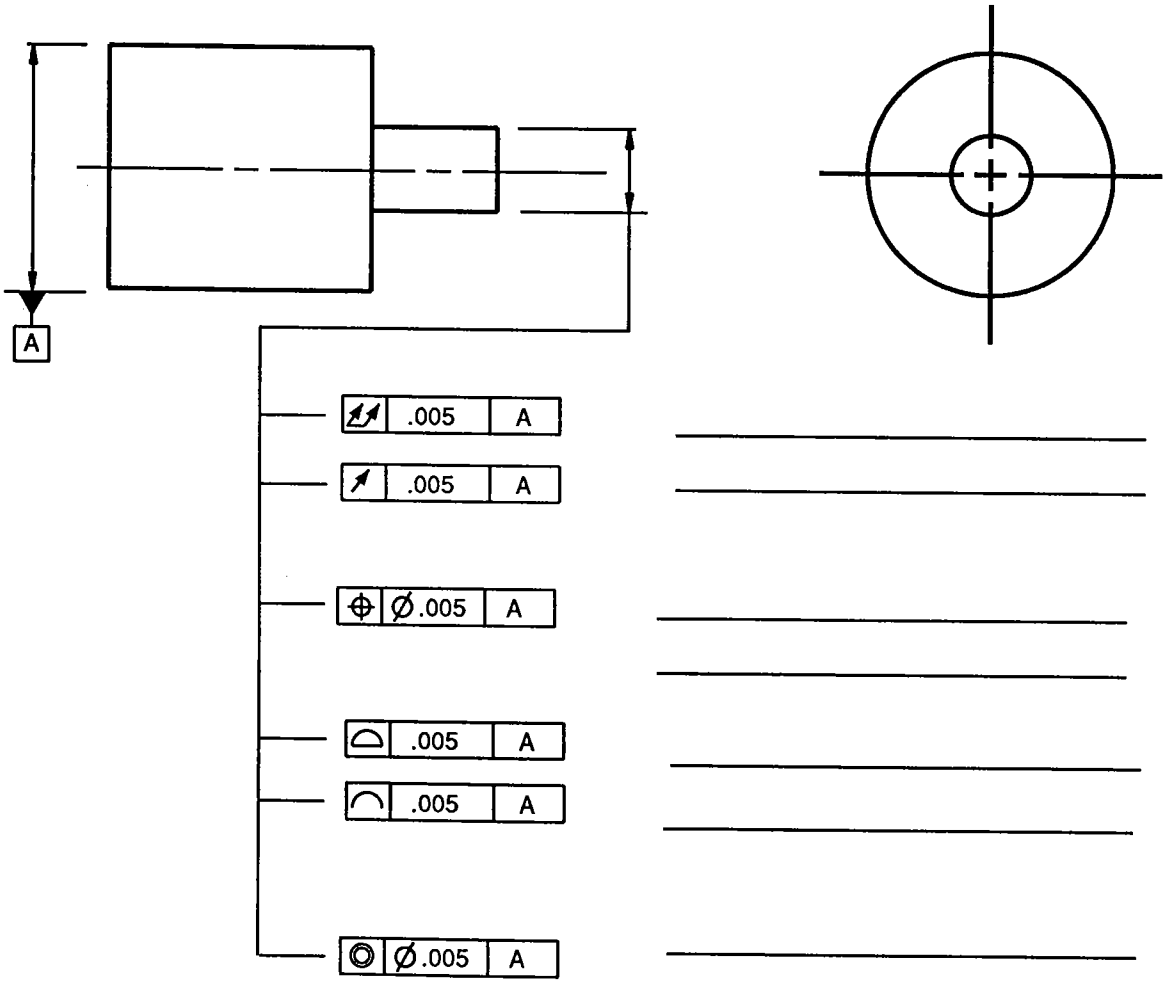
MEDIAN POINTS TO AN AXIS CONTROL

AXIS TO AXIS CONTROL

SURFACE TO AN AXIS CONTROL, INCLUDES SIZE

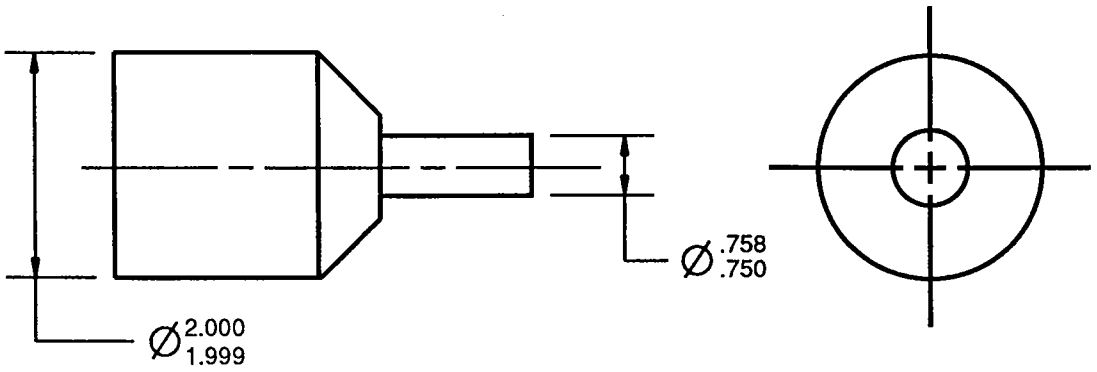
SURFACE TO AN AXIS CONTROL

COMPARISON OF COAXIAL CONTROLS



7. Only one of the controls above are allowed to have the material condition modifiers MMC or LMC applied to the feature tolerance. Which one is it?

WORKSHOP EXERCISE 14.1



9. On the part above, what coaxiality is implied between the two diameters if no geometric specification is applied?

10. On the part above, identify the large diameter as datum feature A. Specify a .003 circular runout specification to the small diameter in relation to datum A.

11. With the circular runout specification applied, how much taper on a diameter is allowed on the small diameter?

12. If the circular runout specification were replaced with a .003 total runout specification, how much taper is allowed on the small diameter?

13. With the circular runout specification applied, how much circularity, on a radius, is allowed on the small diameter?

14. With the circular runout specification applied, How much position, on a diameter, is allowed on the small diameter relative to datum A?

15. On the part above, specify a runout requirement to make the left face perpendicular to to datum axis within .005 total.

16. On the part above, specify a runout requirement to make the right face circular elements perpendicular within .001.

17. What is the technical difference between the circular runout specification applied to the left face and the total runout specification applied to the right face?