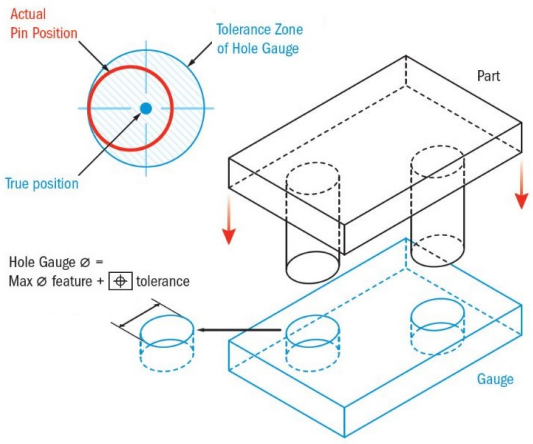
Save and Exit

Quiz Title

Align Quiz to Standard

Enable Sharing
SOC-38019251

1. In order for these two mating parts to fit together, which tolerance should you specify to show the



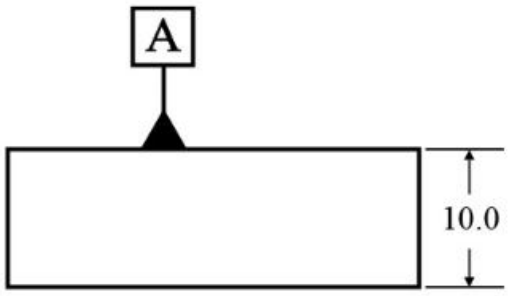
Concentricity

Circularity

Position

Cylindricity

2. The symbol placed on the top of the object is called a



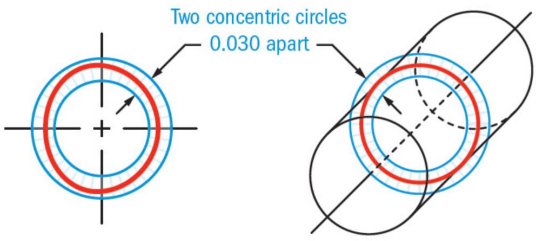
Dimension Text

Dimension Line

Datum

Dimension Feature

3. The _____ is measured by constraining a part, rotating it around the central axis while a



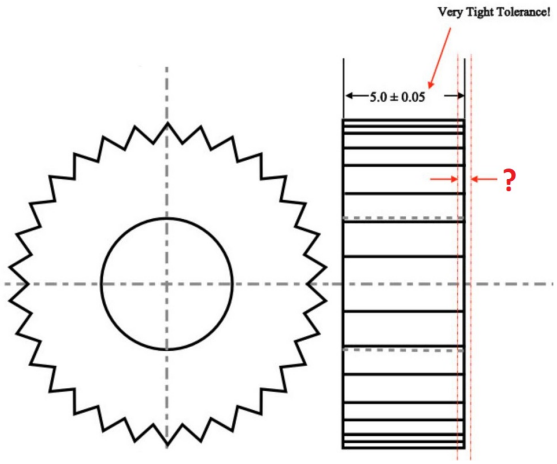
Cylindricity

Concentricity

Circularity

Position

4. _____ is a fairly common symbol that describes a parallel orientation of one referenced



Profile

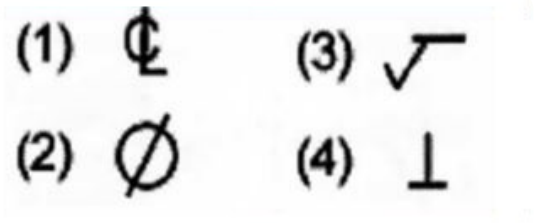
Parallelism

Circularity

Cylindricity

i A gear has to maintain constant axial load on both faces. To ensure even contact one side of the gear is held parallel to the other side, the gear width

5.



(1)

(2)

(3)

(4)

6. These elements of fully defined a technical drawing are

a. Graphics

True

7. What is the fundamental purpose of dimensions?

Aesthetics

Communication

Size

Datums

8. Aligned Dimensions have text placed parallel to the dimension line with vertical dimensions read

True

9. Dimensions should are to be kept _____ of the boundaries of views of objects wherever

radial

perpendicular

outside

inside

10. If it is necessary to include a dimension which is out of scale, the out of scale dimension text must

in bold text

enclosed in a box

underlined

In parenthesis

11. In a dimension, the symbol "X" is used to indicate

a less important feature in the design.

a feature that can be deleted.

the number of times a feature is to be repeated.

None of the above

12. ASME stands for American Society of Mechanical

True

13. ANSI stands for American National Standards

True

14. A general rule of thumb...

A designer should always use as 6 views to fully convey the geometry of the part

A designer should use as few views as possible to fully convey the geometry of the part

A designer should use as many views as possible to fully convey the geometry of the part

A designer should use as isometric views to fully convey the geometry of the part

15. GD&T standardizes the “language” of engineering drawings, so everyone could read a drawing and

True

16. 5. Which standards are considered the authoritative guideline for the design language of

ISO

ASME Y14.5

ANSI

GD&T

17. When one part is to be assembled with other

specified dimensions must never limited.

it must be manufactured to fit into place without further machining or handwork

it must be manufactured so minimal work is done with the use of machines

it must be manufactured so minimal work is done with the use hand tools

18. When specifying Limiting Dimensions:

Only the maximum and minimum dimensions are

When used with dimension lines, the Maximum limit is placed over the Minimum limit

When using with leader lines, the Minimum limit precedes the Maximum limit

All of the above

19. What is true about Reference Dimension?

It is a numerical value shown in with a "plus or minus" symbol and provided for information only

It is a numerical value shown in a box and provided for information only

It is a numerical value shown in parenthesis and provided for information only

None of the above.

20. A tolerance is

the total amount that a specific dimension is permitted

The total amount is considered the difference between the maximum and minimum limits

Both A and B

Neither A nor B

21. Designers must keep within a fixed limit of accuracy. This is accomplished by specifying

underlined to allow to vary from the absolute

in tenths, hundredths, thousandths, or ten thousandths of an inch or millimeter to allow to vary from the

shown in isometric views to allow to vary from the absolute measurement

in parentheses to allow to vary from the absolute

22. A _____ is defined as a numerical value(s) or mathematical expression in appropriate units of

Tolerance

Feature Control Frame

Datum Feature

Dimension

23. A _____ is defined as the total amount that a specific dimension is permitted to vary. This total

Feature Control Frame

Datum Feature

GD&T Symbol

Tolerance

24. Rectangular coordinate dimensioning is the same

Progressive Dimensioning

Datum Dimensioning

Baseline Dimensioning

All of the above

25. When using Datum Dimensioning...

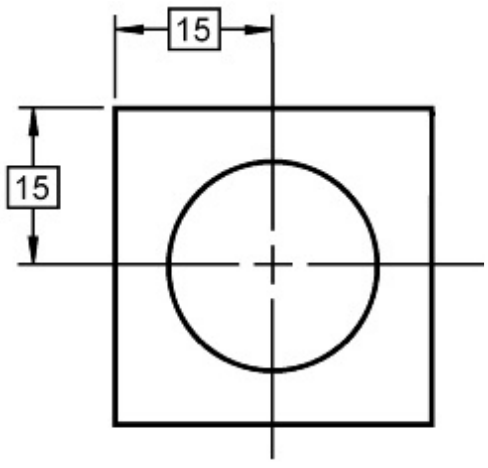
all dimensions start at the same place (origin point) and are calculated as X and Y distances

all dimensions follow the Relative Coordinates

all dimensions follow the POLAR Coordinates

None of the above

26. This dimension represents



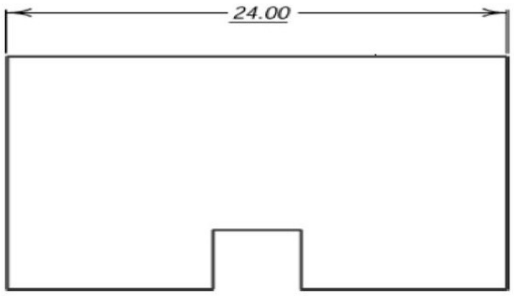
Limit dimensioning

Basic dimensioning

Reference dimensioning

Out of scale dimensioning

27. This dimension represents



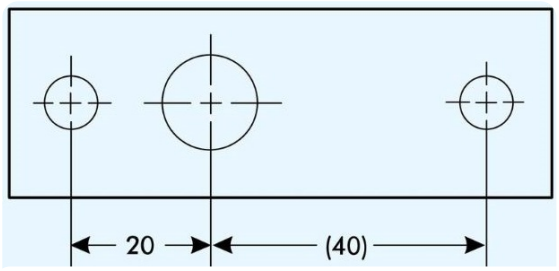
Limit dimensioning

Basic dimensioning

Reference dimensioning

Out of scale dimensioning

28. The dimension in parenthesis represents



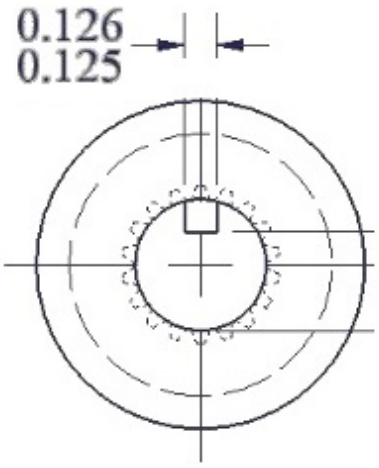
Limit dimensioning

Basic dimensioning

Reference dimensioning

Out of scale dimensioning

29. The dimension in parenthesis represents



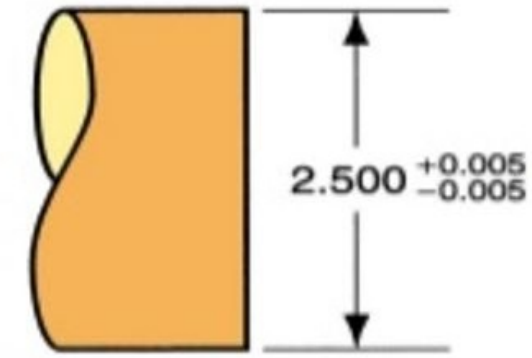
Limit dimensioning

Basic dimensioning

Reference dimensioning

Out of scale dimensioning

30. This dimension in represents



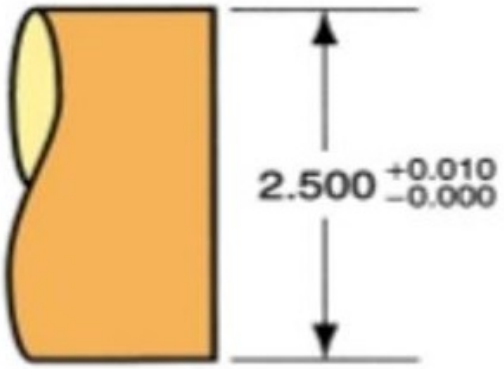
Limit dimensioning

Unilateral dimensioning

Bilateral dimensioning

Trilateral dimensioning

31. This dimension in represents



Limit dimensioning

Unilateral dimensioning

Bilateral dimensioning

Trilateral dimensioning

32. This dimension is used define the perfect location of features with respect to a datum frame

True

Add a Question

Multiple Choice **True / False**

Short Answer