

## TRUE/FALSE

1. Dimensions and tolerances are the elements that create clarity and precision on an engineering drawing.

ANS: T                      PTS: 1                      REF: The Purpose of Dimensions and Tolerances

2. In an engineering drawing, the relationships between part surfaces are defined based on how they function in the product.

ANS: T                      PTS: 1                      REF: The Purpose of Dimensions and Tolerances

3. The tolerances specified in an engineering drawing do not necessarily influence manufacturing.

ANS: F                      PTS: 1                      REF: The Purpose of Dimensions and Tolerances

4. Nominal size is the designation used to show the total amount that a specific dimension is permitted to vary.

ANS: F                      PTS: 1                      REF: Dimensions and Tolerances

5. ASME Y14.5 states, "All limits are absolute."

ANS: T                      PTS: 1                      REF: Interpreting Dimensional Limits

## COMPLETION

1. Dimensions and tolerances communicate/document functional relationships by providing a toolset of \_\_\_\_\_ and \_\_\_\_\_ that can be used to convert product requirements into drawing specifications.

ANS:  
symbols rules  
rules symbols

PTS: 1                      REF: The Purpose of Dimensions and Tolerances

2. In an engineering drawing, the amount of \_\_\_\_\_ represents the amount of variation that will not harm the functional requirements.

ANS: tolerance

PTS: 1                      REF: The Purpose of Dimensions and Tolerances

3. \_\_\_\_\_ or metric SI (millimeter) units are commonly used for linear dimensions on engineering drawings.

ANS:  
English (decimal inch)  
Decimal inch (English)

PTS: 1 REF: Units of Linear Measurement

4. A(n) \_\_\_\_\_ is a numerical value(s) or mathematical expression in appropriate units of measure used to define the size, location, orientation, or form (shape) of a feature (surface) or feature of size

ANS: dimension

PTS: 1 REF: Dimensions and Tolerances

5. A(n) \_\_\_\_\_ is where the allowable variation from the nominal value is all in one direction and zero in the other direction.

ANS: unilateral tolerance

PTS: 1 REF: Methods Used to Express Tolerance on Dimensions

## SHORT ANSWER

1. Explain three common uses of dimensions and tolerances.

ANS:

Three common uses of dimensions and tolerances are to:

1. Communicate and document functional relationships
2. Influence manufacturing choices
3. Influence inspection choices

PTS: 1 REF: The Purpose of Dimensions and Tolerances

2. Explain how dimensions and tolerances in an engineering drawing influence inspection choices.

ANS:

Dimensions and tolerances influence inspection choices in the following ways:

1. The datum system indicates how the part is to be held during inspection.
2. Basic dimensions orient and locate tolerance zones relative to the datums.
3. Geometric tolerances describe the shape and size of tolerance zones.

PTS: 1 REF: The Purpose of Dimensions and Tolerances

3. Explain a limit tolerance.

ANS:

A limit tolerance is where a dimension has its high and low limits stated. In a limit tolerance, the high value is placed on top, and the low value is placed on the bottom. When limit tolerances are expressed in a single line, the low limit is stated first, then a dash, followed by the high limit.

PTS: 1 REF: Methods Used to Express Tolerance on Dimensions

4. Define a plus-minus tolerance.

ANS:

A plus-minus tolerance is where the nominal of a dimension is given first, followed by a plus-minus expression of a tolerance.

PTS: 1

REF: Methods Used to Express Tolerance on Dimensions

5. Explain the difference between general notes, flag notes, and local notes in an engineering drawing.

ANS:

General notes are notes that apply to the entire drawing. General notes (and flag notes) are always numbered as a single-numbered list in the notes area of the drawing.

Flag notes are notes that are located with the general notes but apply only at specific areas or points on the drawing. A flag note should be identified with a flag note symbol, including the note number at each point of application. The flag note symbol is placed around the note number to indicate that it only applies at specific areas on the drawing.

Local notes are notes that are located at the specific area or point of application on the drawing. Local notes are not included in the listing of general or flag notes. A requirement specified by a local note only applies to the specific area or point indicated.

PTS: 1

REF: Engineering Drawing Notes