



---

## Geometric Dimensioning and Tolerancing Quiz

*ASME Y14.5M - 2009*

---

So you think you know Geometric Dimensioning and Tolerancing (GD&T)? Take our real-time interactive quiz. You'll receive your score immediately after completing the quiz.

This test is for the evaluation of knowledge and skills in Geometric Dimensioning & Tolerancing per the ASME Y14.5M-2009 standard.

1.) Which statement best describes what the ASME-Y14.5M-2009 covers?

- a) Dimensioning and tolerancing standards for drafting and gaging of rigid mechanical parts.
- b) A symbolic means of dimensioning and tolerancing.
- c) Dimensioning and tolerancing standard with respect to how parts are manufactured.
- d) Establishes uniform practices for stating and interpreting dimensioning, tolerancing, and related requirements for use on engineering drawings and related documents.

2.) Using GD&T:

- a) Tighter tolerances can be achieved in manufacturing.
- b) Creates a means for expressing the true design requirements for a part.
- c) More tolerance is made available to manufacturing than is possible with only plus/minus tolerancing
- d) Both b) and c)



---

## Geometric Dimensioning and Tolerancing Quiz

*ASME Y14.5M - 2009*

---

3.) How can properly implemented GD&T save money in the manufacturing process?

- a) Better definition of the design requirements, increased availability of tolerances, better communication between design, manufacturing and inspection, fewer engineering changes.
- b) It can't. GD&T is just a way to illustrate how to inspect parts.
- c) By creating extra tolerance that did not physically exist without using GD&T.
- d) By defining the datum references that must be used by manufacturing for fixturing.

4.) What is a Basic Dimension?

- a) An exact dimension with no tolerance associated with it.
- b) A dimension in a box that must be strictly adhered to by manufacturing
- c) The nominal size of a feature.
- d) A numerical value that describes a theoretically exact size, profile, orientation or location of a feature or a datum target. It is the basis from which permissible variations (tolerances) are established.

5.) What is a Feature Control Frame?

- a) A symbolic means of expressing the type of control on a feature, and defining a tolerance zone for that control.
- b) A box on the drawing that tells the manufacturer how to make the part.
- c) A box on the drawing that tells the inspector what measurements have to be made for every part.
- d) A symbol that dictates the tolerance of position for a datum target



## Geometric Dimensioning and Tolerancing Quiz

ASME Y14.5M - 2009

6.) Match the Geometric Characteristic Symbols with their respective descriptions:



|                      |                         |
|----------------------|-------------------------|
| Straightness         | a) <input type="text"/> |
| Flatness             | b) <input type="text"/> |
| Circularity          | c) <input type="text"/> |
| Cylindricity         | d) <input type="text"/> |
| Profile of a Line    | e) <input type="text"/> |
| Profile of a Surface | f) <input type="text"/> |
| Perpendicularity     | g) <input type="text"/> |
| Angularity           | h) <input type="text"/> |
| Parallelism          | i) <input type="text"/> |
| Circular Runout      | j) <input type="text"/> |
| Total Runout         | k) <input type="text"/> |
| Position             | l) <input type="text"/> |
| Concentricity        | m) <input type="text"/> |
| Symmetry             | n) <input type="text"/> |



---

## Geometric Dimensioning and Tolerancing Quiz

*ASME Y14.5M - 2009*

---

7.)What is a "Material Condition Modifier?"

- a) The maximum dimension possible according to the tolerance specified.
- b) The minimum dimension possible according to the tolerance specified.
- c) A symbol placed on a feature of size that states how much tolerance is available for that feature.
- d) A symbol (or lack thereof) placed in a feature control frame that indicates how much, or if, any extra geometric tolerance is available for the feature as the actual mating size of the feature changes.

8.)A positional tolerance defines:

- a) A zone within which the center, axis, or center plane of a feature of size is permitted to vary from a true position.
- b) A boundary located at the true position that may not be violated by the surface or surfaces of the considered feature, if the position specification is on an MMC or LMC basis.
- c) Both a and b.
- d) Neither a nor b.

9.)Which is the correct definition of regardless of feature size?

- a) Regardless of where a feature of size lies within its form tolerance, the tolerance of size must be met.
- b) Control placed on a feature of size stating that the tolerance of form or position must be met regardless of where the feature lies within its form tolerance.
- c) Regardless of where a feature of size lies within its size tolerance, the tolerance of size must be met.
- d) Control placed on a feature of size stating that the geometric tolerance must be met regardless of where the feature lies within its size tolerance.



---

## Geometric Dimensioning and Tolerancing Quiz

*ASME Y14.5M - 2009*

---

10.) What is "Rule 1" (i.e. the "Envelope Rule" or the "Perfect form at MMC Rule"), and what effect does it have, if any, on flatness? ...straightness? ...perpendicularity? ...position?

- a) Unless otherwise specified, the limits of size of an individual feature of size control the form of the feature as well as the size. The rule can control flatness and straightness, but not perpendicularity or position.
- b) Size tolerance controls form tolerance. This rule has no effect on flatness, straightness, or position.
- c) If specified by using an MMC symbol next to the tolerances in a feature control frame, then size tolerance controls form. It affects flatness, straightness, perpendicularity and position.
- d) Rule 1 states that the form of a feature must always be perfect, but it has no effect on perpendicularity.

11.) What is the difference between a Datum and a Datum Feature?

- a) A Datum and Datum Feature are synonymous.
- b) A Datum is theoretical; a Datum Feature is real.
- c) A Datum is a feature of size, and a Datum Feature is a surface feature.
- d) A Datum Feature is theoretical, and a Datum is real.

12.) What is a Datum Reference Frame?

- a) Any arbitrary coordinate system.
- b) A set of target points from which measurements are made.
- c) A symbol on the drawing that specifies the tolerance associated with a particular feature.
- d) A set of three mutually perpendicular planes derived from the specified Datum Features in the order of precedence listed in a feature control frame.



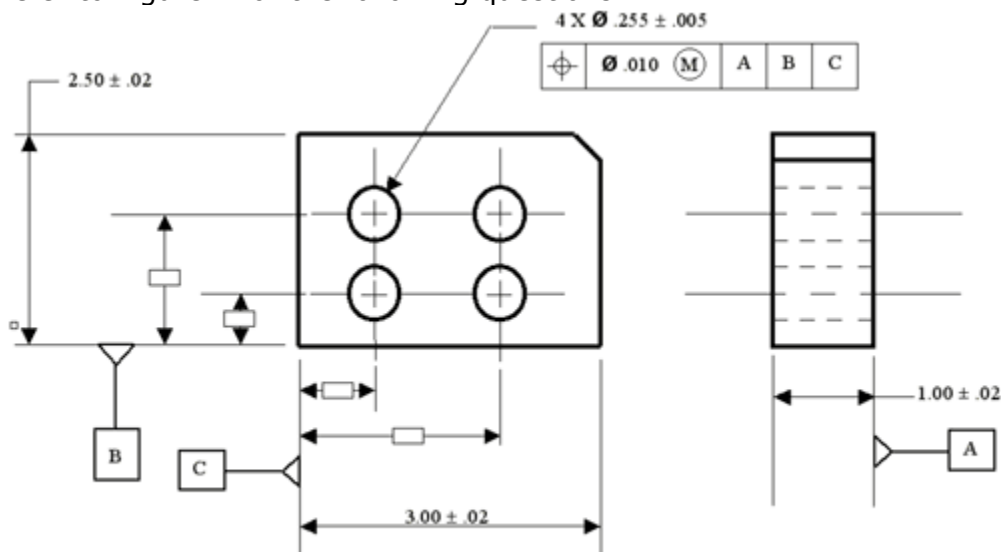
## Geometric Dimensioning and Tolerancing Quiz

ASME Y14.5M - 2009

13.) What is MMC Virtual Condition?

- a) The collective effect of the MMC limit of size of a feature of size, and any applicable geometric tolerance.
- b) The largest external feature (e.g. pin), or the smallest internal feature (e.g. hole).
- c) The perfectly formed envelope that just fits around (or inside of) a feature of size.
- d) Rule 1 states that the form of a feature must always be perfect, but it has no effect on perpendicularity.

Refer to Figure 1 for the following questions:





---

## Geometric Dimensioning and Tolerancing Quiz

*ASME Y14.5M - 2009*

---

14.) What is the MMC size of one of the holes?

- a)  $\text{Ø}.240$
- b)  $\text{Ø}.250$
- c)  $\text{Ø}.260$
- d)  $\text{Ø}.270$

15.) What is the MMC Virtual Condition size of one of the holes?

- a)  $\text{Ø}.240$
- b)  $\text{Ø}.250$
- c)  $\text{Ø}.260$
- d)  $\text{Ø}.270$

16.) What is the LMC size of one of the holes?

- a)  $\text{Ø}.240$
- b)  $\text{Ø}.250$
- c)  $\text{Ø}.260$
- d)  $\text{Ø}.270$

17.) How much total positional tolerance would be available for one hole if that hole was produced at a size of  $\text{Ø}.255$ ?

- a)  $\text{Ø}.010$
- b)  $\text{Ø}.005$
- c)  $\text{Ø}.020$
- d)  $\text{Ø}.015$



## Geometric Dimensioning and Tolerancing Quiz

ASME Y14.5M - 2009

Refer to Figure 2 for the following question:

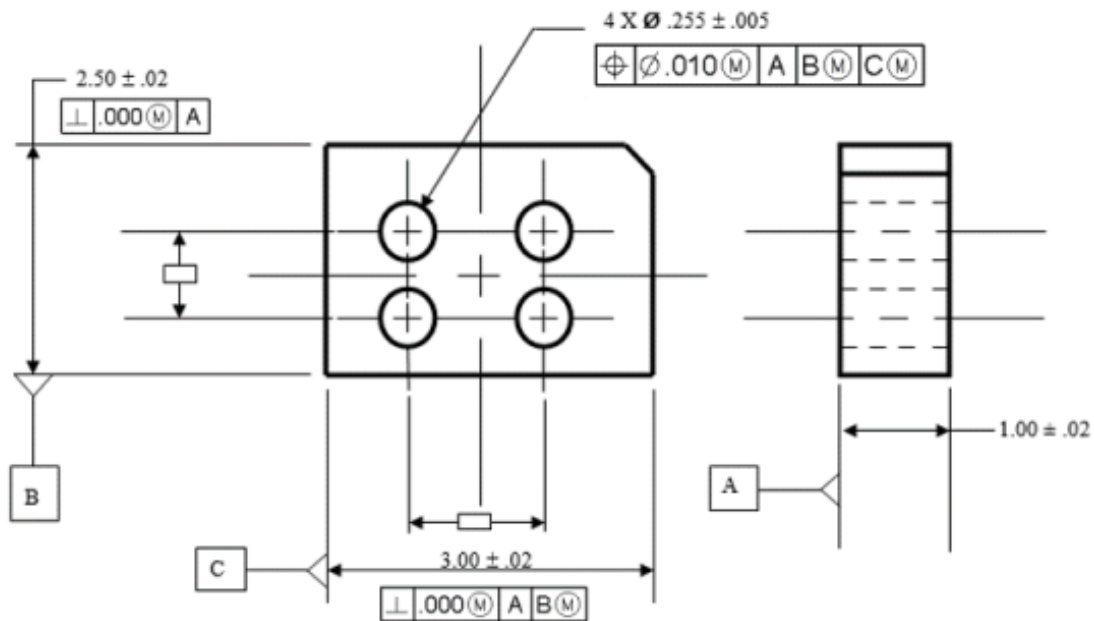


Figure 2

18.) Datum feature "B" was manufactured at a related actual mating size (RAME) of 2.49. Datum feature "C" was manufactured at an RAME of 3.01. Now how much total positional tolerance does that one hole (relative to the other holes in the pattern) have? (The considered hole is still  $\varnothing.255$ .)

- a) .010 in one direction and .030 in another direction (and a little rotation)
- b)  $\varnothing.005$
- c)  $\varnothing.020$
- d)  $\varnothing.015$





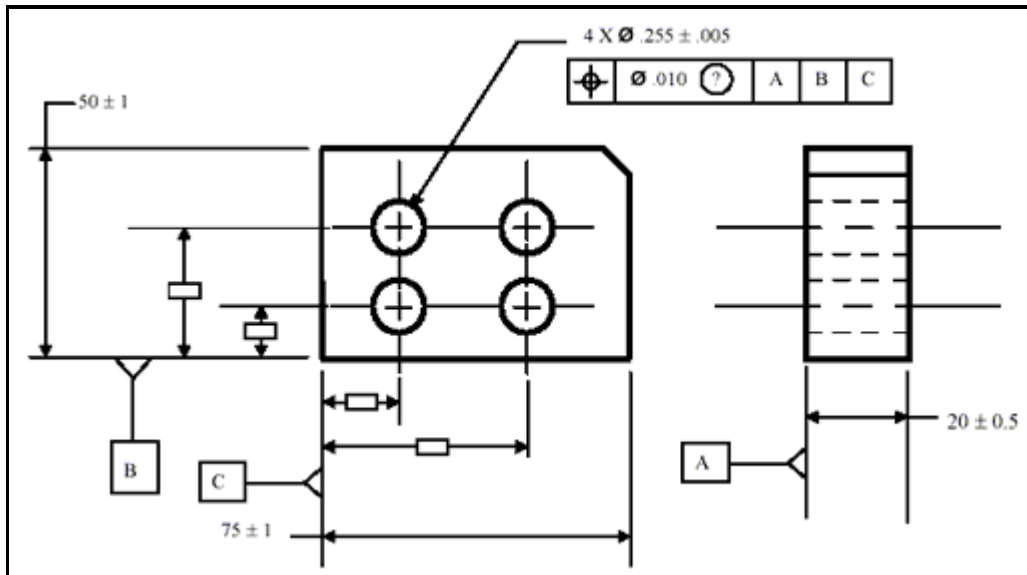
## Geometric Dimensioning and Tolerancing Quiz

*ASME Y14.5M - 2009*

19.) With the same actual datum feature sizes and considered hole size defined in the previous question, how much extra tolerance does the 4-hole pattern have relative to the "center of the part?" (Datum features "B" and "C" define the "Center" of the part.)

- a) .010 in one direction and .030 in another direction (and a little rotation)
- b)  $\varnothing.005$
- c) .040 total
- d)  $\varnothing.015$  relative to 'B' and 'C'

20.) Complete the table for the figure shown below: (all answers should be entered as ".XXX".)



| Actual Manufactured Hole Size (measured) | Total Position Tolerance              |                                       |                                       |
|--|---------------------------------------|---------------------------------------|---------------------------------------|
|  | When ? = M                            | When ? = L                            | When ? is blank (no symbol)           |
| $\varnothing.250$                        | a) $\varnothing$ <input type="text"/> | i) $\varnothing$ <input type="text"/> | q) $\varnothing$ <input type="text"/> |
| $\varnothing.251$                        | b) $\varnothing$ <input type="text"/> | j) $\varnothing$ <input type="text"/> | r) $\varnothing$ <input type="text"/> |





## Geometric Dimensioning and Tolerancing Quiz

ASME Y14.5M - 2009

|       |                           |                           |                           |
|-------|---------------------------|---------------------------|---------------------------|
| Ø.252 | c) Ø <input type="text"/> | k) Ø <input type="text"/> | s) Ø <input type="text"/> |
| Ø.253 | d) Ø <input type="text"/> | l) Ø <input type="text"/> | t) Ø <input type="text"/> |
| Ø.254 | e) Ø <input type="text"/> | m) Ø <input type="text"/> | u) Ø <input type="text"/> |
| Ø.255 | f) Ø <input type="text"/> | n) Ø <input type="text"/> | v) Ø <input type="text"/> |
| Ø.258 | g) Ø <input type="text"/> | o) Ø <input type="text"/> | w) Ø <input type="text"/> |
| Ø.260 | h) Ø <input type="text"/> | p) Ø <input type="text"/> | x) Ø <input type="text"/> |

21.) The use of zero positional tolerance at MMC:

- a) Requires the use of separate Datum Reference Frames.
- b) Results in an increase in the size tolerance for clearance holes.
- c) Allows the diameter symbol to be omitted from the FCF.
- d) Is not allowed.

22.) The use of the term, "BOUNDARY" under a position feature control frame:

- a) Should only be done in cases where extremely tight control of a surface is required.
- b) Is optional, and is a method for utilizing the tolerance afforded by the use of MMC when controlling the location and orientation of an irregular feature of size (not a simple cylinder, sphere or width feature).
- c) Controls location of a feature more tightly than the orientation of the feature.
- d) Is mandatory when it is desired to invoke the method for utilizing the tolerance afforded by the use of MMC when controlling the location and orientation of an irregular feature of size (not a simple cylinder, sphere or width feature).



## Geometric Dimensioning and Tolerancing Quiz

ASME Y14.5M - 2009

23.) A projected tolerance zone:

- a) Is required on all fixed fastener applications.
- b) Is required on assemblies where studs or press-fit pins are located on one of the mating parts.
- c) Is required on both the mating part containing the clearance hole, and on the mating part containing the fixed fastener hole.
- d) Is applicable where threaded or plain holes for studs or pins are located on detail part drawings.

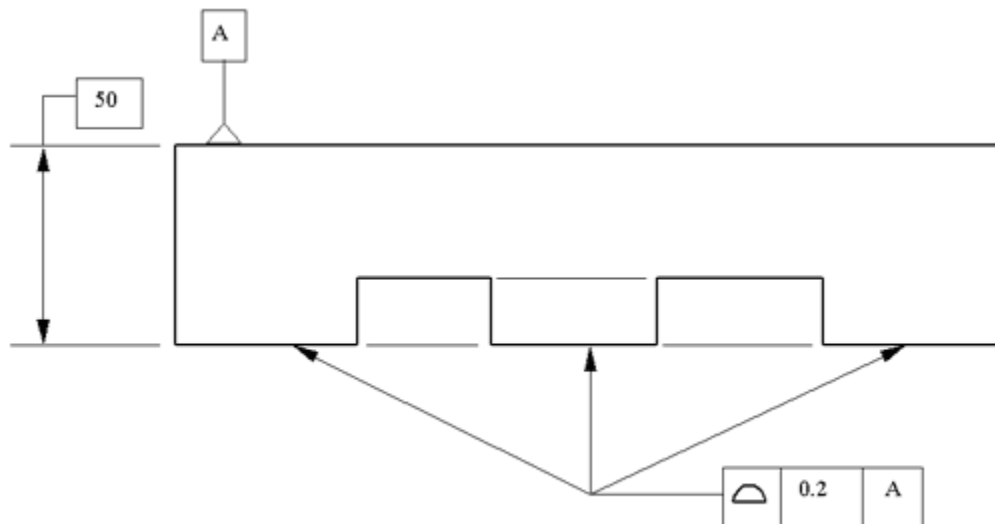


Figure 4

24.) Using Figure 4, how flat must the considered surfaces be (worst case) to meet the print requirements?

- a) 0.1
- b) 0.2
- c) 0.3
- d) 0.4



---

## Geometric Dimensioning and Tolerancing Quiz

ASME Y14.5M - 2009

---

25.) In Figure 4, if the actual of the overall height (when measuring the part against the 50mm specification) was 50.15, would that part be within the print specifications?

- a) Yes
- b) No

---

Your full name:  (optional)

Email address:  (optional)



[Change Image](#) Please enter the characters you see in the box above.

Verification:  \* [What's this?](#)

**\* - Denotes a required answer**





---

## Geometric Dimensioning and Tolerancing Quiz

*ASME Y14.5M - 2009*

---

### **SAFETY DISCLAIMER:**

M-SAMC educational resources are in no way meant to be a substitute for occupational safety and health standards. No guarantee is made to resource thoroughness, statutory or regulatory compliance, and related media may depict situations that are not in compliance with OSHA and other safety requirements. It is the responsibility of educators/employers and their students/employees, or anybody using our resources, to comply fully with all pertinent OSHA, and any other, rules and regulations in any jurisdiction in which they learn/work. M-SAMC will not be liable for any damages or other claims and demands arising out of the use of these educational resources. By using these resources, the user releases the Multi-State Advanced Manufacturing Consortium and participating educational institutions and their respective Boards, individual trustees, employees, contractors, and sub-contractors from any liability for injuries resulting from the use of the educational resources.

### **DOL DISCLAIMER:**

This product was funded by a grant awarded by the U.S. Department of Labor's Employment and Training Administration. The product was created by the grantee and does not necessarily reflect the official position of the U.S. Department of Labor. The Department of Labor makes no guarantees, warranties, or assurances of any kind, express or implied, with respect to such information, including any information on linked sites and including, but not limited to, accuracy of the information or its completeness, timeliness, usefulness, adequacy, continued availability, or ownership.

### **RELEVANCY REMINDER:**

M-SAMC resources reflect a shared understanding of grant partners at the time of development. In keeping with our industry and college partner requirements, our products are continuously improved. Updated versions of our work can be found here: <http://www.msamc.org/resources.html>.

