



**ACROSS**

©2013 Geometric Learning Systems [www.gdtseminars.com](http://www.gdtseminars.com)

- 1- A perfect hole is located at the \_\_\_\_\_ position
- 4- Another name for 15 across
- 5- This organization maintains the GD&T standard for use outside North America
- 7- A point, axis, or plane that serves as a reference for measurements
- 10- A tolerance that varies in only one direction
- 11- New acronym in 2009 standard, similar to MMC
- 13- A hole at its largest diameter is at its \_\_\_\_\_
- 15- A condition where all points of a cross-section are equidistant from the center
- 18- A functional \_\_\_\_\_ simulates a mating part
- 19- A pair of symbols that can be used to control irregular shapes
- 20- This group of symbols ensures coaxiality
- 21- Not primary or secondary
- 23- This gaging term means the overall max to min gage reading (often for runout)
- 25- These standards come from Germany
- 27- A theoretically perfect dimension
- 28- Abbreviated by R before a dimension
- 29- If a hole extends through a part, then this note may be included after the dimension
- 30- Another term for 23 across

- 31- Number of contact points usually needed for a secondary datum plane
- 32- With 37 down, this note means "measure one line at a time"
- 33- When can a surface have the MMC modifier?
- 34- Number of symbols in the form category
- 35- To determine a tolerance stack, the Monte \_\_\_\_\_ method may be used
- 36- One benefit of GD&T is that it can replace a local or general \_\_\_\_\_
- 40- This computerized machine helps gage parts
- 41- Name of the "box" that contains GD&T info
- 43- To check runout, use a \_\_\_\_\_ indicator
- 44- A hole at its smallest diameter is at \_\_\_\_\_
- 45- This symbol is not recommended, since it is difficult and expensive
- 47- Number of datum references that cylindricity requires
- 48- A shaft diameter is a feature of \_\_\_\_\_
- 50- A type of feature that is often the primary datum feature
- 52- GD&T should be based on \_\_\_\_\_ and function
- 53- An MMC symbol after the tolerance means that \_\_\_\_\_ tolerance may be available
- 54- The customary English unit of length
- 57- Straightness is part of this category
- 58- A hole is this type of diameter
- 61- Profile of a \_\_\_\_\_ is used for 2-D control
- 63- Ø
- 65- A formula used in statistical stack calculations (abbreviation)
- 67- Symbol used to keep two features at 90°
- 69- Points of contact required for 21 across
- 70- The 2009 standard now allows this symbol to be applied to a feature of size
- 71- This abbreviation was formerly used for Ø

#### DOWN

- 1- A GD&T tolerance is not plus/minus, but rather a \_\_\_\_\_ tolerance
- 2- Professional society that publishes the GD&T standard
- 3- This symbol controls two surfaces that should not be at an angle
- 6- Similar to 45 across, this symbol is also expensive to inspect
- 8- 44 across is an abbreviation for this
- 9- To specify points for datum contact, use datum \_\_\_\_\_
- 12- Holes are sometimes arranged in a \_\_\_\_\_ circle
- 14- Prior to GD&T, the \_\_\_\_\_ tol method was usually used
- 16- A variable gage must be used if a datum is referenced at \_\_\_\_\_
- 17- This symbol simultaneously checks coaxiality, straightness, and taper
- 19- This datum requires three points of contact
- 22- The combined effect of size tolerance and geometric tolerance creates this
- 24- Callouts for threaded features apply to the \_\_\_\_\_ diameter, unless specified
- 26- A datum may be a plane, axis, or \_\_\_\_\_
- 37- With 32 across, means "one line at a time"
- 38- This category relates features to each other
- 39- This is one of the most common symbols
- 40- Two tolerances that share the same symbol appear in this type of feature control frame
- 41- Symbol for all elements in the same plane
- 42- When positioning a hole, we usually think of controlling the hole's \_\_\_\_\_
- 46- Like Q, these two letters are not for datums
- 48- Another name for metric units (abbreviation)
- 49- Profile of a \_\_\_\_\_ extends in 3-D
- 51- This type of dimension is in parentheses
- 52- If a datum is referenced with the M symbol, then this type of gaging may be used
- 55- If the area being controlled extends beyond a part, a \_\_\_\_\_ tolerance zone may be used
- 56- Before 1994, this group published the standard
- 59- 22 down, for a pin, equals MMC size \_\_\_\_\_ the geometric tolerance
- 60- Number of datum references needed for flatness
- 62- 22 down, for a hole, equals MMC size \_\_\_\_\_ the geometric tolerance
- 64- Calipers & height gages are part of \_\_\_\_\_ setup
- 66- The number of degrees of freedom controlled by the primary, secondary, and tertiary datums
- 68- This statistical factor often has a target of 1.33 or greater