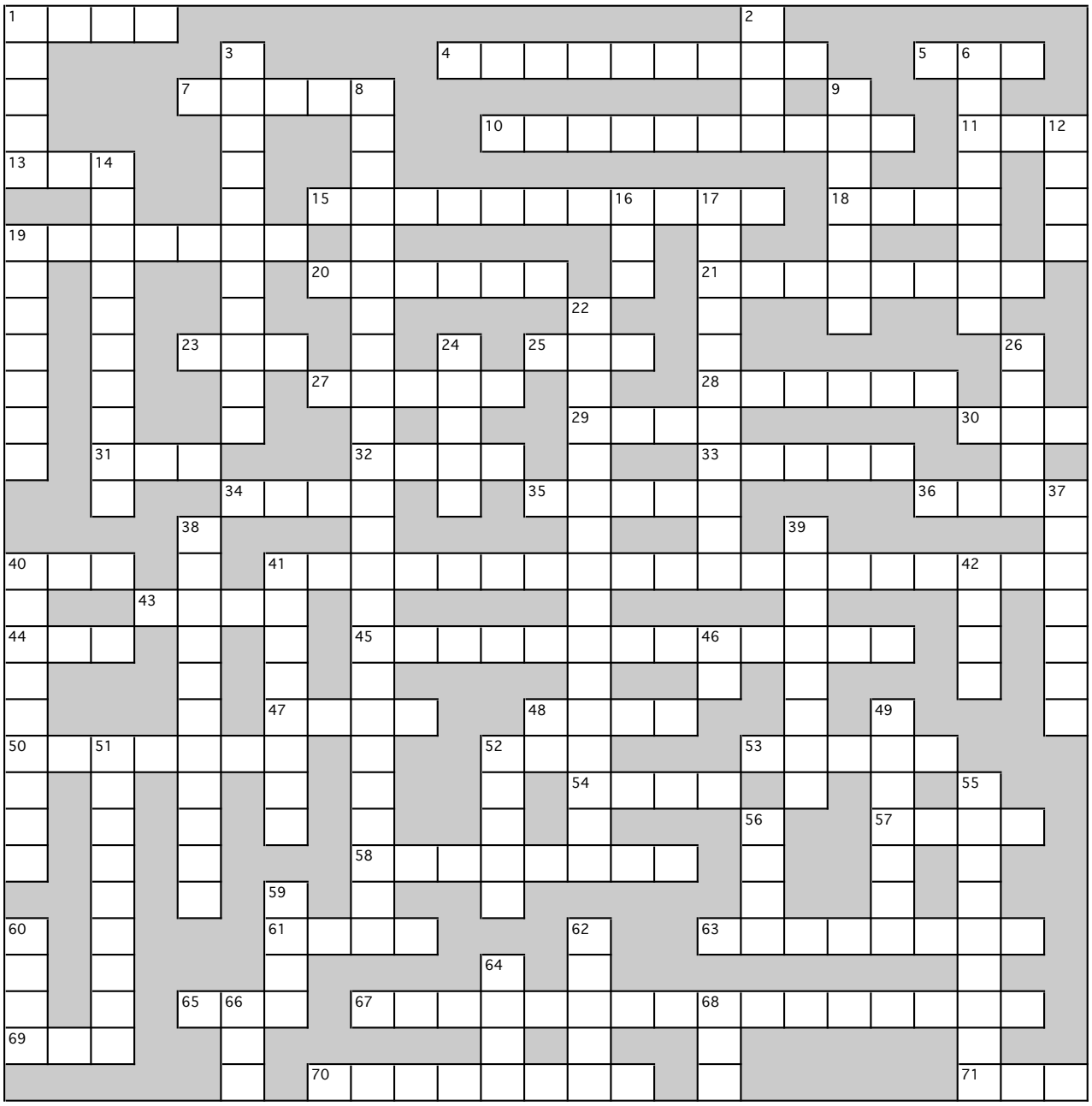

ACROSS

©2009 Geometric Learning Systems 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 are used in 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 required 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 symbol?
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 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 _____ 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 above 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

©2009 Geometric Learning Systems www.gdtseminars.com