Appendix  Example Course Assignments covering GD&T

IME 140 Graphics Communication and Modeling

Students are required to produce solid component and assembly models for the following product, and produce detailed drawings (as shown below) for two of the assembly components.

Example Weekly Lecture GD&T Quiz Questions:

1. In order to manually inspect a straightness tolerance what type of inspection tool is commonly used?
   a. Dial caliper
   b. Gage blocks
   c. Sine Bar
   d. Indicator
   e. none of these answers

2. Which tolerance does not use a datum reference in the feature control frame?
   a. Parallelism
   b. Flatness
   c. Angularity
   d. Profile of a Line
   e. Perpendicularity

3. A ________________ tolerance establishes a tolerance zone made up of two parallel planes or cylindrical zones that are 90° to a given datum plane or axis.
   a. angularity
   b. cylindricity
c. perpendicularity
   d. parallelism
   e. none of these

4. The orientation tolerances control ________________ in addition to their intended orientation control.
   a. flatness
   b. circularity
   c. cylindricity
   d. profile
   e. none of these

5. Which tolerances can be used to control combinations of size, form, and orientation?
   a. Profile Tolerances
   b. Orientation Tolerances
   c. Form Tolerances
   d. None of the Above

6. Positional tolerances create a square tolerance zone as opposed to a conventional cylindrical tolerance zone.
   a. True  b. False

7. Perpendicularity of a true position centerline is controlled by the primary datum reference in a feature control frame while the location is controlled by the secondary and tertiary datum references.
   a. True  b. False

IME 144 Introduction to Design and Manufacturing

In lab project, students set up, machine, and inspect (note fixture and gages below) the block components for an assembly:
a) When the M6 screws are inserted through the cover holes and threaded into the tapped body holes, they can be considered as 6 mm “pins” sticking up from the body.

i. Given the worst-case (i.e., MMC) clearance between these pins and the holes in the cover, specify a hole-to-hole position tolerance for the three cover holes, relative to datum A to ensure absolutely no interference between the holes and pins. Add your answer to the bottom frame in the composite tolerance for the hole positions in the simplified cover drawing. Note that a similar hole-to-hole position tolerance will be specified for the three tapped holes in the body.

ii. Now consider that the pattern of the three holes must be located with respect to the center protrusion (datum B) and the keyway (Datum C) in addition to the main datum A. Add the MMC clearance in those datum features to the hole-to-hole tolerance to get a pattern tolerance. Write your answer into the top frame in the composite position tolerance. Don’t forget to account for the possibility that Datum B (in both cover and body) is not perfectly perpendicular to Datum A. As in part i) above, assume that a similar amount will be specified for the pattern of the three tapped holes in the body.
1. When are production personnel allowed to use fixed-form (i.e., functional) locating elements (e.g., a pin, cylindrical hole, cone, slot, etc,) rather than a slower, more expensive variable locator (e.g., contracting collet, moving plates, expanding mandrel) to locate a feature during fabrication, assembly, or inspection? Select one:
   - a. Whenever traditional coordinate dimensioning is used on the drawing.
   - b. Whenever the feature is referenced using the MMC symbol in a GD&T feature control frame.
   - c. Whenever the feature is located using either a position or profile tolerance.
   - d. Production personnel can always use fixed-form locators.

2. Bonus tolerance means that the stated tolerance zone is increased in direct proportion to how much the feature is actually away from any specified modifier (usually MMC, maximum material condition). For example, if a hole were specified with a size tolerance of Ø.625 ± .010 and a position tolerance of

   \[ Ø.005 \text{ M A B C} \]

   How big would the tolerance zone for the hole axis position be if the hole were drilled with a diameter of Ø.621?

3. Match the tolerance types below to the appropriate functional consideration for which they best apply. Consider one axial (e.g., round) feature as a datum feature and another axial feature as the feature tolerated:
   - a. Concentricity  
   - b. Position  
   - c. Runout  

   _____ i   Rotating relative motion of the features; it is desired that no interference occur with other assembled components during the rotation  
   _____ ii  The features are related because two components (with the common datum) will be assembled together, and proper fit is desired between them.  
   _____ iii Both features will be rotating, and a balance of weight is desired to avoid eccentric motion/vibrations

IME 335 Computer-Aided Manufacturing I

Students create drawings, plan fixturing, machine the part, and inspect per the drawing.
IME 450 Manufacturing Process and Tool Engineering

Example homework assignments

1. Suppose the Ø1.000 ± .005 cylindrical hole feature shown below is expected to be produced and located (with reference frame below) with respect to another cylindrical hole feature (Datum B),

a) What size pin should be used to check the size of the Ø1.000 hole?
b) What size pin (with plate) should be used to check the perpendicularity of the datum hole B?
c) What size pin (with plate and locating walls) should be used to check the position of the datum B?
d) What size pin (with plate) should be used to locate the datum (B) for machining the Ø1.000 hole?
e) What else is needed in the locating fixture for part d) to ensure the proper orientation of the hole-to-hole relationship for machining the Ø1.000 hole?
f) How would the fixture have to be different if there were no MMC symbol associated with Datum B in the position callout for the Ø1.000 hole?
g) How would the fixture have to be different if there were no MMC symbol associated with the position tolerance for the Ø1.000 hole?
h) Suppose datum B is located (as in d and e), what size pin should be used to check the position of the Ø1.000 hole?
2. Consider the part drawing given below and an operation to drill the three Ø.500 holes. Note the position tolerance associated with the holes' locations. Describe all the datums that would need to be located for a fixture and sketch a simple solution (label each element).

In lab, students design and model a fixture to hold the following part during machining.

As a project, students design and fabricate the fixture, then setup and machine the following component.
IME 428 Engineering Metrology

Example homework assignment questions:

1. Consider Rule #1 for GD&T and a rectangular part specified to have length 2.0 +/- .2 inches, width 1.0 +/- .1 inches, and thickness .25 +/- .02 inches. Is the entire part required to fit within the envelope of a theoretically perfectly-formed rectangular box measuring 2.2 x 1.1 x .27? Explain.

2. What is the pitch diameter rule (Rule #3) in GD&T?

3. Which control frame is correct?
   a. 
   b. 
   c. 
   d. 

4. Describe the tolerance zone (shape, size, location, orientation) of the perpendicularity tolerance shown in the drawing.

5. What is the size (diameter) of the tolerance zone in question 4 if
   a. the center hole is produced to be 20.1 mm in diameter, all else at MMC.
   b. all dimensions are at MMC except the width of the part is 9.9 mm.

6. Examine the small figure and consider the situation where this part is to be mounted by inserting the small diameter cylindrical protrusion into a hole on a mating part. This part will then rotate. The outer (large diameter) surface will of course rotate along with the part, but the designers are concerned that if that surface is not perfect in form and location it may interfere with a mating surface during rotation. What kind of tolerance should be applied to control the large diameter surface?

7. Consider Rule #1 for GD&T, and some cylindrical part given specified tolerance Ø.250 +/- .01. Describe a gage (give either conservative toolroom tolerance for making the gage or the resolution for using a standard instrument) and a procedure for measuring each
   a. The basic “envelope rule” requirement of perfect form at MMC
   b. Any other requirement to ensure the part is within the specified size.

8. According to the GD&T standard rules and conventions, under what conditions does Rule #1 (Perfect Form at MMC) NOT APPLY to a feature of size?
9. What type of requirement goes with the symbol？

10. Suppose measurement equipment is available for inspection.
    a. What equipment is best for the angled surface profile?
    b. What equipment is best for the angled surface angularity?

11. Describe the tolerance zone (size, shape, orientation, location) for position of the 1.00 diameter hole.

12. How large is the tolerance zone in the previous question if the hole is produced to a size of 1.007-inch diameter?

13. How does angularity applied to a surface also control flatness? Use the drawing as an example to explain your answer.

14. Which reference control frame in the drawing has an error? Explain.

15. Describe the tolerance zone (size, shape, orientation, location) for the slot location.

16. Describe and sketch the tolerance zone for the elliptical cutout location.

17. Sketch in new requirements to make the hole Datum D and then to ensure the hole has no more than .002 perpendicularity error (i.e., cylindrical zone) with respect to Datum A (RFS).

18. Examine the Block part on the next page, and the large, Ø1.50 hole.
    a. What size pin could be used to check the MMC size (Rule #1) of the hole. Assign conservative tolerances for the pin.
    b. Describe a fixed-form gage that could be used to check the position of the hole. State important feature sizes and dimensions but no need to assign tolerances.
    c. Fully design a fixed-form gage to check the hole perpendicularity. Provide a fully-toleranced GD&T-style drawing for the gage.

19. Examine the four-hole pattern on the Block part.
    a. What size pin could be used to check the size of the holes?
    b. Suppose a fixed-form custom gage were to be designed to check the position of the pattern as a whole. Would the gage need a pin for locating the Ø1.50 hole?
    c. What size pins would be needed to locate the holes in the pattern for the gage in part b?
    d. Can a fixed-form gage be used to check the position of the four holes relative to each other? If so, describe the gage and give critical feature sizes or dimensions (no tolerances necessary). If not, explain how it could be checked.

20. Examine the slot feature on the Block part.
    a. What size block could be used to check the width of the slot? Give the key dimension and conservative tolerances for a gagemaker.
b. If a custom, fixed-form gage were to be created to check the position of the slot, a feature would be needed to locate Datum D. Describe this feature and how it should be sized and oriented.
c. What size block would be needed to check the slot position in the gage in part b (no tolerances necessary)?
d. Describe what the gage in part b would need to locate Datum C.

21. Explain why the 1.75 slot length dimension does not have a box around it.
22. Can a fixed-form, custom gage be used to check the perpendicularity requirement on Datum B? Explain.
23. Examine the Cover part drawing, and the runout specification.
   a. Explain how the runout should be inspected.
   b. Suppose the designer put the runout spec on the drawing because he was concerned that when this part gets assembled into its mating component, the outer 70 mm diameter might interfere with a lip on the mating part. Is Runout the proper tolerance, or should the designer have used something else? Explain.
24. What size hole should be used in a fixed-form gage to check the perpendicularity of Datum B on the Cover part?
25. Examine the pattern of three holes on the Cover part
   a. Describe a gage that could be used to check the position of the holes? Give key sizes or locations but no tolerances necessary.
   b. How could the part designer change the position requirement to make the gage in part a simpler in its design?
   c. What could be added to the drawing to control the hole-to-hole distance (i.e., unrelated to Datums B and C)?
26. Can a fixed-form gage be used to check the keyway position on the Cover part? Explain why or why not.
In Lab, students must inspect parts to the following drawings:

As a project, students must select a part from another class or project, create GD&T drawings and plan and program CMM and automated vision system inspection routines for at least 2 GD&T features each.